## Contents

1 VisIt GUI User Manual 1  
2 VisIt Python (CLI) Interface Manual 421  
3 VisIt Tutorials 667  
4 VisIt Developer Manual 767  
Index 809
1.1 Introduction to VisIt

VisIt is a free, open source, platform independent, distributed, parallel, visualization tool for visualizing data defined on two- and three-dimensional structured and unstructured meshes. VisIt’s distributed architecture allows it to leverage both the compute power of a large parallel computer and the graphics acceleration hardware of a local workstation. VisIt’s user interface is often run locally on a Windows, Linux, or OSX desktop computer while its compute engine component runs in parallel on a remote computer. VisIt’s distributed architecture allows VisIt to visualize simulation data where it was generated, eliminating the need to move the data to a visualization server. VisIt can be controlled by its Graphical User Interface (GUI), through the Python and Java programming languages, or from a custom user interface that you develop yourself. More information about VisIt can be found online at https://wci.llnl.gov/simulation/computer-codes/visit.

This manual explains how to use the VisIt GUI. You will be given a brief overview on how VisIt works and then you will be shown how to start and use VisIt.

1.1.1 Understanding how VisIt works

VisIt’s Core Abstractions

VisIt’s interface is built around five core abstractions. These include:

- Databases
- Plots
- Operators
- Expressions
- Queries
Databases

Databases read data from files and presents the data in the user interface as variables. *VisIt* supports many different types of variables including:

- Meshes
- Scalars
- Vectors
- Tensors
- Materials
- Species

Meshes are the foundation of all the other types of variables. They consist of a discretization of space into cells. All the other variables are defined on the cells of the mesh.

Scalars are single valued fields and examples include density, pressure and temperature. Vectors are multi valued fields that have a direction and magnitude. Examples include velocity and magnetic fields. Tensors are multi valued fields that are typically thought of as 2 x 2 matrices in the case of 2D data and 3 x 3 matrices in the case of 3D data. The typical tensor variable is the stress tensor. Materials are a special type of variable that associates one or more materials with a cell. The location of the material is not specified within the cell and in the case of multi material cells, algorithms must be used to determine where the material is located in the cell, typically by looking at the materials in neighboring cells. Species are variables that are associated with each material. For a given material, species are a further breakdown of a material. The distinctive property of a species is that it is uniformly distributed throughout the material. For example, air consists of many different gases such as oxygen, nitrogen, carbon monoxide, carbon dioxide, etc.

Plots

Plots take variables and generate a visual representation of the variable. Some examples include the Mesh plot, which displays the mesh lines of the mesh, the Pseudocolor plot, which maps scalar variables to color, and the Vector plot, which displays vector glyphs indicating the direction and magnitude of a vector field. Plots work on specific types of variables and the graphical user interface limits the display of variables that can be used with a given plot to the appropriate variables.

Operators

Operators take variables and modify them in some way. Operators perform their operations before they are plotted. Multiple operators may be applied to a variable forming a pipeline. For example, a mesh may be subsetted so that all the values fall within a given range, furthermore, the mesh may be subsetted to a portion of the mesh within a user specified box.

Expressions

Expressions perform calculations on variables to generate new variables. Some common expressions consist of the standard mathematical operations such as addition, subtraction, multiplication and division. It also includes more complex operations such as gradient and divergence.
Queries

Queries summarize data and typically take variables as input and generate either a single value or some small number of values. Queries can also create curves, the most common of which is the result of a query over time that creates a curve of a scalar value over time. Some examples of queries include minimum, maximum, spatial extents and volume.

VisIt's Architecture

VisIt has a client-server architecture that consists of one or more clients that connect to a viewer, which connects to one or more parallel servers. The clients and viewer typically run locally on the users desktop system while the parallel servers run on some remote high performance compute platform. This is shown in Figure 1.1. This is the most general case, but the components can also all run on a single system, either on the desktop or on a remote high performance compute platform. The server can also run in serial and for small data sets is completely sufficient.

VisIt supports a number of different clients including a Graphical User Interface (GUI), a Python based Command Line Interface (CLI), and a Java programming interface. More than one client can be active at a time and VisIt coordinates the state between them so that they are consistent.

The viewer is responsible for displaying the visual results of the plots and coordinating the state information between the various clients.

The server is responsible for reading the data from disk and performing all the manipulations on the data. The server reads and does all of its processing in parallel when running in parallel. The server can either render the data to be displayed in parallel or send the data to be rendered by the viewer. For small data sets, rendering in the viewer is faster and has less latency. For large data sets it is better to render the data in parallel (using scalable rendering) and then send the rendered image to the viewer for display. The implementation of scalable rendering is shown in Figure 1.2. VisIt is by default configured to automatically switch between shipping data to the viewer and performing scalable rendering based on the amount of geometry to be rendered.
VisIt’s Graphical User Interface

When you run the VisIt graphical user interface, you are seeing windows from the Qt based GUI and the viewer. The GUI is a VisIt client that provides the user interface and menus that let you choose what to visualize. The viewer displays all of the visualizations and is responsible for keeping track of VisIt’s state and coordinating this state with the other components. Both the GUI and the viewer are meant to run locally to take advantage of the local computer’s graphics hardware. The next two components can also be run on a client computer but they are more often run on a remote, parallel computer or cluster where the data files are generated.

The viewer supports up to 16 visualization windows. Each window is independent of the others. VisIt uses an active window concept; all changes made in the Main window or one of its popup windows apply to the currently active visualization window. The Main window and visualization window are shown in Figure 1.3.

Servers are launched on each machine where data to be visualized is located. Servers are launched on demand, typically when a database is opened. If there is more than one host profile on a system, VisIt will pop up a window asking which profile to use and additional properties such as the number of processors and nodes to use. The Host Profiles window is used to specify properties about the servers for different machines, such as the number of processors to use by default when running the server. The status of a compute engine is displayed in the Compute Engines window.

1.1.2 Installing and Starting VisIt

VisIt runs on the following platforms:

- Linux (including Ubuntu, RedHat, SUSE, TOSS)
- Mac OSX
- Microsoft Windows

A new version of VisIt is usually released every 2-3 months, you can find VisIt release executables at: https://wci.llnl.gov/simulation/computer-codes/visit/executables.

Download a binary release compatible with the machine you want to install VisIt on. If you are installing VisIt on Linux, also download the visit-install script.
1.1. Introduction to VisIt

Fig. 1.3: VisIt’s graphical user interface
Installing on Mac OSX

VisIt releases include an app-bundle for Mac OSX packaged in a DMG image. Download and open the DMG file and copy the VisIt app-bundle to your applications directory or any other path. To run VisIt double click on the VisIt app-bundle. The visit-install script can also be used to install tarball packaged OSX binaries. For this case follow the Linux installation instructions.

Installing on Linux

Installing VisIt on Linux (and optionally on Mac OSX) is done using the visit-install script. Make sure that the visit-install script is executable by entering the following command at the command line prompt:

```
chmod +x visit-install
```

The visit-install script has the following usage:

```
visit-install version platform directory
```

The version argument is the version of VisIt being installed. The platform argument depends on the type platform VisIt is being installed for. The platform argument can be one of the following: linux, linux-x86_64, darwin. The directory argument specifies the directory to install VisIt into. If the specified directory does not exist then VisIt will create it.

For example, to install an x86_64 version of VisIt 3.0.0, use:

```
visit-install 3.0.0 linux-x86_64 /usr/local/visit
```

This command will install the 3.0.0 version of VisIt into the /usr/local/visit directory. Note that when you enter the above command, the file visit3_0_0.linux-x86_64.tar.gz must be present in the current working directory.

The visit-install script will prompt you to choose a network configuration. A network configuration is a set of VisIt preferences that provide information to enable VisIt to identify and connect to remote computers and run VisIt in client/server mode. VisIt includes network configuration files for several computing centers with VisIt users.

After running visit-install, you can launch VisIt using bin/visit. For example, if you installed to /usr/local/visit, you can run using:

```
/usr/local/visit/bin/visit
```

We also recommend adding visit to your shell’s path. For bash users this can usually be accomplished by modifying the PATH environment variable in ~/.bash_profile, and for c-shell users accomplished by modifying the path environment variable in ~/.cshrc.

The exact procedure for this varies with each shell and may be customized at each computing center, so please refer to your shell and computing center documentation.

Installing on Windows

VisIt release binaries for Windows are packaged in an executable installer. To install on Windows run the installer and follow its prompts.

The VisIt installation program adds a VisIt program group to the Windows Start menu and it adds a VisIt shortcut to the desktop. You can double-click on the desktop shortcut or use the Start menu’s VisIt program group to launch VisIt. In addition to creating shortcuts, the VisIt installation program creates file associations for .silo, .visit, and .session/.vses files so double-clicking on files with those extensions opens them with VisIt.
Startup Options

VisIt has many startup options that affect its behavior (see the *Startup Options* for complete documentation).

### 1.1.3 The Main Window

VisIt’s **Main** window, shown in Figure 1.4, contains three main areas: the file area, the plot area and the notepad area. The file area contains controls for working with sources and selecting the current time state. The plot area contains controls for creating and modifying plots and operators. The notepad area is a region where frequently used windows may be posted for quick and convenient access.

![VisIt's Main window](image)

Fig. 1.4: VisIt’s Main window
Posting a window

Each time a window posts to the notepad area, a new tab is created in the notepad and the posted window’s contents are added to the new tab. Clicking on a tab in the notebook displays a posted window so that it can be used.

![Post button](image)

Fig. 1.5: An unposted and posted window

Postable windows have a **Post** button to post the window. Clicking on the **Post** button hides the window and adds its controls to a new tab in the notepad area. Posting windows allows you to have several windows active at the same time without cluttering the screen. When a window is posted, its **Post** button turns to an **UnPost** button that, when clicked, removes the posted window from the **Notepad** area and displays the window in its own window. Figure 1.5 shows an example of a window with a **Post** button and also shows the same window when it is posted to the notepad area.

Using the main menu

**VisIt**’s **Main** menu contains seven menu options that allow you to access many of **VisIt**’s most useful features. Each menu option displays a submenu when you click it. The options in the submenus perform an action such as saving an image. Menu options that contain a name followed by ellipsis open another **VisIt** window. Some menu options have keyboard shortcuts that activate windows. The **File** menu contains options that deal with files and simulations. The **Controls** menu contains options that open **VisIt** windows that, for the most part, set the look and feel of **VisIt**’s visualization windows. The **Options** menu contains options that allow you to set the appearance of the GUI, manage host profiles, manage **VisIt** plugins, set various preferences and save **VisIt**’s settings to a configuration file. The **Windows** menu contains controls that manage visualization windows. The **PlotAtts** and **OpAtts** menus allow access for setting the attributes of all the plots and operators. The **Help** menu provides options for viewing online help, **VisIt**’s copyright agreement, and release notes which describe the major enhancements and fixes in each new version of **VisIt**. The options for each menu except for the plot and operator attribute menus are shown in Figure 1.6 and will be described in detail later in this manual.

The **Main** menu and the **Plots** and **Operators** menus are merged in the OSX version of **VisIt** because OSX applications always have all menus in the system menu along the top of the display.
1.1. Introduction to VisIt

Fig. 1.6: VisIt’s main menus
Viewing status messages

VisIt informs the user of its progress as it creates a visualization. As work is completed, status messages are displayed in the bottom of the Main window in the status bar. In addition to status messages, VisIt sometimes displays error or warning messages. These messages are displayed in the Output window, shown in Figure 1.7. To open the Output window, click the Output indicator in the lower, right hand corner of the Main window. When the Output window contains an unread message, the Output indicator changes colors from blue to red.

Fig. 1.7: The output window and output indicator

Applying settings

When using one of VisIt’s control windows, you must click the Apply button for the new settings to take effect. All control windows have an Apply button in the lower left corner of the window. By default, new settings are not applied until the Apply button is clicked because it is more efficient to make several changes and then apply them at once. VisIt has a mode called Auto apply that makes all changes in settings take place immediately. Auto apply is not enabled by default because it can cause plots to be regenerated each time settings change and for the database sizes for which VisIt is designed, auto apply may not always make sense. If you prefer to have new settings apply immediately, you can enable auto apply by clicking on the Auto apply check box in the upper, right hand corner of the Main window. If Auto apply is enabled, you do not have to click the Apply button to apply changes.

1.1.4 Getting Started

The rest of this manual details the ins and outs to using VisIt, but you can also very quickly visualize your data by opening a database and creating plots. You must first select databases to visualize. Sample data files are usually installed with VisIt in a data directory in the directory in which VisIt was installed. If you are running VisIt on the
Windows platform, you can double-click on one of the sample Silo data files to open it in VisIt or you can run VisIt and open the **File open** window from the **Main** window’s **File** menu. Using the **File open** window, navigate to the appropriate directory, highlight a file, and click the **Ok** button. If the database was successfully opened, the **Add** menu will be enabled.

Once you have opened a database, you can use it to create a plot by selecting a plot type and database variable from the **Add** menu. Once a plot is created, the **Active plot** list will show that the new plot has been added by displaying a description of the plot drawn in green text. The color green indicates that the plot is in the new state and has not been drawn yet. To draw the plot, click the **Draw** button in the middle of the **Main** window. That’s all there is to creating a plot using VisIt. For more detailed information on creating plots and performing specific actions in VisIt, refer to the other chapters in this book.

### 1.2 Working with Databases

In this chapter, we will discuss how to work with databases in VisIt. A database can be either a set of files on disk or a running simulation. You can manage both types of databases using the same VisIt windows. First we’ll learn about **Supported File Types**, then the **File Open Window** which allows you to browse the local system or a remote host to find your files. Next, we’ll learn how to open databases for visualization using the **Sources Pane**. After that we’ll learn how to control animation in the **Time Pane** before learning how to examine information about a database using the **File Information Window**.

#### 1.2.1 Supported File Types

VisIt can create visualizations from databases that are stored in many types of underlying file formats. VisIt has a database reader for each supported file format and the database reader is a plugin that reads the data from the input file and imports it into VisIt. If your data format is not listed in **File formats supported by VisIt** then you can first translate your data into a format that VisIt can read (e.g. Silo, VTK, etc.) or you can create a new database reader plugin for VisIt. For more information on developing a database reader plugin, refer to the **Getting Data Into VisIt** manual or send an e-mail inquiry to visit-users@elist.ornl.gov.

**File extensions**

VisIt uses file extensions to decide which database reader plugin should be used to open a particular file format. Each database reader plugin has a set of file extensions that are used to match a filename to it. When a file’s extension matches (case sensitive except on MS Windows) that of a certain plugin, VisIt attempts to load the file with that plugin. If the plugin cannot load the file then VisIt attempts to open the file with the next suitable plugin, before trying to open the file with the default database reader plugin. If your files do not have file extensions then VisIt will attempt to use the default database reader plugin. You can provide the **-default_format** command line option with the name of the database reader plugin to use if you want to specify which reader VisIt should use when first trying to open a file. For example, if you want to load a PDB/Flash file, which usually has no file extension, you could provide: **-default_format** PDB on the command line.
Example Data Files

As part of VisIt's regular testing, a number of example data files VisIt reads can be found in VisIt's data subdirectory of the main code repository. In particular, if you are looking for examples of various of the human readable ASCII formats VisIt reads so that you can produce a compatible file, you may find examples there that help.

More Details of ASCII Formats

Here we describe more details specific to some of the ASCII formats VisIt reads.

Creating .visit Files

To create a .visit file, simply make a new text file that contains the names of the files that you want to visualize and save the file with a .visit extension.

- Visit will take the first entry in the .visit file and attempt to determine the appropriate plugin to read the file.
- Not all plugins can be used with .visit files. In general, MD or MT formats sometimes do not work.
  - An MT file is a file format that provides multiple time steps in a single file. Thus, grouping multiple MT files to produce a time series may not be supported.
  - An MD file is one that provides multiple domains in a single file. Thus, grouping multiple MD files to produce a view of the whole may not be supported.

Here is an example .visit file that groups time steps together. These files should contain 1 time step per file.

```
timestep0.silo
timestep1.silo
timestep2.silo
timestep3.silo
...`

Here is an example .visit file that groups various smaller domain files into a whole dataset that VisIt can visualize. Note the use of the !NBLOCKS directive and how it designates the number of files in a time step that constitute the whole domain. The !NBLOCKS directive must be on the first line of the file. In this example, we have 2 time steps each composed of 4 domain files.

```
!NBLOCKS 4
timestep0_domain0.silo
timestep0_domain1.silo
timestep0_domain2.silo
timestep0_domain3.silo
timestep1_domain0.silo
timestep1_domain1.silo
timestep1_domain2.silo
timestep1_domain3.silo
...`

You may also explicitly indicate the time associated with a file (or group of block files) using the !TIME directive like so...

```
!NBLOCKS 4
!TIME 1.01
timestep0_domain0.silo
timestep0_domain1.silo
...`

(continues on next page)
timestep0_domain2.silo
timestep0_domain3.silo
!TIME 2.02
timestep1_domain0.silo
timestep1_domain1.silo
timestep1_domain2.silo
timestep1_domain3.silo
...

Point3D Files

Point3D files are four or fewer columns of ASCII values with some header text to indicate the variable names associated with each column and a coordflag entry to indicate how to interpret the columns of data as coordinates. Point3D files can be used to define discrete points in 1, 2 and 3 dimensions having a single scalar value associated with each point. Some examples are below. The Point3D file...

<table>
<thead>
<tr>
<th>x</th>
<th>y</th>
<th>z</th>
<th>value</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>0</td>
<td>0</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>0</td>
<td>1</td>
<td>0</td>
<td>2</td>
</tr>
<tr>
<td>0</td>
<td>1</td>
<td>1</td>
<td>3</td>
</tr>
<tr>
<td>1</td>
<td>0</td>
<td>0</td>
<td>4</td>
</tr>
<tr>
<td>1</td>
<td>0</td>
<td>1</td>
<td>5</td>
</tr>
<tr>
<td>1</td>
<td>1</td>
<td>0</td>
<td>6</td>
</tr>
<tr>
<td>1</td>
<td>1</td>
<td>1</td>
<td>7</td>
</tr>
</tbody>
</table>

Defines a collection of 8 points in 3 dimensions have a scalar variable named value. Below, the #coordflag directive is used to define the same collection of 8 points in 3 dimensions as the previous example except where the columns holding the z-coordinate and the scalar variable are interleaved.

<table>
<thead>
<tr>
<th>x</th>
<th>y</th>
<th>value</th>
<th>z</th>
</tr>
</thead>
<tbody>
<tr>
<td>#coordflag xyvz</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>0</td>
<td>0</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>0</td>
<td>1</td>
<td>2</td>
<td>0</td>
</tr>
<tr>
<td>0</td>
<td>1</td>
<td>3</td>
<td>1</td>
</tr>
<tr>
<td>1</td>
<td>0</td>
<td>4</td>
<td>0</td>
</tr>
<tr>
<td>1</td>
<td>0</td>
<td>5</td>
<td>1</td>
</tr>
<tr>
<td>1</td>
<td>1</td>
<td>6</td>
<td>0</td>
</tr>
<tr>
<td>1</td>
<td>1</td>
<td>7</td>
<td>1</td>
</tr>
</tbody>
</table>

In the example below, the #coordflag directive is used to define a collection of points in two dimensions where each point has a velocity magnitude value associated with it.

<table>
<thead>
<tr>
<th>x</th>
<th>y</th>
<th>velocity</th>
</tr>
</thead>
<tbody>
<tr>
<td>#coordflag xyv</td>
<td></td>
<td></td>
</tr>
<tr>
<td>0</td>
<td>0</td>
<td>1</td>
</tr>
<tr>
<td>0</td>
<td>1</td>
<td>1.01</td>
</tr>
<tr>
<td>1</td>
<td>0</td>
<td>2.02</td>
</tr>
</tbody>
</table>

Likewise, for a collection of points in just one dimension, we would have

<table>
<thead>
<tr>
<th>x</th>
<th>y</th>
<th>velocity</th>
</tr>
</thead>
<tbody>
<tr>
<td>#coordflag xv</td>
<td></td>
<td></td>
</tr>
<tr>
<td>0</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>0</td>
<td>1</td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>0</td>
<td></td>
</tr>
</tbody>
</table>
There are some additional examples of Point3D files on the VisIt wiki pages.

### 1.2.2 File Open Window

The File Open Window allows you to select files and simulations by browsing file system either on your local computer or the remote computer of your choice. You can open the File Open Window by choosing the Open option from the Sources section of the main GUI panel (shown in Figure 1.9), or by Choosing the Open File option from the File dropdown menu. When the window opens, its current directory is set to the current working directory or a directory from VisIt’s preferences. See Figure 1.10.

![VisIt GUI panel showing Sources section](image_url)

Fig. 1.9: Main gui panel showing Sources section
1.2. Working with Databases

Fig. 1.10: File Open Window
Changing hosts

One of VisIt’s strengths is its ability to operate on files that exist on remote computers. The default host is: “localhost”, which is a name understood by the system to be the name of your local computer. To access the files on a remote computer, you must provide the name of the remote computer in the Host text field by either typing the name of a remote computer and pressing the Enter key or by selecting a remote computer from the list of recently visited hosts. To access the list of recently visited hosts, click on the down-arrow at the far right of the Host text field.

Changing the host will cause VisIt to launch a database server on the specified computer so you can access files there. Note that if you do not have an account on the remote computer, or if VisIt is not installed there, you will not be able to access files. Also note that VisIt may prompt you for a password to authenticate your access to the remote computer. To set up password-less access to remote computers, refer to Setting Up Password-less SSH.

Once a database server is running on the remote computer, its file system appears in the directory and file lists. The host name for each computer you access is added to the list of recently visited computers so that you may switch easily to computers you have recently accessed. If you installed VisIt with the provided network configurations then the list of recently visited computers also contains the hosts from the host profiles, which are covered later in this document.

Changing directories

To select data files, you must often change the active directory. This can be done in two ways. The first way is to enter the entire directory path into the Path text field and press Enter. You can use UNIX shell symbols, like the “~” for your home directory, or the “./” to go up one directory from your current directory. The directory conventions used depend on the type of computer being accessed. A MS Windows computer expects directories to be specified with a disk drive and a path with back slashes (e.g. C:\temp\data) while a UNIX computer expects directories with forward slashes (e.g. /usr/local/data). Keep the type of computer in mind when entering a path. After a path has been typed into the Path text field, VisIt will attempt to change directories using the specified path. If VisIt cannot change to the specified directory, the Output Window will appear with an error message and the Path text field will revert to the last accepted value. Another way to change directories is to double click the mouse on any of the entries in the directory list. Note that as you change directories, the contents of the File list change to reflect the files in the current directory. You can immediately return to any recently visited directory by selecting a directory from the Path text field’s pull-down menu.

Default directory

By default, VisIt looks for files in the current directory. This is often useful in a UNIX environment where VisIt is launched from a command line shell in a directory where database files are likely to be located. When VisIt is set to look for files in the current directory, the Use “current working directory” by default check box is set. If all of your databases are located in a central directory that rarely changes, it is worthwhile to uncheck the check box, change directories to your data directory, and save settings so the next time VisIt runs, it will look for files in your data directory.

Changing filters

A filter is a pattern that is applied to the files in the File list to determine whether or not they should show up in the list. This mechanism allows the user to exclude many files from the list based on a naming convention, which is useful since VisIt’s data files often share some part of their names.

The Filter text field controls the filter used to display files in the file list. Changing the filter will often change the File list as files are shown or hidden. The Filter text field accepts standard UNIX C-Shell pattern matching, where, for example, a “*” matches filter (“*”) shows all files in the File list. Note that you can specify more than one filter provided you separate them with a space.
Virtual databases

A virtual database is a time-varying database that VisIt artificially creates out of smaller, single time step databases that have related filenames. Virtual databases allow you to access time-varying data without having to first create a .visit file. The files that are grouped into a virtual database are determined by the file filter. That is, only files that match the file filter are considered for grouping into virtual databases. You can change the definition of a virtual database by changing the file filter. A virtual database appears in the file list as a set of filenames that are grouped under a single filename that contains the “*” wildcard character. (Figure 1.11) When you click on any of the filenames in the virtual database, the entire database is selected.

You can tell VisIt to not automatically create virtual databases by selecting the Off option in the File grouping pull-down menu. When automatic file grouping is turned off, no files are grouped into virtual databases and groups of files that make up a time-varying database will not be recognized as such without a .visit file. See Figure 1.12.

![File open dialog box](image)

Fig. 1.11: File grouping turned on (Smart setting)

VisIt has two levels of automatic file grouping. The default level is Smart file grouping, which enables automatic file grouping but has extra rules that prevent certain groups of files from being grouped into virtual databases. If you find that Smart file grouping does not provide the virtual databases that you expect, you can back the file grouping mode down to On or turn it off entirely.
Fig. 1.12: File grouping turned off
Refreshing the file list

Scientific simulations often write out new data files as they run. The **Refresh** button makes VisIt re-read the current directory to pick up any new files added by a running simulation. If the active source is a virtual database whose definition was changed by refreshing the file list, then VisIt will close and reopen the active source so information about new time states is made available.

Clearing out recently visited paths

The **File Open Window** maintains a list of all of the paths that have ever been visited and adds those paths to the recently visited paths list, which can be accessed by clicking on the down-arrow at the far right of the **Paths** text field. When you click on a path in the recently visited paths list, VisIt sets the database server’s path to the selected path retrieves the list of files in that directory. If you visit many paths, the list of recently visited paths can become quite long. Click the **File Open Window’s Remove Paths** button to activate the **Remove Recent Paths** window. The **Remove Recent Paths** window allows you to select paths from the recently visited paths list and remove them from the list. The **Remove Recent Paths** window is shown in Figure 1.13.

![Remove recent paths window](image)

Fig. 1.13: Remove recent paths window

Connecting to a running simulation

Computer simulations often take weeks or months to complete and it is often necessary to visualize data from the simulation before it has completed in order to diagnose potential problems. VisIt comes with a simulation interface library that can be linked into your serial or parallel simulation application in order to provide hooks so VisIt can plot data from your running simulation. When instrumented with the VisIt simulation interface library, your simulation can periodically check for incoming VisIt connections. When VisIt successfully connects to your simulation, all of your simulation variables are available for plotting without having to write plot files to disk. During the time that VisIt is connected, your simulation acts as a VisIt compute engine in addition to its regular responsibilities. You can pause the simulation while using VisIt to interact with the data or you can choose to have the simulation continue and push new...
data to VisIt for plotting. For more information about instrumenting your simulation code with the VisIt simulation library interface, see the Getting Data Into VisIt manual.

VisIt currently treats simulations as though they were ordinary files. When the VisIt simulation interface library is enabled in your application, it writes a special file with a .sim2 extension to the .visit/simulations directory in your home directory (%Documents%\VisIt\simulations on Windows). Each .sim2 file encodes the time and date it was created into the file name so you can distinguish between multiple simulations that VisIt can potentially open. A .sim2 file contains information that VisIt needs in order to connect via sockets to your simulation. If you want to connect to a simulation, you must select the .sim2 files corresponding to the simulations to which you want to connect. (Figure 1.14). Once that is done, connecting to a simulation is the same as opening any other disk file.

![File Open Window](image)

**Fig. 1.14: Accessing a simulation using the File Open Window**

### 1.2.3 Sources Pane

The **Sources pane**, near the top of the **Main Window**, displays the currently active source, and contains controls to open, close, reopen, and overlay sources. Sources are most frequently database files.

**Opening a file**

To open a file, you want to visualize, click on the **Open** button. This opens the **File Open Window**. Once a file is open, the **Close** and **Reopen** buttons become enabled.
If you have opened multiple files, the **Active source** drop-down menu allows you to switch between the files.

![Controls for setting the active source](image)

**Fig. 1.15: Controls for setting the active source**

When the **ReOpen** button is clicked, all cached information about the open database is deleted, the database is queried again for its information, and any plots that use that database are regenerated using the new information. This allows VisIt to access data that was added to the database after VisIt first opened it.

### Reopening a database

Sometimes it is useful to begin visualizing simulation data before the simulation has finished writing out data files for all time steps. When you open a database in VisIt and create plots and later want to visualize new time steps that have been generated since you first opened the database, you can reopen the database to force VisIt to get the data for the new time steps. To reopen a database, click the **ReOpen** button in the **Sources pane**. When VisIt reopens a database, it clears the geometry for all plots that used that database and cached information about the database is erased so that when VisIt reopens the database, plots are regenerated using the new data files.

### Replacing a database

If you have created a plot with one database and want to see what it looks like using data from another database, you can replace the database using the **File panel’s Replace** button. To replace a database, first select a new database by clicking on a file in the **File panel’s Selected files list** and then click the **Replace** button. This will make VisIt try to replace the databases used in the plots with the new database. If the replace operation is a success, the plots are regenerated using the new database and they are displayed in the visualization window.

### Overlaying a database

Overlaying a database is a way to duplicate every plot in the plot list using a new database. To overlay plots, select a new database from the **Active sources** dropdown, then click the **Overlay** button. This copies each plot in the **Active plot list** and replaces the database with the specified database. If the operation succeeds, the plots are generated and displayed in the visualization window. It is important to remember that each time the **Overlay** button is clicked, the number of plots in the plot list doubles.
1.2.4 Time Pane

The **Time Pane** contains controls for setting the active timestep, and VCR controls for playing animations.

**Setting the active time step**

When a time-varying database is open, the animation controls are activated so any time step in the database can be used. Note that the animation controls are only active when visualizing a time-varying database or when VisIt is in keyframe animation mode.

Time-varying databases are composed of one or more time steps which contain data to be visualized. The active time step is the time step within a time-varying database that VisIt uses to generate plots. The **Time pane** is located just below the **Sources pane** and contains controls that allow you to set the active time step used for visualization. The **Animation slider** and the **Animation text field** show the active time step. To set the active time step, you can drag the **Animation slider** and release it when you get to the desired time step, or you can type in a cycle number into the **Animation text field**. If you type in a cycle number that is not in the database, the active time step will be set to the time step with the closest cycle number to the cycle that was specified.

![Fig. 1.16: Controls for setting the active time step](image)

**Playing animations**

The **Time pane** also contains a set of **VCR buttons** that allow you to put VisIt into an animation mode that plays your visualization using all of the time steps in the database. The **VCR buttons** are only active when you have a time varying database. The leftmost VCR button moves the animation back one frame. The VCR button second from the left plays the animation in reverse. The middle VCR button stops the animation. The VCR button second from the right plays the animation. The VCR button farthest to the right advances the animation by one frame. As the animation progresses, the **Animation slider** and the **Animation Text Field** are updated to reflect the active time step.

1.2.5 File Information Window

This **File Information Window**, shown in Figure 1.17, displays information about the currently open file. The **File Information Window** is opened by choosing the **Files information** option from the **Main Window's File** menu. The window displays the names and properties of the open file's meshes, scalar variables, vector variables, and materials. The window updates each time the active file changes such as when switching between plots in the **Active plot list** or opening a new file using the controls in the **File panel**.
Fig. 1.17: File Information Window
1.3 Plots

This chapter explains the concept of a plot and goes into detail about each of VisIt’s different plot types.

1.3.1 Working with Plots

A plot is a viewable object, created from a database, that can be displayed in a visualization window. VisIt provides several standard plot types that allow you to visualize data in different ways. The standard plots perform basic visualization operations like contouring, pseudocoloring as well as more sophisticated operations like volume rendering. All of VisIt’s plots are plugins so you can add new plot types by writing your own plot plugins. See the wiki at visitusers.org for more details on creating new plot plugins or send an e-mail inquiry to visit-users@elist.ornl.gov.

Managing Plots

To visualize your data, you will iteratively create and modify many plots until you achieve the end result. Since plots may be created and deleted many times, VisIt provides controls in its Main Window to handle these functions. The Plots area, shown in Figure 1.18, contains the controls for managing plots.

![Fig. 1.18: The active plots area](image)

The most prominent feature of the Plots area, the plot list contains a list of the plots that are in the active visualization window. The entries in the plot list contain the plot name and variable. Plot list entries change colors depending on the state of the plot. When plots are initially created, their plot list entries are green indicating that they are new and have not been submitted to the compute engine for processing. When a plot is being created on the compute engine, its plot list entry is yellow. When a plot has finished generating on the compute engine, its plot list entry turns black to indicate that the plot is done. If the compute engine cannot generate a plot, the plot’s plot list entry turns red to indicate an error with the plot.

The plot list displays more than just the names of the visualization window’s plots. The plot list also allows you to set the active plots, that is, those plots that can be modified. Highlighted plot entries are active.

The Add menu, an important part of the Plots area, contains the options that create new plots.
Creating a plot

To use any of VisIt’s capabilities, you must know how to create a plot. First, make sure you have opened a database. Once you have an open database, use the Add menu to create a plot.

![Fig. 1.19: The Add menu](image)

Selecting the Add menu pops up a list of VisIt plot types. Plots for which the open database has no data are disabled. If a plot type is enabled, pulling the mouse toward the right while holding down the left button shows which variables can be plotted. Release the mouse button when the mouse cursor is over the variable that you want to plot, and a new plot list entry will appear in the plot list. The new plot list entry will be colored green in the plot list until VisIt is told to draw when you click the Draw button. The Add menu is disabled until a database is open.

Deleting a plot

VisIt deletes all the selected plots when you click the Delete button. If the plot list has keyboard focus, you can also delete a plot using the Delete key.

Selecting a plot

Since VisIt will only let you modify active plots, you must be able to select plots. To select a plot, click on its entry in the plot list. Multiple plots can be selected by holding down the Ctrl key and clicking plot entries one at a time.
Alternatively, groups of plot entries can be selected by clicking on a plot entry and then clicking another plot entry while holding down the Shift key.

**Drawing a plot**

When you add a plot to the plot list, it won’t be drawn until you click the Draw button. Once you do, the new plot’s plot list entry switches from green to yellow in the plot list to indicate that its results are pending and the compute engine starts generating the plot. Clicking the Draw button causes all new plots to be drawn.

**Hiding a plot**

When you are visualizing your data, you will often have many different plots in the same visualization window. Sometimes you might want to temporarily hide plots from view to more easily view the other plots in the window. To hide the selected plots, click the Hide/Show button in the Plots area. When a plot is hidden, its plot list entry is gray and contains the word hidden to indicate that the plot is hidden. To show a hidden plot, select the hidden plot and click the Hide/Show button again. Note that plots must exist for the Hide/Show button to be enabled.

**Setting plot attributes**

Each plot type has its own plot attributes window used to set attributes for that plot type. Plot attributes windows are activated by double-clicking a plot entry in the plot list. You can also open a plot attribute window by selecting a plot type from the PlotAtts (Plot Attributes) menu shown in Figure 1.20.

**Changing plot variables**

When examining a plot, you might want to look at another variable. For example, you might want to switch from looking at density to pressure. VisIt allows the plot variable to be changed without having to delete and recreate the plot. To change the plot variable, first make sure the plot is active, then select a new variable from the available variable names in the Variables menu. The Variables menu contains only the variables from the database that are compatible with the plot.

### 1.3.2 Standard Plot Types

VisIt comes with eighteen standard plots: Boundary, Contour, Curve, FilledBoundary, Histogram, Label, Mesh, Molecule, MultiCurve, ParallelCoordinates, Pseudocolor, Scatter, Spreadsheet, Subset, Tensor, Truecolor, Vector, and Volume. This section explains each plot in detail.

**Common Controls**

There are a number of attributes of plots that are common to many, if not all plots. These include such things as Color table, Foreground and Background colors, Opacity, Line style and Point type, Log or Linear scaling, the Legend checkbox and others. These common plot attributes are described here first using the Pseudocolor plot as an example. Then, attributes specific to each plot type are described in the remaining sections.
1.3. Plots

Fig. 1.20: The PlotAtts menu

Fig. 1.21: The Variables menu
Fig. 1.22: Example of Pseudocolor plot attribute window
Plot buttons

All plot attribute windows have several buttons at the bottom for common operations. Use the Apply after you have changed one or more attributes of a plot to make the new settings take effect. The Make default button is used to take the current settings and make those the default for the remainder of the VisIt session. Each time a new plot of that type is created, it will be created with whatever the current defaults are for that plot. If you want these settings to persist across VisIt sessions, you can either Save session, and then restart from this saved session later, or Save settings and then all VisIt sessions will use those defaults. For more about saving sessions and settings, see How to Save Settings. The Save and Load buttons give you the option of saving and loading plot attributes using their own separate XML. This allows users to easily share individual plot attributes. The reset button will return the plot’s attributes to whatever the current defaults are. The Dismiss button will dismiss the window. The Post button will place the window in the Notepad area (see Posting a window).

Plot colors

By default, VisIt uses the Hot color table which maps values at the minimum of the data range to blues, values at the maximum of the data range to reds with transitions from blue to violet, to green, to yellow in between. However, many plots offer the option of selecting a specific color table. In the picture of the Pseudocolor plot attributes window, above, the color table may be changed by selecting the currently named table. A pull-down list will appear from which you can select a different table. For more information about Color tables, see Color Tables.

In addition, many plots have options to control colors and transparency (opacity) of individual plot elements such as lines on the Mesh plot or contours on the Contour plot.

Point type and size

The Pseudocolor, Mesh and Scatter plots can use eight different point types for drawing point meshes (see Figure 1.23). The default option of Point is fastest and forces the plot to draw all of its points as tiny points. The Sphere option applies textures to the points so it is nearly as fast as Point. Any of the other options place a glyph at each point, taking longer to render. To set the point type choose an option from the Point type menu. Setting the Point type to anything other than Point will have no effect if the plotted mesh is not a point mesh.

If you choose any of the point types except Point, then you can also specify a point size by typing a new value into the Point size text field. The point size is used to determine the size of the glyph. For example, if you choose Box, and you enter a Point size of 0.1, then the length of all of the edges on the Box glyphs will be 0.1. If you use Point, then the Point size text field becomes the Point size (pixels) text field and you can set the point size in terms of pixels.

For Mesh and Pseudocolor plots, the point size can also be scaled by a scalar variable if you check the Scale point size by variable check box and select a new scalar variable from the Variable menu. The value default must be replaced with the name of another scalar variable if you want VisIt to scale the points with a variable other than the one being plotted.

Boundary and FilledBoundary Plots

The Boundary plot and FilledBoundary plot are discussed together because of their similarity. Both plots concentrate on the boundaries between materials but each plot shows the boundary in a different way. The Boundary plot, shown in Figure 1.25, displays the surface or lines that separate materials.

The FilledBoundary plot (see Figure 1.26) shows the entire set of materials, each using a different color. Both plots perform material interface reconstruction on materials that have mixed cells, resulting in the material boundaries used in the plots.
Fig. 1.23: Point types: Box, Axis, Icosahedron, Octahedron, Tetrahedron, Sphere Geometry, Point, Sphere

Fig. 1.24: Point type menu, expanded
1.3. Plots

Fig. 1.25: Boundary plot and its plot attributes window

Fig. 1.26: FilledBoundary plot and its plot attributes window
Fig. 1.27: FilledBoundary plot combined with subsets
Combining the FilledBoundary plot with subsets (see Figure 1.27) can provide a insight into where each material is inside the mesh by turning off materials in a particular domain. For more information about subsets, see the Subsetting chapter.

Changing colors

The main portion of the Boundary plot attributes window and FilledBoundary plot attributes window, also known as the Boundary colors area, is devoted to setting material boundary colors. The Boundary colors area contains a list of material names with an associated material color. Boundary plot and FilledBoundary plot colors can be assigned three different ways, the first of which uses a color table. A color table is a named palette of colors that you can customize to suite your needs. When the Boundary plot or FilledBoundary plot use a color table to color subsets, they selects colors that are evenly spaced through the color table based on the number of subsets. For example, if you have three materials and you are coloring them using the “xray” color table, three colors are picked out of the color table so your material boundaries are colored black, gray, and white. To color a Boundary plot or FilledBoundary plot with a color table, click on the Color table radio button and choose a color table from the Color table menu to right of the Color table radio button.

If you want all subsets to be the same color, click the Single radio button at the top of the Boundary plot attributes window and select a new color from the Popup color menu that is activated by clicking on the Single color button. The opacity slider next to the Single color button sets the opacity for the single color.

Clicking the Multiple radio button causes each material boundary to be a different, user-specified color. By default, multiple colors are set using the colors of the discrete color table that is active when the Boundary or FilledBoundary plot is created. To change the color for any of the materials, select one or more materials from the list of materials and click on the Color button to the right of the Multiple radio button and select a new color from the Popup color menu. To change the opacity for a material, move Multiple opacity slider to the left to make the material more transparent or move the slider to the right to make the material more opaque.

The Boundary plot attributes window contains a list of material names with an associated color. To change a material’s color, select one or more materials from the list, click the color button and select a new color from the popup color menu.

Opacity

The Boundary plot’s opacity can be changed globally as well as on a per material basis. To change material opacity, first select one or more materials in the list and move the opacity slider next to the color button. Moving the opacity slider to the left makes the selected materials more transparent and moving the slider to the right makes the selected materials more opaque. To change the entire plot’s opacity globally, use the Opacity slider near the bottom of the window.

Wireframe mode

The Boundary plot and the FilledBoundary plot can be modified so that they only display outer edges of material boundaries. This option usually leaves lines that give only the rough shape of materials and where they join other materials as seen in. To make the Boundary or FilledBoundary plots display in wireframe mode, check the Wireframe check box near the bottom of the window.

Geometry smoothing

Sometimes visualization operations such as material interface reconstruction can alter mesh surfaces so they are pointy or distorted. The Boundary plot and the FilledBoundary plot provide an optional Geometry smoothing option to
Fig. 1.28: Filled mode and wireframe mode

smooth out the mesh surfaces so they look better when the plots are visualized. Geometry smoothing is not done by default, you must click the Fast or High radio buttons to enable it. The Fast geometry smoothing setting smooths out the geometry a little while the High setting works produces smoother surfaces.

Drawing only clean zones

The FilledBoundary plot, since it deals almost exclusively with plotting materials, has an option to only draw clean zones, which are zones that contain a single material. When only clean zones are drawn, all clean cells are drawn normally but all zones that contained more than one material are drawn with a color that can be set to match the vis window’s background color (see). Drawing clean zones is primarily used to examine how materials mix in 2D databases. To make VisIt draw only the clean zones, click the Clean zones only check box. After that, you can set the mixed color by clicking on the Mixed color color button and selecting a new color from the popup color palette.

Setting point properties

Albeit rare, the Boundary and FilledBoundary plots can be used to plot points that belong to different materials. Both plots provide controls that allow you to set the representation and size of the points. You can change the points’ representation using the different Point Type radio buttons. The available options are:

- Box
- Axis
- Icosahedron
- Octahedron
- Tetrahedron
- Point
- Sphere
Fig. 1.29: All zones and clean zones

The default point type is **Point** because that is the fastest to draw, followed by **Sphere**. The other point types create additional geometry and can take longer to appear on the screen and subsequently draw. To change the size of the points when the point type is set to **Box**, **Axis**, or **Icosahedron**, you can enter a new floating point value into the **Point size** text field. When the point type is set to **Point** or **Sphere**, the **Point size** text field becomes the **Point size (pixels)** text field and you should enter your point size in terms of pixels. Finally, you can opt to scale the points’ glyphs using a scalar expression by turning on the **Scale point size by variable** check box and by selecting a scalar variable from the **Variable** button to the right of that check box. Note that point scaling does not occur when the point type is set to **Point** or **Sphere**.

Fig. 1.30: Point types (left-to-right): Box, Axis, Icosahedron, Point, Sphere

**Contour Plot**

This plot, shown in Figure 1.31, displays the location of values for scalar variables like density or pressure using lines for 2D plots and surfaces for 3D plots. In visualization terms, these plots are isosurfaces. VisIt’s Contour plot allows you to specify the number of contours to display as well as the colors and opacities of the contours.
Fig. 1.31: Example of Contour plot
VisIt User Manual Documentation, Release 3.1

Fig. 1.32: Contour plot attributes window

1.3. Plots 37
Setting the number of contours

By default, VisIt constructs 10 levels into which the data fall. These levels are linearly interpolated values between the data minimum and data maximum. However, you can set your own number of levels, specify the levels you want to see or indicate the percentages for the levels.

To choose how levels are specified, make a selection from the Select by menu. The available options are: N levels, Levels, and Percent. N levels, the default method, allows you to specify the number of levels which will be generated, with 10 being the default. Levels requires you to specify floating point numbers for the levels you want to see. Percent takes a list of percentages like 50.5, 60, and 40.0. Using the numbers just mentioned, the first contour would be placed at the value which is 50.5% of the way between the minimum and maximum data values. The next contour would be placed at the value which is 60% of the way between the minimum and maximum data values, and so forth. You specify all values for setting the number of contours by typing into the text field to the right of the Select by menu.

Setting Limits

The Contour plot attributes window provides controls that allow you to specify artificial minima and maxima for the data in the plot. This is useful when you have a small range of values that are of interest and you only want the contours to be generated through that range. To set the minimum value, click the Min check box to enable the Min text field and then type a new minimum value into the text field. To set the maximum value, click the Max check box to enable the Max text field and then type a new maximum value into the text field. Note that either the min, max or both can be specified. If neither minimum nor maximum values are specified, VisIt uses the minimum and maximum values in the database.

Scaling

The Contour plot typically creates contours through a range of values by linearly interpolating to the next value. You can also change the scale to a logarithmic function to get the list of contour values through the specified range. To change the scale, click either the Linear or Log radio buttons in the Contour plot attributes window.

Setting contour colors

The main portion of the Contour plot attributes window, also known as the Contour colors area, is devoted to setting contour colors. Contour plot colors can be assigned three different ways, the first of which uses a color table. A color table is a named palette of colors that you can customize to suite your needs. When the Contour plot uses a color table to color the levels, it selects colors that are evenly spaced through the color table based on the number of levels. For example, if you have five levels and you are coloring them using the “rainbow” color table, the Contour plot picks five colors out of the color table so your levels are colored magenta, blue, cyan, green, yellow, and red. The colors change when increasing or decreasing the number of levels when you use a color table because VisIt uses the new number of levels to sample different locations in the color table. As a rule, increasing the number of levels results in coloration that is closer to the color table because more colors from the color table are represented. To color a Contour plot with a color table, click on the Color table radio button and choose a color table from the Color table menu to right of the Color table radio button.

If you want all levels to be the same color, click the Single radio button at the top of the Contour plot attributes window and select a new color from the Popup color menu that is activated by clicking on the Single color button. The opacity slider next to the Single **color button sets the opacity for the single color.

Clicking the Multiple radio button causes each level to be a different, user-specified color. By default, multiple colors are set using the colors of the discrete color table that is active when the Contour plot is created. To change the color for any of the levels, click on the level’s Color button and select a new color from the Popup color menu. To change
the opacity for a level, move its opacity slider to the left to make the level more transparent or move the slider to the right to make the level more opaque.

**Wireframe view**

The *Contour plot attributes window* provides a *Wireframe* toggle button used to draw only the lines along the edges of the contour. This option only has an effect on 3D Contour plots.

**Curve Plot**

The Curve plot, shown in *Figure 1.33*, displays a simple group of X-Y pair data such as that output by 1D simulations or data produced by Lineouts of 2D or 3D datasets. Curve plots are useful for visualizations where it is useful to plot 1D quantities that evolve over time.

![Fig. 1.33: Curve plot](image)

**Setting curve color**

The Curve plot’s color is set up to *Cycle* by default. In other words, each new curve created will be a different color. This can be turned off by selecting the *Custom* radio button, and a new color can be chosen by clicking on the *Color button* and making a selection from the *Popup color menu*.

**Showing curve labels**

Curve plots have a label that can be displayed to help distinguish one Curve plot from other Curve plots. Curve plot labels are on by default, but if you want to turn the label off, you can uncheck the *Labels* check box.
Space-filled curves

The space below a curve can be filled with color by changing Fill mode to either Solid, Horizontal Gradient or Vertical Gradient, then choosing one or two colors based upon the mode chosen.

Setting line style and line width

Several Curve plots are often drawn in the same visualization window so it is necessary that Curve plots can be distinguished from each other. Fortunately, VisIt provides controls to change the line style and line width so that Curve plots can be told apart. Line style is a pattern used to draw the line and it is solid by default but it can also be dashed, dotted, or dash-dotted. You choose a new line style by making a selection from the Line Style combo box on the Geometry tab (see Figure 1.36). The line width, which determines the boldness of the curve, is set by making a
selection from the **Line Width** combo box.

![Fig. 1.36: Curve plot attributes, geometry tab](image)

**Drawing points on the Curve plot**

The Curve plot is composed of a set of (X,Y) pairs through which line segments are drawn to form a curve. To make VisIt draw a point glyph at the location of each (X,Y) point, click the **Show points** check box on the **Geometry** tab. You can control the size of the points by typing a new point size into the **Point size** text field. You can choose the type of symbol used to represent the points by using the **Symbol** combo box.

The number of points drawn can be controlled by the **Static** or **Dynamic** radio buttons. For **Static** mode, points are drawn at regular intervals controlled by the value of the **Point stride** text box. For **Dynamic** mode, the number of points drawn is view-dependent, with density controlled by the **Point density** text box.

**Adding Time Cues**

Time cues are most often used in conjunction with movie making. They allow for markers to be placed at certain positions along a curve, and/or for the curve to be cropped at the specified position. Time cues make it easier to see the current time position along a curve. Though most often created and controlled via scripting, the **Extras** tab in the **Curve attributes** window can also be used (see Figure 1.37). There are two types of markers: Ball and Line. They are controlled by the **Add Ball** and **Add Line** check boxes. They have separate color and size controls. To crop the line, select the **Crop** check box. The **Position of cue** text box controls the location along the curve where the ball and line are placed and where the cropped curve ends. **Figure 1.38** shows examples of curves created using different time cue settings.

**Polar coordinate system conversion**

If the curve data is in Polar instead of Cartesian coordinates, you can tell VisIt to convert by selecting the **Polar to Cartesian** option on the **Extras** tab. You can choose the **Order** to be **R_Theta** or **Theta_R** and choose **Radians** or **Degrees** for the **Units**. **Figure 1.39** shows an example.
Fig. 1.37: Curve plot attributes, extras tab

Fig. 1.38: Curve plot with time cues added at different positions, both uncropped and cropped.

Fig. 1.39: Curve plot before and after Polar coordinate transform (R-theta, radians)
Histogram Plot

The Histogram plot divides the data range of a scalar variable into a number of bins and groups the variable’s values, weighted by cell area or revolved volume, into different bins. The values in each bin are then used to create a bar graph or curve that represents the distribution of values throughout the variable’s data range. The Histogram plot can be used to determine where data values cluster in the range of a scalar variable. The Histogram plot is shown in Figure 1.40.

Setting the histogram data range

By default, the Histogram plot profiles a variable’s entire data range. If you want to restrict the Histogram plot so it only takes a subset of a variable’s data range into consideration when assigning values to bins, you can set the minimum and maximum values that will be considered by the Histogram plot. To specify a data range, click the Specify Range check box and then type in floating point numeric values into the Minimum and Maximum text fields in the Histogram plot attributes window (see Figure 1.41) before clicking its Apply button. Once the data range is set, the Histogram plot will restrict the values that it considers to the specified data range.

Setting the type of graph

The Histogram plot has two mode in which it can appear: curve and block. When the Histogram plot is drawn as a curve, it looks like the Curve plot. When the Histogram plot is drawn in block mode, it is drawn as a bar graph where each bin is plotted along the X-axis and the height of each bar corresponds to the number of values that were assigned to that bin. You can set change the Histogram plot’s appearance by clicking the Curve or Block radio buttons.
Setting the number of bins

The Histogram plot divides a variable’s data range into a number of bins and then counts the weighted values that fall within each bin. The bins and the counted data are then used to create a graph that represents the distribution of data within the variable’s data range. As the Histogram plot uses more bins, the graph of data distribution becomes more accurate. However, the graph can also become rougher because as the number of bins increases, the likelihood that no data values fall within a particular bin also increases. To set the number of bins for the Histogram plot, type a new number of bins into the **Number of Bins** text field and click the **Apply** button in the **Histogram plot attributes window**.

Setting the histogram calculation method

When the Histogram plot groups data values into bins, it weights the data value by the surface area or revolved volume of the cell so contributions from different sizes of cells are compared fairly. To change the calculation method used to weight the cells, click on the **Area** radio button to make VisIt use surface area or click on the **Revolved volume** radio button to make VisIt use the revolved volume of a 2D cell as the weighting multiplier used to group cells into the right bins.

Data scaling

There are three radio buttons that controls how the data values are scaled. The three options are:

- **Linear**: no scaling is applied. This is the default option.
- **Log**: the logarithms of all the scalars are binned.
- **Square Root**: the square roots of all scalars are binned.

Label Plot

The Label plot, shown in Figure 1.42, can display mesh information, scalar fields, vector fields, tensor fields, array variables, subset names, and material names. The Label plot is often used as a debugging device for simulation codes since it allows the user to see labels containing the exact values at the computational mesh’s nodes or cell centers. Since the Label plot’s job is to display labels representing the computational mesh or the fields defined on that mesh, it does not convey much information about the actual mesh geometry. Since having a Label plot by itself does not usually give enough information to understand the plotted dataset, the Label plot is almost always used with other plots.

Choosing the Label plot’s variable

You can choose the Label plot’s variable using the **Variable** menu under the **Plot list** the same way as you would with any other type of plot. One special property that distinguishes the Label plot from some of VisIt’s other plots is that it can plot multiple types of variables. The Label plot can display information for meshes, scalars, vectors, tensors, array variables, subsets, and materials so you will typically find more variables available for the Label plot than you would for other plots. When you choose a mesh variable for the Label plot, you can display both the mesh node numbers and cell numbers otherwise you are limited to displaying only the variable being plotted.

Showing node and zone numbers

The Label plot can display the node and cell numbers for the computational mesh if you have selected a mesh variable to plot. By default, the Label plot will display cell numbers only. The cell numbers will be displayed in the format...
Fig. 1.42: Label plot of the mesh overlayed on Pseudocolor and Mesh plots
Fig. 1.43: Label plot attributes window
most natural to the underlying mesh representation, which means that unstructured meshes will have cell numbers that are displayed as single integers while structured meshes will be displayed in i,j,k format when possible. If you want the Label plot to show a mesh’s node numbers in addition to its cell numbers, you can click on the Show nodes check box. If you no longer want the Label plot to show the mesh’s cell numbers, you can turn off the Show cells check box.

Restricting the number of labels

Most computational meshes contain many thousands, millions, or even billions of nodes and cells. Adding that many labels would quickly become burdensome on the computer and would result in a Label plot so dense that individual labels could no longer be read or even associated with their cell or node.

VisIt’s Label plot restricts the number of labels by default to some user-settable number of labels that can comfortably fit on the screen. The method used to restrict the number of labels differs for 2D and 3D plots. For 2D plots, the viewable portion of world space is periodically subdivided, based on the zoom level, into some number of bins to which labels are then assigned. As you zoom in on the Label plot, labels that go beyond the viewport are no longer drawn and new labels that were previously hidden take their place. This allows the Label plot to efficiently draw many labels without crowding the labels on top of each other. For 3D plots, the Label plot divides up the screen into a user-settable number of bins. All label coordinates are transformed so that they can be assigned to a screen bin and the label wins the screen bin if it is closer than the label that was previously in the bin. This ensures that a small subset of all possible labels is drawn and that they do not usually overlap on the screen. If you find that the labels appear to be from the back of the mesh instead of from the front, it’s quite possible that the normals generated for your mesh were inverted for some reason. To combat this problem, select Back or Front or Back from the Draw labels that face menu.

If you want to set the number of labels that the Label plot will draw, you can type in a new value into the spin box next to the Restrict number of labels to check box or use the up and down arrows on the spin box. If you want to force the Label plot to draw all labels, you can turn off the Restrict number of labels to check box. Sometimes making the Label plot draw all of the labels can be faster than drawing a subset of labels.

Depth testing for 3D Label plots

When VisIt draw plots in the visualization window, the plots’ geometries often correspond to only the outer surfaces of the originating datasets when those datasets are 3D. This means that the majority of plots consist of convex geometry and the normal test for only drawing labels that face front is often adequate to remove any labels that appear on faces that point away from the current camera. Some plots have geometries that consist of many concave regions, which the afore-mentioned test does not handle well. Plots with concave geometries will often have various pieces be incorrectly visible because though the surfaces may face the camera, they may be obscured by other geometry. When VisIt’s Label plot draws 3D geometry, it tries to enable additional depth testing to prevent front-facing labels in back of other surfaces from being drawn. Depth testing can degrade performance so, by default, it is allowed only when you are running VisIt on your local workstation. You can set the Label plot’s depth test mode to tell VisIt when to enable depth testing. To change the values for the depth test mode, click on one of the Auto, Always, Never radio buttons to the right of the Depth test mode label. If VisIt wants to use depth testing but is not allowed to then a warning message will be issued and you can set the depth test mode to Always.

Formatting labels

The Label plot provides several options for setting label format. First and foremost, you can set the label display format, which is how mesh node and cell numbers are displayed. By default, the Label plot will display labels in their most appropriate format with cell and node numbers for structured meshes displayed as logical i,j,k indices. Setting the label format is only possible for Label plots of structured meshes. To change the label format, select a new option from the Label display format menu.
The Label plot’s default behavior is to use the vis window’s foreground color but if you want labels to be a specific color, you can turn off the Use foreground color check box and select a new label color by clicking on the Label color color button.

The Label plot also allows control over the font used for the labels. Font name menu allows you to choose from among Arial, Courier and Times options. The labels can be bold or italic by checking the appropriate check boxes. Font scale is used to control the font size.

Note that when you are plotting a mesh variable, VisIt will make more controls in the Label plot attributes window so you can set color and font options for cells and nodes independently (see Figure ??).

Finally, the Label plot attributes window provides controls to determine the horizontal and vertical text justification used when drawing each label. To change the horizontal text justification, select a new value from the Horizontal justification menu. To change the vertical text justification, select a new value from the Vertical justification menu.

**Labeling subset names and material names**

The Label plot can label subset names and material names in addition to meshes and fields defined on those meshes. To add subset names or material names to your visualization, be sure to create a Label plot using a variable of either of those types. An example of a Label plot of material names is presented in Figure 1.46.

**Mesh Plot**

The Mesh plot, shown in Figure 1.47, displays the computational mesh over which a database’s variables are defined. The mesh plot is often added to the visualization window when other plots are visualized to allow individual cells to be clearly seen.
Fig. 1.45: Cell and node labels can be different colors when labeling mesh variables using additional controls in the Label plot attributes window.
Fig. 1.46: Label plot of materials
DB: f5e_05.obj

Mesh
Var: OBJ/Mesh

Fig. 1.47: Mesh plot
Fig. 1.48: Mesh plot window
Mesh plot opaque modes

By default, VisIt's Mesh plot draws in opaque mode so that hidden surface removal is performed when the plot is drawn and each face of the externally visible cells are outlined with lines. When the Mesh plot's opaque mode is set to automatic, the Mesh plot will be drawn in opaque mode unless it is forced to share the visualization window with other plots, at which point the Mesh plot is drawn in wireframe mode. When the Mesh plot is drawn in wireframe mode, only the edges of each externally visible cell face are drawn, which prevents the Mesh plot from interfering with the appearance of other plots. In addition to having an automatic opaque mode, the Mesh plot can be forced to be drawn in opaque mode or wireframe mode by clicking the On or Off Radio buttons to the right of the Opaque mode label in the Mesh plot attributes window.

Showing internal zones

Sometimes it is useful to create mesh plot that shows all internal zones for a 3D database. Rather then plotting just the externally visible zones, which is the Mesh plot’s default behavior, you can click the Show internal zones check box to force the Mesh plot to draw the edges of every internal zone.

Changing colors

There are two color controls for a Mesh plot. One, the mesh color, controls the color of mesh edge lines while the other, the opaque color, controls the color of mesh surface (areal) facets. For each color option, there are three choices

- A custom color chosen by the user.
- A random color chosen by VisIt.
- The Foreground (for mesh lines) or Background (for opaque facets) color.

The default is to use Foreground color for the mesh and Background color for the opaque color. In this mode, when these colors are changed via the Annotation controls, the Mesh plot obeys the newly selected colors. Otherwise, the Mesh plot maintains its chosen color (either custom or random).

The random color option is useful when displaying multiple meshes and the user simply needs to be able to easily distinguish among them.

Changing mesh line attributes

The Mesh plot’s mesh lines have two user-settable attributes that control their width and line style. You can set the line width and line style are set by selecting new options from the Line style or Line width menus at the top of the Mesh plot attributes window.

Changing point type and size

Controls for points are described in Point type and size.

Geometry smoothing

Sometimes visualization operations such as material interface reconstruction can alter mesh surfaces so they are pointy or distorted. The Mesh plot provides an optional Geometry smoothing option to smooth out the mesh surfaces so they look better when the mesh is visualized. Geometry smoothing is not done by default, you must click the Fast or High
radio buttons to enable it. The **Fast** geometry smoothing setting smooths out the geometry a little while the **High** setting works produces smoother surfaces.

**Pseudocolor plot**

The **Pseudocolor** plot, shown in Figure 1.49, maps a scalar variable’s data values to colors and uses the colors to “paint” values onto the variable’s computational mesh. The result is a clear picture of the database geometry painted with variable values that have been mapped to colors. You might try this plot first when examining a scientific database for the first time since it reveals so much information about the plotted variable.

![Fig. 1.49: Pseudocolor plot](image-url)
Data tab options

VisIt’s Pseudocolor plot attributes window Data tab allows you to change the data scaling, limits and centering, as well as change colors, opacity and control the plot Legend and lighting. (shown in Figure 1.50)

![Pseudocolor plot attributes window Data tab](image)

Fig. 1.50: Pseudocolor plot attributes window Data tab

Scaling the data

The scale maps data values to color values. VisIt provides three scaling options: Linear, Log, and Skew. Linear, which is the default, uses a linear mapping of data values to color values. Log scaling is used to map small ranges of data to larger ranges of color. Skew scaling goes one step further by using an exponential function based on a skew
factor to adjust the mapping of data to colors. The function used in skew scaling is \((s^d-1)/(s-1)\) where \(s\) is a skew factor greater than zero and \(d\) is a data value that has been mapped to a range from zero to one. The mapping of data to colors is changed by changing the skew factor. A skew factor of one is equivalent to linear scaling but values either larger or smaller than one produce curves that map either the high or low end of the data to a larger color range. To change the skew factor, choose Skew scaling and type a new skew factor into the Skew factor text field.

**Limits**

Setting limits for the plot imposes artificial minima and maxima on the plotted variable. This effectively restricts the range of data used to color the Pseudocolor plot. You might set limits when you are interested in only a small range of the data or when data limits need to be maintained for multiple time steps, as when playing an animation. In fact, we recommend setting the limits when producing an animation so the colors will correspond to the same values instead of varying over time with the range of the plotted variable. Setting limits often highlights a certain range in the data by assigning more colors to that data range.

To set the limits for the Pseudocolor plot, you must first select the limit mode. The limit mode determines whether the original data extents (data extents before any portions of the plot are removed), are used or the current plot data extents (data extents after any portions of the plot are removed), are used. To select the limit mode, choose either Use Original Data or Use Current Plot from the Limits menu.

The limits for the Pseudocolor plot consist of a minimum value and a maximum value. You may set these limits, and turn them on and off, independently of one another. That is, the use of one limit does not require the use of the other. To set a limit, check the Min or Max check box next to the Min or Max text field and type a new limit value into the Min or Max text field.

**Variable centering**

Variables in a database can be associated with a mesh in various ways. Databases supported by VisIt allow variables to be associated with a mesh’s zones (cells) or its nodes. When a variable is associated with a mesh’s zones, the variable field consists of one value for each zone and is said to be Zone-centered. When a variable is associated with a mesh’s nodes, there are values for each vertex making up the zone and the variable is said to be Node-centered.

There are three settings for variable centering: Natural, Nodal, and Zonal. Natural variable centering displays the data according to the way the variable was centered on the mesh. This means that node-centered data will be displayed at the nodes with colors being linearly interpolated between the nodes, and zone-centered data will be displayed as zonal values, giving a slightly “blocky” look to the picture. If Nodal centering is selected, all data is displayed at the nodes regardless of the variable’s natural centering. This will produce a smoother picture, but for variables which are actually zone-centered, you will lose some data (local minima and maxima). If you select Zonal centering, all data is displayed as if they were zone-centered. This produces a blockier picture and, again, it blurs minima/maxima for node-centered data.

**Changing the color table**

The Pseudocolor plot can specify which VisIt color table is used for colors. To change the color table, click on the Color table button, shown in Figure 1.51, and select a new color table name from the list of color tables. The list of color tables always represents the list of available VisIt color tables. If you do not care which color table is used, choose the Default option to use VisIt’s active continuous color table. New color tables can be defined using VisIt’s Color table window which is described later in this manual.
Fig. 1.51: Color table button

Default

- Accent
- Blues
- BrBG
- BuGn
- BuPu
- Dark2
- GnBu
- Greens
- Greys
- OrRd
- Oranges
- PRGn
- Paired
- Pastel1
- Pastel2
- PiYG
- PuBu
- PuBuGn
- PuOr
- PuRd
- Purples
- RdBu
- RdGy
- RdPu
- RdYlBu
- RdYlGn
- Reds
- Set1
- Set2
- Set3
- Spectral
- YIGn
- YIGnBu
- YIOrBr
- YIOrRd
- amino_rasmol
- amino_shapely
- bluehot
- caleblack
- calewhite
- contoured
- cpk_jmol
Opacity

You can make the Pseudocolor plot transparent by changing its opacity using the Opacity menu. There are four options:

1. **Fully opaque**: (the default), no transparency is applied.
2. **From color table**: opacity values are obtained from the active color table for the plot. If the color table doesn’t support opacities, the plot will be fully opaque.
3. **Constant**: A constant opacity is applied everywhere. A slider is provided to modify the opacity value. Moving the opacity slider to the left makes the plot more transparent while moving the slider to the right makes the plot more opaque.
4. **Ramp**: Opacity is applied on a sliding scale ranging from fully transparent (applied to the lowest values), to the opacity value chosen on the slider. If the slider is fully to the right, then the maximum values being plotted will be fully opaque.

Legend Behavior

The legend for the Pseudocolor plot is a color bar annotated with tick marks and numerical values. Below the color bar the minimum and maximum data values are also displayed. Setting the limits for the plot changes only the color-bar portion of the plot’s legend. It does not change the Min and Max values printed just below the color bar. Those values will always display the original data’s minimum and maximum values, regardless of the limits set for the plot or the effect of any operators applied to the plot.

Lighting

Lighting adds detail and depth to the Pseudocolor plot, two characteristics that are important for animations. The Lighting check box in the lower part of the Pseudocolor plot attributes window turns lighting on and off. Since lighting is on by default, uncheck the Lighting check box to turn lighting off.

Geometry tab options

VisIt’s Pseudocolor plot attributes window Geometry tab allows you to modify the appearance of lines and points, and change rendering options (shown in Figure 1.52)

Lines

The lines section can be useful when visualizing the results of a Poincare or Integral Curve operation.

There are three options for Line type: Lines (default), Tubes, and Ribbons.

The width of Lines can be changed by choosing an option from the Line width menu. The Tubes type has a Resolution option which represents the roundness of the tube. The higher the resolution, the rounder the tube.

Both the Tubes and Ribbons type have various methods for affecting the radius. The Radius option can be expressed either as an Absolute quantity or Fraction of the Bounding Box (default) by choosing one of these via the menu. A Variable can be chosen for the radius by checking the Variable radius checkbox, and choosing a variable from the menu.

Lines can also have glyphs at their head and tail. Glyph options are None (default), Sphere, and Cone. You can also specify Resolution and Radius for the glyphs.
Fig. 1.52: Pseudocolor plot attributes window, geometry tab
Point

Controls for points are described in *Point type and size*.

Representation

By default, the Pseudocolor plot renders as a Surface. It can also render in Wireframe or Points mode. Choose the representation by checking one or any combination of the three. Wireframe and Points will be rendered in the color specified by their corresponding Color buttons.

Geometry smoothing

Sometimes visualization operations such as material interface reconstruction can alter mesh surfaces so they are pointy or distorted. The Pseudocolor plot provides an optional Geometry smoothing option to smooth out the mesh surfaces so they look better when the plot is visualized. Geometry smoothing is not done by default, you must click the Fast or High radio buttons to enable it. The Fast geometry smoothing setting smooths out the geometry a little while the High setting produces smoother surfaces.

Scatter Plot

The Scatter plot (see Figure 1.53) allows you to combine multiple scalar fields into a point mesh so you can investigate the relationships between multiple input variables. You might, for example, want to see the behavior of pressure vs. density colored by temperature. The Scatter plot can take up to four scalar fields as input and can use up to three of them as coordinates for the created point mesh while one input variable can be used to assign colors to the point mesh. The Scatter plot provides individual controls for setting the limits of each input variable and also allows each input variable to be scaled so that all of the resulting points from disparate data ranges fit in a unit cube.

The Scatter plot attributes window is divided into two tabs: Inputs and Appearance. The Inputs tab is further subdivided into tabs for each input variable. Each tab for an input variable contains controls that pertain to selecting the input variable, settings its limits, or setting the role that the input variable will perform within the Scatter plot. Each input variable can have one of five roles that will be covered later. The Appearance tab contains controls for changing the Scatter plot’s appearance. Under the two main tabs, the Scatter plot attributes window features a small section that lists the roles that are used in the plot and which input variables are assigned to each role.

Scatter plot wizard

Plots are typically created in VisIt when you choose a variable from one of the Plot menus. Since the Scatter plot takes as input up to four input variables and typical plot creation only initializes one variable, you can imagine that if a Scatter plot was created the usual way, only one of its many input variables would be initialized. Furthermore, to initialize the plot, you would have to open the Scatter plot attributes window and select the other variables. Since that would not be a very straightforward way to create a Scatter plot, VisIt now has support for plot wizards. A plot wizard is a simple dialog window that pops up when you select a variable to plot. A plot wizard leads you through a series of questions that allow VisIt to more fully initialize a new plot. The Scatter plot wizard prompts you for the scalar variable to use for the Y-Axis, the variable to use for the Z-Axis (optional), and the variable to use for the plot’s colors (optional).

Selecting a variable

Three of the Scatter plot’s four input variables can be set in the Scatter plot attributes window. The first input variable cannot be changed from within the Scatter plot attributes window because that is the default variable used
Fig. 1.53: Example of Scatter plot
Fig. 1.54: Scatter plot attributes window
by the plot. If you want to change the first input variable, you can use the Variables menu under the Plot list. If you want to select a different variable for any of the other input variables, you would first click on the input variable’s tab and then you would select a new variable by making a selection from the tab’s Variable button. Note that any combination of nodal and cell-centered variables can be chosen. The Scatter plot will recenter any input variables whose centering does not match the first input variable’s centering.

**Setting an input variable’s role**

Each of the Scatter plot’s input variables has a role that you can set which determines how the input variable is used by the Scatter plot. An input variable can be used for the X, Y, Z coordinates, for the color, or it can have no role. The role of the input variable is not fixed because you might want to change roles many times and it is much less work to change only the roles instead of reselecting variables, limits, and scaling for an input variable. The flexibility of selecting a role for an input variable makes it convenient to turn off colors or the Z coordinate with little effort. To change the role for an input variable, select a new role from the input variable’s Role combo box. If you select a role that is already played by another input variable, VisIt will give the current input variable the selected role and set the input variable that previously had the selected role so that it has no role.

Each of the Scatter plot roles and their associated input variables are listed in the bottom of the Scatter plot attributes window. Roles that have an input variable have the name of the input variable printed next to the name of the role so looking through all of the input variable tabs to determine what the Scatter plot should look like is not required. Roles that have no assigned input variable are grayed out.

**Setting the minimum and maximum values**

The Scatter plot allows you to set minimum and maximum limits on the values considered for inclusion into the created point mesh. If an input variable’s data value does not lie in the specified minimum/maximum value data range then the point is not included in the created point mesh. Note that setting limits does not cause points to be removed.
when data values in the color role fall outside of the specified limits. To set the minimum value to be allowed in the
created point mesh, click on the Min check box and type a new minimum value into the Min text field. To set the
maximum value to be allowed in the created point mesh, click on the Max check box and type a new value into the
Max text field.

Scaling an input variable

Sometimes input variable data values are clustered in a certain range of the data. When this is the case, the points in
the Scatter plot will bunch up in one or more dimensions. For more uniformly spaced points, you might try scaling
one or more input variables. Each input variable can be scaled in the three common ways: Linear, Log, and Skew. To
set the scaling method used for the input variable, click on the Linear, Log, or Skew radio buttons. If you choose the
Skew scaling method then you should also enter a value greater than zero into the Skew factor text field to determine
the function used for skew scaling.

Since the Scatter plot’s input variables are likely to have wildly different data ranges, the Scatter plot provides an
option to independently scale each input variable so it is in the range [0,1] so the resulting plot fits entirely in a cube.
If you prefer to see the Scatter plot without this corrective scaling, you can turn off the Scale to cube check box on
the Scatter plot attribute window’s Appearance tab.

Setting the colors

The Scatter plot can map scalar values to colors like the Pseudocolor plot (Pseudocolor plot) does or it can color all
points using a single color. If you have set one of the input variables to have a color role then the Scatter plot will
map that input variable’s data values to colors using the specified color table. To change the color table used by the
Scatter plot, click on the Color table button and select a new color table from the list of available color tables. If the
Scatter plot has been configured such that none of the input variables is playing the color role then the Scatter plot’s
points will be drawn using one color. When the Scatter plot draws its points using a single color, its default behavior
is to color the points using the vis window’s foreground color. If you want to instead use a different color, turn off the
Use foreground check box and click on the Single color color button to select a new color.

Setting point properties

Controls for points are described in Point type and size.

Subset Plot

The Subset plot (example in Figure 1.57) is used to display subsets. The typical scientific database can be decomposed
into many different subsets. Frequently a database is decomposed into non-material subsets such as domains or groups.
In AMR meshes, subsets can consist of levels or patches. The Subset plot draws the database with its various subsets
color coded so they can be distinguished. For more information about subsets, see the Subsetting chapter.

Changing colors

The main portion of the Subset plot attributes window, also known as the Subset colors area, is devoted to setting
subset colors. The Subset colors area contains a list of subset names with an associated subset color. Subset plot
colors can be assigned three different ways, the first of which uses a color table. A color table is a named palette of
colors that you can customize to suite your needs. When the Subset plot uses a color table to color subsets, it selects
colors that are evenly spaced through the color table based on the number of subsets. For example, if you have three
subsets and you are coloring them using the “xray” color table, the Subset plot picks three colors out of the color table
Fig. 1.56: Scatter plot attributes window’s Appearance tab
Fig. 1.57: Example of Subset plot of an AMR Mesh
Fig. 1.58: Subset plot attributes window

- **Subset colors**
  - Color table
  - Default
  - Invert
  - Single
  - Multiple
  - Subset colors for "level0, patch0" to "level1, patch9"

- **Opacity**
  - 100%

- **Options**
  - Wireframe
  - Draw internal surfaces

- **Point / Line Style**
  - Point type: Point
  - Point size (pixels): 2
  - Scale point size by variable
  - Line style: solid
  - Line width: 1

- **Geometry**
  - Smoothing: None
  - Fast
  - High

- **Misc**
  - Legend

Buttons:
- Make default
- Load
- Save
- Reset
- Apply
- Post
- Dismiss
so your levels are colored black, gray, and white. To color a Subset plot with a color table, click on the Color table radio button and choose a color table from the Color table menu to right of the Color table radio button.

If you want all subsets to be the same color, click the Single radio button at the top of the Subset plot attributes window and select a new color from the Popup color menu that is activated by clicking on the Single color button. The opacity slider next to the Single color button sets the opacity for the single color.

Clicking the Multiple radio button causes each subset to be a different, user-specified color. By default, multiple colors are set using the colors of the discrete color table that is active when the Subset plot is created. To change the color for any of the subsets, select one or more subsets from the list of subsets and click on the Color button to the right of the Multiple radio button and select a new color from the Popup color menu. To change the opacity for a subset, move Multiple opacity slider to the left to make the subset more transparent or move the slider to the right to make the subset more opaque.

The Subset plot attributes window contains a list of subset names with an associated subset color. To change a subset’s color, select one or more subsets from the list, click the color button and select a new color from the popup color menu.

**Opacity**

The Subset plot’s opacity can be changed globally as well as on a per subset basis. To change subset opacity, first select one or more subsets in the subset list and move the opacity slider next to the color button. Moving the opacity slider to the left makes the selected subsets more transparent and moving the slider to the right makes the selected subsets more opaque. To change the entire plot’s opacity globally, use the Opacity slider near the bottom of the window.

**Setting point properties**

Albeit rare, the Subset plot can be used to plot points that belong to different subsets so the Subset plot attributes window provides controls that allow you to set the representation and size of the points. You can change the points’ representation using the Point Type combo box. The available options are: Box, Axis, Icosahedron, Point, and Sphere. To change the size of the points, you can enter a new floating point value into the Point size text field. Finally, you can opt to scale the points’ glyphs using a scalar expression by turning on the Scale point size by variable check box and by selecting a scalar variable from the Variable button to the right of that check box.

**Wireframe mode**

The Subset plot can be modified so that it only displays outer edges of subsets. This option usually leaves lines that give only the rough shape of subsets and where they join other subsets. To make the Subset plot display in wireframe mode, check the Wireframe check box near the bottom of the Subset plot attributes window.

**Drawing internal surfaces**

When you make one or more subsets transparent, you might want to make the Subset plot draw internal surfaces. Internal surfaces are normally removed from Subset plots to make them draw faster. To make the Subset plot draw internal surfaces, check the Draw internal surfaces check box near the bottom of the Subset plot attributes window.

**Geometry smoothing**

Sometimes visualization operations such as material interface reconstruction can alter mesh surfaces so they are pointy or distorted. The Subset plot provides an optional Geometry smoothing option to smooth out the mesh surfaces so
they look better when the plot is visualized. Geometry smoothing is not done by default, you must click the **Fast** or **High** radio buttons to enable it. The **Fast** geometry smoothing setting smooths out the geometry a little while the **High** setting works produces smoother surfaces.

**Tensor plot**

The Tensor plot, shown in **Figure 1.59**, displays tensor variables using ellipsoid glyphs to convey information about a tensor variable’s eigenvalues. Each glyph’s scaling and rotation is controlled by the eigenvalues/eigenvectors of the tensor as follows: for each tensor, the eigenvalues (and associated eigenvectors) are sorted to determine the major, medium, and minor eigenvalues/eigenvectors. The major eigenvalue scales the glyph in the x-direction, the medium in the y-direction, and the minor in the z-direction. Then, the glyph is rotated so that the glyph’s local x-axis lies along the major eigenvector, y-axis along the medium eigenvector, and z-axis along the minor.

![Example of Tensor plot](image)

**Fig. 1.59: Example of Tensor plot**
Changing the tensor colors

The Tensor plot can be colored by a solid color or by the corresponding to the largest eigenvalue. To color the Tensor plot by eigenvalues, click the Eigenvalues radio button and then select a color table name from the color table button to the right of the Eigenvalues radio button. To make all tensor glyphs be the same color, click the Constant radio button and choose a color by clicking on the Constant color button and selecting a new color from the Popup color menu.
Setting the tensor scale

The Tensor plot’s tensor scale affects how large the ellipsoidal glyphs that represent the tensor are drawn. By default, VisIt computes an automatic scale factor based on the length of the bounding box’s diagonal to multiply by the user-specified scale factor. This ensures that the tensors are some reasonable size independent of the size of the mesh. To change the tensor scale, type a new floating point number into the Scale text field and click the Apply button in the Tensor plot attributes window. If you want to turn off automatic scaling so the size of the tensors is solely determined by the scale in the Scale text field, turn off the Auto scale check box. Yet another scaling option for tensors is scaling by magnitude. When the Scale by magnitude check box is checked, the magnitude of the tensor’s longest eigenvector is used as a scale factor that is multiplied into the scale determined by the user-specified scale and the automatic scale factor.

Setting the number of tensors

When visualizing a large database, a Tensor plot will often have too many tensors to effectively visualize so the Tensor plot provides controls to reduce the number of tensors to a number that looks appealing in a visualization. You can accomplish this reduction by setting a fixed number of tensors or by setting a stride. To set a fixed number of tensors, select the N tensors radio button and enter a new number of tensors into the N tensors text field. To reduce the number of tensors by setting the stride, select the Stride radio button and enter a new stride value into the Stride text field.

Truecolor plot

The Truecolor plot, shown in Figure 1.61, is used to plot images of observational or experimental data so they can be compared to other plots, possibly of related, simulated data, in the same visualization window. The Truecolor plot takes in a color variable, represented in VisIt as a three or four component vector, and uses the vector components as the red, green, blue, and alpha values for the plotted image. This allows you access to many more colors than other plots like the Pseudocolor plot, which can be used only to plot a single color component of an image.

Vector plot

The Vector plot (example shown in Figure ??) displays vector variables using glyphs that indicate the direction and magnitude of vectors in a vector field.

Setting vector color

The vectors in the Vector plot can be colored by the magnitude of the vector variable or they can be colored using a constant color. Choose the coloring method by clicking on either the Magnitude radio button or the Constant color button. When vectors are colored by a constant color, you can change the color by clicking on the color button next to the Constant radio button and choosing a new color from the Popup color menu. When vectors are colored by magnitude, the color is determined by one of VisIt’s color tables, which can be chosen from the Color table button next to the Magnitude radio button.

If you choose to color the vectors by their magnitudes, you have the option of also specifying minimum and maximum values to aid in the mapping of vector magnitude to color. The options that are used to aid coloring are collectively known as limits. Limits can apply to all vectors that exist in the dataset or just the vectors that have been drawn by the Vector plot. To specify which, choose the appropriate option from the Limits combo box. When you specify a minimum value all vectors with magnitudes less than the minimum value are colored using the color at the bottom of the color table. When you specify a maximum value all vectors with magnitudes larger than the maximum value are colored using the color at the top of the color table. To provide a minimum value, check the Min check box and type a new minimum value into the Min text field. To provide a maximum value, check the Max check box and type a new maximum value into the Max text field.
Fig. 1.61: Truecolor Plot
Fig. 1.62: Truecolor Plot Attributes
DB: globe.silo
Cycle: 0    Time: 0
1.3. Plots

Fig. 1.63: Vector Plot Attributes and Example Rendering

- Vector placement: Adapted to resolution of mesh, Uniformly located throughout mesh
- Vector amount: Fixed number 1200, Stride 1
- Only show vectors on original nodes/cells

Limits
- Use Original Data
- Minimum 0, Maximum 1

Color
- Magnitude: Default, Invert
- Constant

Misc
- Legend

Scale
- Scale 0.3
- Scale by magnitude, Auto scale

Style
- Glyph type: Arrow, Draw head
- Sizer 0.3
- Arrow body: Line, Style solid
- Width 1
Vector scaling

The size of the vector glyphs has a tremendous effect on the Vector plot’s readability. VisIt uses an automatically computed scaling factor based on the diagonal of the bounding box as the size for the largest vector. You can augment this size by entering a new scale factor in to the Scale text field. It is also possible to turn off automatic scaling by turning off the Auto scale check box. When automatic scaling is turned off, the vectors in the Vector plot are the length specified in the Scale text field.

If you want each vector to be further scaled by its own magnitude, you can turn on the Scale by magnitude check box. When the Scale by magnitude check box is off, all vectors are the same length as determined by the automatically computed scale factor and the user-specified scale.

Heads on the vector glyph

You can control the vector head size by typing a new value into the Head size text field, which is the fraction of the entire vector’s length that will be devoted to the vector’s head. Vectors in the Vector plot can be drawn without vector heads so that only the line part of the vector glyph is drawn. This results in cleaner plots, but the vector direction is lost. To turn off vector heads, uncheck the Draw head check box at the bottom of the Vector Attributes Window.

Tails on the vector glyph

The length of the tails on the vector glyph are determined by the vector scaling factors that have been enabled. You can also set properties that determine the location and line properties used to draw a vector glyph’s tail. First of all, you can set the line style used to draw the vector glyph’s tail by choosing a line style from the Line style combo box. You can choose a new line width for the vector glyph’s tail by choosing a new line width from the Line width combo box. Finally, you can determine where the origin of the vector is on the vector glyph. The vector origin is a point along the length of the vector that is aligned with the node or cell center where the vector glyph will be drawn. The available options are: Head, Middle, and Tail. You can choose a new Vector origin by clicking on one of the Head, Middle, or Tail radio buttons.

Setting the number of vectors

When visualizing a large database, a Vector plot will often have too many vectors. The Vector plot becomes incomprehensible with too many vectors. VisIt provides controls to thin the number of vectors to a number that looks appealing in a visualization. You can accomplish this reduction by setting a fixed number of vectors or by setting a stride. To set a fixed number of vectors, select the Fixed vectors radio button and enter a new number of vectors into the corresponding text field. To reduce the number of vectors by setting the stride, select the Stride radio button and enter a new stride value into the Stride text field.

Volume plot

This plot, shown in (Figure 1.64), uses both color and transparency to visualize 3D scalar fields. The values in the data range have associated color and opacity values that allow parts of the dataset to become partially or completely transparent. This plot captures internal details while keeping the whole dataset at least partially visible.

The Volume plot uses a visualization technique known as volume-rendering, which assigns color and opacity values to a range of data values. The colors and opacities are collectively known as a volume transfer function. The volume transfer function determines the colors of the plot and which parts are visible. The Volume plot uses three types of volume-rendering to visualize data.

The first volume rendering method, hardware-accelerated splatting, resamples the entire database onto a small rectilinear grid and then, at each node in the grid, draws a small textured polygon. The polygon gets its colors and opacity
Fig. 1.64: Volume Plot Example
from the transfer function. This method is fast due to its use of graphics hardware but it can require a large number of points in the resampled mesh to look accurate.

Like the first volume rendering method, the second method, hardware-accelerated 3D texturing, resamples the entire database onto a small rectilinear grid. Once the data has been resampled, it is converted into a 3D texture using the Volume plot’s volume-transfer function and gets loaded into the video card’s texture memory. The Volume plot then draws a set of planes that are perpendicular to the view vector from back to front, with each plane getting the pre-loaded texture mapped onto it. The resulting image is very crisp and captures details not evident when the splatting method is used.

The third volume-rendering technique, called ray-casting, used by the Volume plot is not hardware accelerated. In ray-casting, a ray is followed in reverse from the computer screen into the dataset. As a ray progresses through the dataset, sample points are taken and the sample values are used to determine a color and opacity value for the sample point. Each sample point along the ray is composited to form a final color for the screen pixel. Rays are traced from closest to farthest to allow for early ray termination which stops the sampling process when the pixel opacity gets above a certain threshold. This method of volume-rendering yields superior pictures at the cost of speed and memory use.

The Volume Attributes Window, shown in (Figure ??), is divided into two main tabs. The Rendering Options tab controls the rendering setting. Each volume rendering method has a different set of inputs. For example, splatting and 3D Texturing both specify the total number of samples, while ray-casting settings are specified in samples-per-ray. Additionally, the Rendering Options tab contains controls for lighting. 1D transfer function tab controls how the data is mapped onto colors and the opacities to use for different scalar values.

Setting colors

You can design the color component of the volume transfer function using the controls in 1D Transfer Function tab of the Volume Attributes Window. The controls are similar to the controls for the Color Table Window. There is a color spectrum that has color control points which determine the final look of the color table. Color control points are added and removed using the + and – buttons. Dragging control points with the mouse moves them and changes their order. Right-clicking on a color control point displays a popup color menu from which a new control point color can be chosen.

Limits

The 1D transfer function tab provides controls for setting the limits of the variable being plotted. Limits are artificial minima or maxima that are specified by the user. Setting the limits to a smaller range of values than present in the database cause the plot’s colors to be distributed among a smaller range of values, resulting in a plot with more color variety.

To set the limits are set by first clicking the Min or Max check box next to the Min or Max text field. Clicking a check box enables a text field into which you can type a new minimum or maximum value.

Like VisIt’s other plots that map scalar values to colors, the Volume plot allows for the data values to be scaled using Linear, Log, and Skew functions. To select a scaling function other than linear where values in the data range are mapped 1:1 to values in the color range, click on the Log or Skew radio buttons.

Setting opacities

The 1D transfer function tab provides several controls that allow you to define the opacity portion of the volume transfer function. The opacity portion of the volume transfer function determines what can be seen in the volume-rendered image. Data values with a lower opacity allow more to be seen and give the plot a gel-like appearance, while data values with higher opacity appear more solid and occlude objects behind them. The controls for setting opacities are located at the button of the window in the Opacity area.
1.3. Plots
Fig. 1.65: Volume Attributes Window
You can set opacity three ways. You can hand-draw an opacity map, create it by designing curves that specify the opacity when they are added together, or use the opacities in the color table, if present. All methods use the controls shown in Figure ??.

The interaction mode determines how opacity is set. Clicking on the Freeform or Gaussian radio buttons selects the interaction mode. If the interaction mode switches from Gaussian to Freeform, the shape constructed by the Gaussian controls is copied to the Freeform control. Both controls pretend that the plot’s data range is positioned horizontally such that the values on the left of the control correspond to the low data values while the values on the right of the control correspond to high data values. In addition to the color map, there is a histogram of the current data to aide in setting opacity of interesting values. The vertical direction corresponds to the opacity for the given data value. Taller curves are more opaque while shorter curves are more transparent.

To design an opacity map using the Freeform control, position the mouse over it and click the left mouse button while moving the mouse. The shape traced by the mouse is entered into the Freeform control so you can draw the desired opacity curve. Immediately under the Freeform control, there are four buttons, shown in (Figure 1.67), which can be used to manipulate the curve. The first three buttons initialize a new curve. The black button makes all data values completely transparent. The ramp button creates a linear ramp of opacity that emphasizes high data values. The white button makes all data values completely opaque. The Smooth button smooths out small bumps in the opacity curve that occur when drawing the curve by hand.

The Gaussian control used during Gaussian interaction mode is complex but it provides precise control over the shape
of a curve. The basic paradigm followed by the Gaussian control is that new curves are added and reshaped to yield the desired opacity curve. You add new curves by clicking and dragging in the control. Right clicking with the mouse on an existing curve removes the curve. Each curve has five control points which can change the curve’s position and shape. The control points are shown in along with the shapes that a curve can assume. A control point changes color when it becomes active so there you know which control point is used. Curves start as a smooth Gaussian shape but they can change between the shapes shown in by moving the shape control point up and down or left and right. Opacity maps are typically created by adding several curves to the window and altering their shapes and sizes until the desired image is obtained in the visualization window. The Attenuation slider, the final control involved in creating an opacity map, controls the opacity of the entire opacity map defined by the Freeform or Gaussian controls. It provides a knob to scale all opacities without having to modify the opacity map.

Changing the opacity variable

The variable used to determine opacity does not have to be the plotted variable. Having a different opacity variable than the plotted variable is useful for instances in which you want to determine the opacity using a variable like density while coloring the plot by another variable such as pressure. To change the opacity variable, select a new variable from the Opacity variable variable menu. By default, the plotted variable is used as the opacity variable. This is implied when the Opacity variable variable button contains the word default. Even when “default” is chosen, it is possible to set artificial data limits on the opacity variable by entering new values into the Min or Max text fields.

Controlling image quality

When the Volume plot is drawn with graphics hardware, the database is resampled onto a rectilinear grid that is used to place the polygons that are drawn to produce the image. You can control the coarseness of the resampled grid with the Number of samples text field and slider. To increase the number of sample points, enter a larger number into the Number of samples text field or move the slider to the right. Note that the slider is on an exponential scale and moving it to the right increases the number of sample points exponentially.

In addition to setting the number of samples, when the Volume plot is using the 3D texturing method, you can set the number of planes to be drawn from back to front. Increasing the number of planes can help to reduce the amount of aliasing in the resulting image. However, as the Volume plot uses a higher number of planes, more work must be done to draw the plot and it takes a little longer to draw. To set the number of planes, enter a new number of planes into the Number of slices text field.

When the Volume plot is drawn in ray casting mode, the number of samples along each ray that is cast through the data becomes important. Having too few sample points along a ray gives rise to sampling artifacts such as rings or voids. You should adjust this number until you are satisfied with the image. More samples generally produce a better image, though the image will take longer to produce. To change the number of samples per ray, enter a new number of samples per ray into the Samples per ray text field.

When using lighting, the gradient calculation method that the Volume plot uses influences the quality of the images that are produced. By default, VisIt uses the Sobel operator, which uses more information from adjacent cells to calculate a gradient. When the Sobel operator is used to calculate the gradient, lighting usually looks better. The alternative gradient calculation method is centered-differences and while it is much less compute intensive than the Sobel operator, it also produces lesser quality gradient vectors, which results in images that are not lit as well. To change the gradient calculation method, click on either the Centered diff or Sobel radio buttons.

Software rendered images

The Volume plot uses hardware-accelerated graphics by default. While you will want to operate in this mode most of the time, since it’s faster, images drawn by software are more accurate. To get a more accurate image, select Ray casting from the Rendering method combo box. When the Volume plot is set to use ray casting as its rendering mode,
VisIt recalculates what the image should look like in software mode. Note that this can be a time-consuming process if the database being used is large or if the visualization window is large. We recommend shrinking the size of the visualization window before changing the rendering method to ray casting to reduce the time and resources required to draw the plot. It is worth noting that if you have a large dataset with intricate details, the software volume rendering method is the best method to use because it scales well in parallel. Using a parallel compute engine can greatly speed up the rate at which software volume rendering operates as long as the dataset is domain-decomposed into equal-sized pieces.

**Lighting**

The Volume plot can use lighting to enhance the look of the plot. Lighting is enabled by default but you can disable it by unchecking the **Lighting** check box near the bottom of the window. Note that lighting is not currently available when the Volume plot is using the ray casting volume renderer.

### 1.4 Operators

This chapter explains the concept of an operator and goes into detail about each of VisIt’s operators.

#### 1.4.1 Working with Operators

An operator is a filter applied to a database variable before the compute engine uses that variable to generate a plot. VisIt provides several standard operator types that allow various operations to be performed on plot data. The standard operators perform data restriction operations like planar slicing, spherical slicing, and thresholding, as well as more sophisticated operations like peeling off mesh layers. All of VisIt’s operators are plugins and new operators can be written to extend VisIt in new ways. See the [wiki](#) for more details on creating new operator plugins or send an e-mail inquiry to visit-users@elist.ornl.gov.

**Managing operators**

When an operator is applied to a plot, it modifies the data that the plot uses to generate a visualization. Any number of operators can be applied to a plot. Each operator added to a plot restricts or modifies the data that is supplied to the plot. Very sophisticated visualizations can be created by using a series of operators.

The controls for the operators are found in the same location as the plot controls. The plot list, which displays the list of plots found in the current visualization window, also displays the operators applied to each plot. Each entry in the plot list displays the database name (when there is more than one open source), the plot type, the variable, and all operators that are applied to the plot. When an operator is applied to a plot, the name of the operator is inserted in front of the plot variable. If multiple operators are applied to a plot, the most recently added operator appears first when reading left to right while the operator that was applied first appears just to the left of the variable name. Plot list entries can also be expanded to allow the user to add, remove, reorder, and change the attributes of operators.

**Adding an operator**

Operators are added by selecting an operator from the **Operators** menu, shown in Figure 1.70. If an operator listed in this chapter is not listed in the **Operators** menu then the operator might not be loaded by default. To enable additional operators, use the **Plugin Manager Window**. When an operator is added, it applies the operator to the selected plots in the plot list unless the **Apply operators to all plots** check box is checked, in which case, the selected operator is applied to all plots in the plot list. By default, operators are applied to all plots in the plot list.
Fig. 1.69: The plots area

Fig. 1.70: The operators menu
When an operator is added to a plot, the name of the operator appears in the plot list entry to the left of the variable or any previously applied operator. When an operator is added to an already generated plot, the plot is reset back to the new state to allow the user an opportunity to set the operator’s attributes before the plot is regenerated. To regenerate the plot with the newly added operator, press the **Draw** button. It is also possible to apply an operator by clicking an operator attributes window’s **Apply** button. When this occurs, a dialog window appears asking the user if the operator should be applied to the selected plots (see Figure 1.71).

![Fig. 1.71: The add operator dialog](image)

**Expanding plots**

Plot list entries are normally collapsed by default with the operators applied to the plots shown in the plot list as a series of nested operators, which finally take a variable as an argument. The plot list allows plot list entries to be expanded on a per-plot basis so the user can get to each individual operator that is applied to a plot. To expand a plot list entry, click on its expand button, shown in Figure 1.72. When a plot list entry is expanded, the plot’s database (if there is more than one open source), the variable, all the operators, and finally the plot get their own line in the plot list entry. This is significant because it allows operators to have additional controls to let you reposition them in the pipeline or remove them from the middle of the pipeline without having to first remove other operators.

![Fig. 1.72: The add operator dialog](image)
Changing the order of operators

Sometimes with several operators applied, it is useful to change the order of the operators. For example, the user might want to apply a Slice operator before a Reflect operator instead of after it to reduce the amount of data that VisIt must process in order to draw your plot. The order in which operators are applied often has a significant impact on the visualization. Using the previous example, suppose a plot is sliced before it is reflected. The resulting visualization is likely to have a reflected slice of the original data. If the order of the operators was reversed so that the Reflect operator came first, the Slice operator’s slice plane might not intersect the reflected data in the same way, which could result in a totally different looking visualization.

The plot list entry must be expanded in order to change the order of its operators. Once the plot list entry is expanded, each operator is listed in the order in which they were applied and each operator has small buttons to the right of its name that allow the operator to be moved up or down in the pipeline. To move an operator closer to the database so it is executed before it would have been executed before, click on the Up button next to an operator’s name. Moving the operator closer to the database in the pipeline is called demoting the operator. Clicking the Down button next to an operator’s name moves the operator to a later stage of the pipeline. Moving an operator to a later stage of the pipeline is known as promoting the operator since the operator appears closer to the plot in the expanded plot entry. Operators in the plot list entry that can only be moved in one direction have only the Up button while operators in the middle of the pipeline have both the Up button and the Down button.

Removing operators

There are two ways to delete an operator from a plot. The last two entries in the Operators menu have options that remove one or more operators. To remove only the last applied operator, select the Remove last option from the Operators menu. To remove all the operators applied to a plot, select the Remove all option from the Operators menu. Unless the Apply operator to all plots check box is checked, operators are only removed from selected plots. When an operator is removed in this manner and the plot has already been generated, it is immediately regenerated. The Operators menu has controls that allow the last operator applied to a plot to be removed or all of a plot’s operators to be removed. VisIt also provides controls that let you remove specific operators from the middle of a plot’s operator list. First expand the plot list entry by clicking its Expand button and then click on the red X button next to the operator to be deleted. When an operator is deleted using the red X buttons, the plot is reset back to the new state so
the **Draw** button must be clicked to regenerate the plot. See Figure 1.74 for an example of deleting an operator from the middle of a plot’s operator list.

![Fig. 1.73: The controls for changing operator order](image)

**Setting operator attributes**

Each operator type has its own attributes window used to set attributes for that operator type. Operator attribute windows are brought up by selecting the operator type from the **OpAtts** (Operator attributes) menu shown in Figure 1.75.

When there is only one operator of a given type in a plot’s operator list, setting the attributes for that operator type will affect that one operator. When there are multiple instances of the same type of operator in a plot’s operator list,
only the active operator’s attributes are set if the active operator is an operator of the type whose attributes are being set. The active operator is the operator whose attributes are set when using an operator attributes window and can be identified in an expanded plot entry by the highlight that is drawn around it (see Figure 1.76). To set the active operator, expand a plot entry and then click on an operator in the expanded plot entry’s operator list.

Setting the active operator is useful when there are multiple operators of the same type applied to the same plot. For example, there might be two Transform operators applied to a plot in order to scale a plot with one operator and then rotate the plot with the second Transform operator. In this case the user could add two Transform operators, make the first Transform operator active, set the scaling attributes, make the second Transform operator active, and set the rotation attributes.
1.4.2 Operators that Generate New Variables

Some of VisIt’s operators act more like expressions in that they generate new variables that can be plotted. The variable type they output does not necessarily match the variable type they accept as input. For example, the IntegralCurve operator accepts a Vector and outputs a Scalar, while the ConnectedComponents operator accepts a Mesh and outputs a Scalar.

Most of the operators that generate new variables are best applied using the operators submenu of a particular plot’s variable menu. See Figure 1.77.

It is probably best after applying an operator in this fashion to open the Operator’s attributes window to ensure good settings for your data before clicking Draw.

Operators that generate Scalars:
1. Connected Components
2. DataBinning
3. Flux
4. IntegralCurve
5. LCS
6. LimitCycle
7. ModelFit
8. Poincare
9. StatisticalTrends

Operators that generate Vectors:
1. LCS
2. SurfaceNormal

Operators that generate Curves:
Fig. 1.77: The menu for applying an operator that generates a new variable.
1. DataBinning
2. Lagrangian
3. LimitCycle
4. Lineout

### 1.4.3 Operator Types

VisIt is installed with operator plugins, which perform a wide variety of functions. Some of the operators are not be enabled by default so they do not show up in the Operator menu. Use the Plugin Manager Window, which can be opened by clicking on the Plugin Manager option in the Main Window’s Preferences menu, to enable additional operators or disable operators that you rarely use.

**Box operator**

The Box operator, which is mostly intended for use with 3D datasets, removes areas of a plot that are either partially or completely outside of the volume defined by an axis-aligned box. The Box operator does not clip cells that straddle the box boundary, it just removes the cells from the visualization leaving jagged edges around the edges of the box where cells were removed.

![Box operator example](image)

**Fig. 1.78: Box operator example (original on left, with Box operator applied on right)**

**Setting how cells are removed**

The Box operator can either remove cells that are totally outside of the box or it can remove those cells outside of the box and cells that are only partially outside of the box. By default, the Box operator only removes cells that are completely outside of the box. To make the Box operator also remove cells that are partially outside of the box, you click the All radio button in the Box attributes window (shown in Figure 1.78). Selecting the Inverse option will return everything in the mesh except those cells bounded by the selected box.
Resizing the box

The Box operator uses an axis aligned box to remove cells from the visualization so the box can be specified as a set of minimum and maximum values for X, Y, and Z. To set the size of the box using the Box operator attributes window, you type new coordinates into the X Minimum, X Maximum, Y Minimum, Y Maximum, Z Minimum, or Z Maximum text fields.

The Box operator can also be resized interactively with VisIt’s Box tool (for more information, see the Interactive Tools chapter). If you want to use the Box tool to resize the Box operator’s box, first make sure to select the plot that uses the Box operator in the Plot list and then enable the Box tool. When the Box tool appears, it uses the same box as the Box operator. Moving or resizing the Box tool causes the Box operator to also move or be resized and the plots in the visualization window get regenerated with the new box.

Clip operator

The Clip operator can remove certain shapes from a dataset before it is plotted. More specifically, the Clip operator can clip away box- or sphere-shaped regions from a database. The database remains in its original dimension after being clipped by the Clip operator and since the Clip operator manipulates the database before it is plotted, the surfaces bounding the removed regions are preserved in the final plot. While being geared primarily towards 3D databases, the Clip operator also clips 2D databases. When applied to 2D databases, the Clip operator can remove rectangular or circular regions from the database. Figure 1.80 shows a Pseudocolor and Mesh plots with a Clip operator.

Removing half of a plot

The Clip operator uses up to three planes to define the region that is clipped away. Each plane is specified in origin-normal form where the origin is a point in the plane and the normal is a vector that is perpendicular to the plane. When a plane intersects a plot, it serves as a clipping boundary for the plot. The plane’s normal determines which side of the plane is clipped away. The region on the side of the plane pointed to by the normal is the region that the Clip operator
clips away. If more than one plane is active, the region that is left as a result of the first clip operation is clipped by the
next plane, and so on.

Only one plane needs to be used to remove half of a plot. Find the center of the database by inspecting the 3D axis
annotations in the visualization window. Type the center as the new plane origin into the Origin text field for plane
1 then click the Plane 1 check box for plane 1 (see Figure 1.81). When the Apply button is clicked, half of the plot
should be removed. You can rotate the clipping plane by entering a new normal vector into the Normal text field. The
normal is specified by three floating point values separated by spaces.

The Accurate option can be used when multiple planes are specified, to ensure accuracy when planes intersect a zone
but do not clip the vertices. It can be up to 6x slower than the Fast option.

Removing one quarter of a plot

To remove a quarter of a plot, you need two clipping planes. To remove one of the plot, first remove one half of the
plot. Now, enable the second clipping plane and make sure that it has the same origin as the first clipping plane but
a different normal. To remove exactly one quarter of the plot, make sure that the normal is perpendicular to plane
1’s normal. Also make sure that plane 2’s new normal points into the region that was clipped away by plane 1. The
two planes, when considered together, remove one quarter of the plot. For an illustration of this, see Figure 1.82. In
general, the Clip operator removes regions defined by the intersection of the regions removed by each clipping plane.
Follow the same procedure with the third clipping plane to remove only one eighth of the plot.

Spherical clipping

The Clip operator not only uses sets of planes to clip databases, it can also use a sphere. To make the Clip operator use
a clipping sphere, click on the Sphere tab. To specify the location and size of the sphere, enter a new center location
into the Center text field on the Sphere tab of the Clip attributes window and then enter a new sphere radius.

Inverting the clipped region

Once the Clip operator has been applied to plots and a region has been clipped away, clicking the Inverse check box
brings back the clipped region and clips away the region that was previously unclipped. Using the Inverse check box
is an easy way to get only the clipped region back so it can be used for other operations.

A common trick when creating animations is to have two identical plots with identical Clip operators applied and then
switch one Clip operator to have an inverted clipping region. This will make the plot appear whole. The plot with the
Fig. 1.81: Clip attributes window

Fig. 1.82: Removing one quarter of a plot using two clip planes: Plane1 clipped region + Plane2 clipped region = One quarter removed
inverted clipping region can then be transformed independently of the first plot so it appears to slide out of the first plot. Then it is common to fade out the second plot and zoom in on the first plot’s clipped region.

**Cone operator**

Like the Slice operator, the Cone operator is also a slice operator. The Cone operator slices a 3D database with a cone, creating a surface that can be left in 3D or be projected to 2D. Plots to which the Cone operator has been applied become surfaces that exist on the surface of the specified cone. The resulting plot can be left in 3D space or it can be projected to 2D space where other operations can be done to it. A Pseudocolor plot to which a Cone operator has been applied is shown in Figure 1.83.

![Figure 1.83: Cone operator example: original plot; sliced with cone; sliced with cone and projected to 2D](image)

**Specifying the slice cone**

You can specify the slice cone by setting various fields in the Cone attributes window, shown in Figure 1.84. To specify how pointy the cone should be, type a new angle (in degrees) into the Angle text field. The cone is defined relative to its origin, which is the point at the tip of the cone. To move the cone, type in a new origin vector into the Origin text field. The origin is represented by three floating point numbers separated by spaces. Once the cone is positioned, you can set its direction (where the cone points) by entering a new direction vector into the Direction text field.

The cone can extend forever or it can be clipped at some distance along its length. To clip the cone at a certain length, check the Cut cone off check box and enter a new length value into the Length text field.

**Projecting the slice to 2D**

The Cone operator usually flattens sliced plots to 2D along the cone’s direction vector. This results in circular 2D plots in the visualization window. The Cone operator can also unfold sliced plots into a cylinder and then into rectangular 2D plots. Alternatively, the Cone operator can leave the sliced plots in 3D space where their cone shape is obvious. To set the cone projection mode, click on one of the following radio buttons: In 3D, Project to 2D, or Cylindrical.

**Connected Components operator**

The Connected Components operator is in a special class of operators, one that creates a new variable. In this case, the operator accepts as an input variable the name of a mesh, and constructs a scalar variable as output.
The operator creates unique labels for each connected mesh sub-component and tags each zone in the mesh with the label of the connected component it belongs to. Figure 1.85.

The operator has one option which controls the use of Ghost Zone Neighbors for connectivity between domains. This option is turned on (set to true) by default. Figure 1.86

### Cylinder operator

The Cylinder operator, shown in Figure 1.87, slices a dataset with a cylinder whose size and orientation are specified by the user. The result is a cylindrical surface.

#### Setting the cylinder’s endpoints

There are two ways to set the endpoints for the Cylinder operator. First of all, you can open the Cylinder operator window (see Figure 1.88) and type new 3D points into the Endpoint 1 and Endpoint 2 text fields. The second, and more interactive way to set the endpoints for the Cylinder operator is to use VisIt’s interactive Line tool, which is discussed in the Interactive Tools chapter. The Line tool lets you interactively place the Cylinder operator’s endpoints anywhere in the visualization. The Line tool’s endpoints correspond to the centers of the cylinder’s top and bottom circular faces.

#### Setting the radius

To set the radius used for the Cylinder operator’s clipping cylinder, type a new radius into the Radius text field in the Cylinder attributes window.

#### Inverting the cylinder region

Once the Cylinder operator has been applied to plots and a cylindrical region has been clipped away, clicking the Inverse check box brings back the cylindrical region and removes the region that was previously shown.
Fig. 1.85: Connected Components operator shown with Pseudocolor Plot.

Fig. 1.86: Connected Components operator window.
Fig. 1.87: Cylinder operator example: original plot; plot clipped by cylinder

Fig. 1.88: Cylinder operator window.
Decimate operator

The Decimate operator, shown in Figure 1.90, removes nodes and cells from an input mesh, reducing the cell count while trying to maintain the overall shape of the original mesh. The Decimate operator can currently operate only on the external surfaces of the input geometry. This means that in order to apply the Decimate operator, you must first apply the ExternalSurface operator, which will be covered later in this chapter. The Decimate operator is not enabled by default but it can be turned on in the Plugin Manager Window.

Using the Decimate operator

The Decimate operator simplifies mesh geometry. This can be useful for producing models that have lower polygon counts than the model before the Decimate operator was applied. Models with lower polygon count can be useful for speeding up operations such as rendering. The Decimate operator has a single knob that influences how many cells are removed from the input mesh. The Target Reduction value is a floating point number in the range (0,1) and it can be set in the Decimate attributes window (see Figure 1.91). The number specified is the proportion of number of polygonal cells in the output dataset “over” the number of polygonal cells in the original dataset. As shown in Figure 1.90, higher values for Target Reduction value cause VisIt to simplify the mesh even more.
Fig. 1.90: Decimate operator applied to reduce the number of cells in the mesh. (Left-to-right, top-to-bottom): Original Mesh, Reduction = 0.1, Reduction = 0.5, Reduction = 0.75

Fig. 1.91: Decimate attributes window

**DeferExpression operator**

The DeferExpression operator is a special-purpose operator that defers expression execution until later in VisIt’s pipeline execution cycle. This means that instead of expression execution taking place before any operators are applied, expression execution can instead take place after operators have been applied.
Plotting surface normals

VisIt can use the DeferExpression operator in conjunction with the ExternalSurface operator and the surface_normal expression to plot surface normals for your plot geometry. To plot surface normals, first create a vector expression using the surface_normal expression, which takes the name of your plot’s mesh as an input argument. Once you have done that, you can create a Vector plot of the new expression. Be sure to apply the ExternalSurface operator first to convert the plot’s 2D cells or 3D cells into polygonal geometry that can be used in the surface_normal expression. Finally, apply the DeferExpression operator and set its variable to your new vector expression. This will ensure that the surface_normal expression is not evaluated until after the ExternalSurface operator has been applied.

Displace operator

The Displace operator deforms a mesh variable using a vector field that is defined on the nodes of that mesh. Many engineering simulation codes write a mesh for the first time state of the simulation and then write vector displacements for the mesh for subsequent time states. The Displace operator makes it possible to use the mesh and the time-varying vector field to observe the behavior of the mesh over time. The Displace operator provides a multiplier that can amplify the effects of the vector field on the mesh so slight changes in the vector field can be exaggerated. An example showing a mesh and a vector field, along with the results of the mesh displaced by the vector field is shown in Figure 1.93.

Fig. 1.92: DeferExpression operator example

Fig. 1.93: Mesh and Vector plots and a Mesh plot that uses the Displace operator to deform the mesh using a vector field.
Using the Displace operator

The Displace operator takes as inputs a mesh variable and a vector variable and a displacement multiplier value. For each node in the mesh, the Displace operator adds the vector field defined at that node to the node’s coordinates. Before adding the vector to the mesh, VisIt multiplies the vector by the displacement multiplier so the effects of the vector field can be exaggerated. To set a new value for the displacement multiplier, type a new value into the Displacement multiplier text field in the Displace attributes window (see Figure 1.94). To set the name of the vector variable that VisIt uses to displace the mesh, select a new vector variable from the Displacement variable variable button.

Fig. 1.94: Displace attributes window

Elevate operator

The Elevate operator uses a scalar field on a 2D mesh to elevate each node in the input mesh, resulting in a topologically 2D surface in 3D. The Elevate operator allows you to perform much of the same functionality as a Surface plot and it allows you to do additional things like elevate plots that do not accept scalar variables. The Elevate operator can also elevate plots whose input data was produced from higher dimensional data that has been sliced. Furthermore, the Elevate operator allows you to display multiple scalar fields in a single plot such as when a Pseudocolor plot of scalar variable A is elevated by scalar variable B (see: Figure 1.95).

Fig. 1.95: Elevate operator example: 2D plot of rainfall; 2D plot of elevation; Plot of rainfall elevated by elevation

Using the Elevate operator

The Elevate operator can be used to create plots that look much like a Surface plot if you simply apply the Elevate operator to a plot that accepts scalar values. The Elevate operator is more flexible than a Surface plot because whereas
the Surface plot limits you to elevating by one variable and coloring by the same variable, the Elevate operator can be used with any plot and still achieve the Surface plot’s elevated effect. You could use the Elevate operator to elevate a Pseudocolor plot of rainfall by elevation. You could also take Vector or FilledBoundary plots (among others) and elevate them by a scalar variable.

Since the Elevate operator uses a scalar variable to elevate all of the points in the mesh, the Elevate operator has a number of controls related to scaling scalar data. For example, the Elevate operator allows you to artificially set minimum and maximum values for the scalar variable so you can eliminate data that might otherwise cause your elevated plot to be stretched undesirably in the Z direction. To set minimum and maximum values for the Elevate operator, click on the Min or Max check boxes in the Elevate attributes window (see Figure 1.96) and type new values into the adjacent text fields. The options for scaling the plots created using the Elevate operator are the same as those for scaling Surface plots. For more information on scaling, see the Surface plot documentation.

The most useful feature of the Elevate operator is its ability to elevate plots using an arbitrary scalar variable. By default, the Elevate operator uses the plotted variable in order to elevate the plot’s mesh. This only works when the plotted variable is a scalar variable. When you apply the Elevate operator to plots that do not accept scalar variables, the Elevate operator will fail unless you choose a specific scalar variable using the Elevate by Variable variable menu in the Elevate attributes window.

**Changing elevation height**

The Elevate operator uses a scalar variable’s data values as the Z component when converting a mesh’s 2D coordinates into 3D coordinates. When the scalar variable’s data extents are small relative to the mesh’s X and Y extents then you often get what appears to be a flat 2D version of the data floating in 3D space. It is sometimes necessary to scale the scalar variable’s data extents relative to the spatial extents in order to produce a visualization where the Z value differs noticeably. If you want to exaggerate the Z values that the scalar variable contributes to make differences more obvious, you can click on the Elevation height relative to XY limits check box in the Elevate attributes window.
The Elevate operator can be used to simply place a 2D plot in 3D space by use of the **Elevate with zero height** option. This will assign a value of zero to all of the z coordinates when converting into 3D.

**Explode operator**

The Explode operator has three primary targets, which are **materials, domains, and cells**. There are three different origins of explosion—**point, plane, and cylinder**—all of which have unique results and can be applied to any of the above mentioned targets. While this operator is primarily meant to be used on datasets containing materials or domains, the capability of exploding all cells remains available for datasets that lack either.

**Using the Explode operator**

The Explode operator has three areas for user definition. These are the **Origin** of explosion, **Material Explosion** settings, and **Cell Explosion** settings. You can add as many explosions as you’d like to a single instance of the operator, and you have the ability to **Add, Remove, or Update** explosions through the **Explode attributes window** shown below.
Fig. 1.99: Explode operator example: original plot; exploding cells of a material; exploding materials.

Fig. 1.100: Explode attributes window
Explode origin

As mentioned earlier, there are three different choices for an explode Origin. To explode from a Point, click the tab labeled Point in the Origin section of the Explode attributes window. You will then have the opportunity to enter a 3D coordinate defining your point. Similarly, to explode from a Plane, you must click on the Plane tab. You will then have the option to define a plane by a point located on that plane and the plane’s normal. Lastly, to explode from a Cylinder, first click on the Cylinder tab, and then enter two points that lie on a line traveling through the center (lengthwise) of your cylinder. By default, the cylinder has a radius of zero and is treated as a line to explode from. If you do define a positive radius, any data that is located within that radius will not be exploded when executing this explosion.

Exploding materials

Exploding a material results in an individual material within a dataset being displaced by a specified Factor from a specified origin. Both the factor with which the material is displaced and the actual material to be acted upon are set within the Material Explosion section of the attributes window. If you refer to the far right image in Figure 1.99, you will find an example of two material explosions. In this example, we see the materials Cord and Steel, shown in blue and green, being exploded from the Tire dataset.

Exploding domains

To explode the domains of a dataset, you must first make sure that your dataset has domains that can be plotted using the Subset plot. If this condition is met, all you need to do is apply the Explode operator to a Subset plot of your domains. The domains will then be substituted in for materials and treated as such. You can then refer to the section on exploding materials for usage tips.

Exploding cells

Exploding cells results in the separation and displacement of the cells within your dataset. This can either be applied to an individual material or the entire dataset. If you refer to the middle image in Figure 1.99, you will see the cells of the material Rubber, shown in red, being exploded by a plane. As a result, the material is split open and separated to allow us to see the inner contents. As before, you also have control over the explosion Factor that is applied to the cells. Additionally, you have two options for the Explosion Pattern. The first option is to explode through Impact, which results in cells that are closest to the origin being displaced furthest from the origin. The second option is to explode through Scatter, which results in cells furthest from the origin being displaced furthest from the origin.

ExternalSurface operator

The ExternalSurface operator takes the input mesh and calculates its external faces and outputs polygonal data. The ExternalSurface operator is not enabled by default but it can be turned on in the Plugin Manager Window. The ExternalSurface operator can be useful when creating plots that only involve the external geometry of a plot - such as when you create a Vector plot of surface normals.

Index Select operator

The Index Select operator selects a subset of a 2D or 3D structured mesh based on ranges of cell indices. Structured meshes have an implied connectivity that allows each cell in the mesh to be specified by an i,j or i,j,k index depending on the dimension of the mesh. The Index Select operator allows you to specify different ranges for each mesh dimension. The ranges are used to select a brick of cells from the mesh. In addition to indices, the Index Select operator uses
stride to select cells from the mesh. Stride is a value that allows the operator to count by 2’s, 3’s, etc. when iterating through the range indices. Stride is set to 1 by default. When higher values are used, the resulting mesh is more coarse since it contains fewer cells in each dimension. The Index Select operator attempts to preserve the size of the mesh when non-unity stride values are used. An example of the Index Select operator appears in Figure 1.102.

Fig. 1.102: Index Select operator example: original plot; index selected (stride=1); index selected (stride=2)

Setting a selection range

The Index Select attributes window, shown in Figure 1.103, contains nine spin boxes that allow you to enter minimum and maximum ranges for i,j,k. To select all cells in the X dimension whose index is greater than 10, you would enter 10 into the spin box in the I row and Min column. Then you would enter max into the spin box in the Max column in the I row. Finally, you would enter a stride of 1 into the spin box in the Incr column in the I row. If you wanted to sub-select cell ranges for the Y dimension, you could follow a similar procedure using the spin boxes in the J row and so forth. To set a range, first select the maximum number of dimensions to which the Index Select operator will apply. To set the dimension, click on the 1D, 2D, 3D radio buttons. Note that if the chosen number of dimensions is larger than the number of dimensions in the database, the extra dimension ranges are ignored. It is generally best to select the same number of dimensions as the database. The three range text fields are listed in i,j,k order from top to bottom. To restrict the number of cells in the X-dimension, use spin boxes in the I row. To restrict the number of cells in the Y-dimension, use the spin boxes in the J row. To restrict the number of cells in the Z-dimension, use the spin boxes in the K row.
Restricting to a subset of the whole database

Some databases are composed of multiple groups of meshes, which are often called groups or blocks. Some databases are composed of multiple meshes, often called blocks or domains. Some are composed of both groups and domains. When examining a database, you might want to look at only one block or group at a time. By default, the Index Select operator is applied to all blocks in the database. This means that each index range is applied to each block in the database and will probably result in an image featuring several small chunks of cells. When the Index select operator is set to apply to just one block or group, the index ranges are relative to the specified block or group.

To make the Index Select operator apply to just one block or group, uncheck the Use Whole Collection check box. The Category and Set combo boxes will be filled according to how the database has named the groups or sub-meshes. Choose the correct category from the Category combo box, and the desired set from the Set combo box. Figure ?? shows a single mesh selection for a multiple mesh database whose sub-meshes are called domains.

InverseGhostZone operator

The InverseGhostZone operator makes ghost cells visible and removes real cells from the dataset so plots to which the InverseGhostZone operator have been applied show only the mesh’s ghost cells. Ghost cells are a layer of cells around the mesh that usually correspond to real cells in an adjacent mesh when the whole mesh has been decomposed into smaller domains. Ghost cells are frequently used to ensure continuity between domains for operations like contouring. The InverseGhostZone operator is useful for debugging ghost cell placement in simulation data and for database reader plugins under development.

The InverseGhostZone operator’s attributes window (Figure 1.105) has various Show options allowing you to select which types of ghost cells are returned. By default all options are turned on.

Isosurface operator

The Isosurface operator extracts surfaces from 2D or 3D databases and allows them to be plotted. The Isosurface operator takes as input a database and a list of values and creates a set of isosurfaces through the database. An
Fig. 1.104: InversetGhostZone example

Fig. 1.105: InversetGhostZone window
isosurface is a surface where every point on the surface has the same data value. You can use an isosurface to see a
surface through cells that contain a certain value. The Isosurface operator performs essentially the same visualization
operation as the Contour plot, but it allows the resulting data to be used in VisIt’s other plots. For example, an
Isosurface operator can be applied to a Pseudocolor plot where the Isosurface variable is different from the Pseudocolor
variable. In that case, not only are the isosurfaces shown, but they are colored by another variable. An example of the
Isosurface operator is shown in Figure 1.106.

Setting isosurface levels

By default, VisIt constructs 10 levels into which the data fall. These levels are linearly interpolated values between the
data minimum and data maximum. However, you can set your own number of levels, specify the levels you want to
see or indicate the percentages for the levels.

To choose how levels are specified, make a selection from the Select by menu. The available options are: N levels,
Levels, and Percent. N levels, the default method, allows you to specify the number of levels that will be generated,
with 10 being the default. Levels requires you to specify real numbers for the levels you want to see. Percent takes a
list of percentages like 50.5 60 40. Using the numbers just mentioned, the first isosurface would be placed at the value
which is 50.5% of the way between the minimum and maximum data values. The next isosurface would be placed at
the value that is 60% of the way between the minimum and maximum data values, and so forth. You specify all values
for setting the number of isosurfaces by typing into the text field to the right of the Select by menu.
Setting Limits

The Isosurface attributes window, shown in Figure 1.107, provides controls that allow you to specify artificial minima and maxima for the data in the plot. You might set limits when you have a small range of values that you are interested in and you only want the isosurfaces to be generated through that range. To set the minimum value, click the Minimum check box to enable the Minimum text field and then type a new minimum value into the text field. To set the maximum value, click the Maximum check box to enable the Maximum text field and then type a new maximum value into the text field. Note that either the minimum, maximum or both can be specified. If neither minimum nor maximum values are specified, VisIt uses the minimum and maximum values in the dataset.

Scaling

The Isosurface operator typically creates isosurfaces through a range of values by linearly interpolating to the next value. You can also change scales so a logarithmic function is used to get the list of isosurface values through the specified range. To change the scale, click either the Linear or Log radio buttons in the Isosurface attributes window.

Setting the isosurfacing variable

The Isosurface operator database variable can differ from the plotted variable. This enables plots to combine information from two variables by having isosurfaces of one variable and then coloring the resulting surfaces by another variable. You can change the isosurfacing variable, by selecting a new variable name from the Variable variable button.

Sometimes it is useful to set the isosurfacing variable when the plotted variable is not a scalar. For example, you might want to apply the Isosurface operator to a Mesh plot but the Mesh plot’s plotted variable is not a scalar so the Isosurface operator does not know what to do. To avoid this situation, you can set the isosurfacing variable to one you know to be scalar and the operator will succeed.

Isovolume operator

The Isovolume operator creates a new unstructured mesh using only cells and parts of cells from the original mesh that are within the specified data range for a variable. The resulting mesh can be used in other VisIt plots. You might use this operator when searching for cells that have certain values. The Isovolume operator can either use the plotted variable or a variable other than the plotted variable. For instance, you might want to see a Pseudocolor plot of pressure...
while using the Isovolume operator to remove all cells and parts of cells below a certain density. An example of a plot to which an Isovolume operator has been applied is shown in.

**Using the Isovolume operator**

![Isovolume Attributes Window](image)

The Isovolume operator iterates over every cell in a mesh and determines which parts of the cell, if any, contain a value that falls within a specified data range. If any parts of the cell are within the specified data range, they are kept as part of the operator’s output. The Isovolume operator uses an isosurfacings algorithm to determine the interfaces where cells should be split so the interfaces for neighboring cells are all continuous and fairly smooth. To specify a data range, type new upper and lower bounds into the **Lower bound** and **Upper bound** text fields in the **Isovolume Attributes Window**, which is shown in Figure 1.109.

The variable that the Isovolume operator uses does not necessarily have to match the plotted variable. If the plotted variable is to be used, the **Variable** text field must contain the word: default. If you want to make the Isovolume operator use a different variable so you can, for example, plot temperature but only look at regions that have a density greater than 2g/mL, you can set the Isovolume’s variable to temperature. To make the Isovolume operator use a different variable, select a new variable from the **Variable** variable button in the **Isovolume Attributes Window**.

If you apply this operator to a plot that does not operator on scalar variables such as the Mesh or Subset plots, be sure to set the variable because the default variables for those plots is never a scalar variable. Without a scalar variable, the Isovolume operator will not work.

**Lineout operator**

The Lineout operator samples data values along a line, producing a 1D database from databases of greater dimension. This operator is used implicitly by VisIt’s Lineout capability and cannot be added to plots. For more information on Lineout, see the Lineout section in the Quantitative Analysis chapter.

**Merge operator**

VisIt’s Merge operator merges all geometry that may exist on separate processors into a single geometry dataset on a single processor. The Merge operator can be useful when applying other operators like the Decimate operator or when creating Streamline plots. The Merge operator is not enabled by default.

**OnionPeel operator**

The OnionPeel operator creates a new unstructured mesh by taking a seed cell or node from a mesh and progressively adds more layers made up of the initial cell’s neighboring cells. The resulting mesh is then plotted using any of VisIt’s standard plots. The OnionPeel operator is often useful for debugging problems with scientific simulation codes, which
often indicate error conditions for certain cells in the simulated model. Armed with the cell number that caused the simulation to develop problems, the user can visualize the simulation output in VisIt and examine the bad cell using the OnionPeel operator. The OnionPeel operator takes a cell index or a node index as a seed from which to start growing layers. Only the seed is shown initially but as you increase the number of layers, more of the cells around the seed are added to the visualization. An example of the OnionPeel operator is shown in Figure 1.110.

![Fig. 1.110: Onion peel operator example](image)

### Setting the seed

The OnionPeel operator uses a seed cell or a seed node as the seed to which all cells from other layers are added. When a layer is added around the seed, the new cells are those immediately connected to the seed. You specify the seed as a cell index or a node index by typing a new seed value into the Seed# or ij[k] text field. VisIt interprets the seed as a cell index by default. If you want to start growing cell layers around a given node, click on the Node radio button before entering a new seed value. The form of the seed index depends on how the underlying mesh is organized. Unstructured meshes, which are a collection of independent cells, require only a single integer value for the seed while structured meshes are indexed with i,j or i,j,k indices depending on the dimension of the mesh. To set the seed using i,j,k indices, type the i and j and k indices, separated by spaces, into the Seed# or ij[k] text field.

Some meshes that have been decomposed into multiple smaller meshes known blocks or domains have an auxiliary set of cell indices and node indices that allow cells and nodes from any of the domains to be addressed as though each domain was part of a single, larger whole. If you have such a mesh and want to specify seed indices in terms of global cell indices or global node indices, be sure to turn on the Seed# is Global check box.

The OnionPeel operator can only operate on one domain at a time and when the operator grows layers, they do not cross domain boundaries. The seed cell index is always relative to the active domain. To make a cell in a different domain the new seed cell, change the domain number by selecting a new domain from the Set combo box.

### Growing layers

The OnionPeel operator starts with a seed and adds layers of new cells around that seed. The added cells are determined by the layer number and the adjacency information. The cell adjacency rule determines the connectivity between cells.
Cells are next to each other if they share a cell face or a cell node. The visualization will differ slightly depending on which adjacency rule is used. To change the adjacency rule, click the Node or the Face radio buttons in the OnionPeel attributes window, shown in Figure 1.111.

The OnionPeel operator initially shows zero layers out from the seed, so only the seed is shown in the visualization when the OnionPeel operator is first applied. Consequently, the visualization might appear to be empty since some seed cells are very small. To add more layers around the seed, enter a larger layer number into the Layer Number text field. Clicking the up or down buttons next to the Layer Number text field also increments or decrements the layer number.

By default, Onion Peel will honor the structure of the original mesh. In some cases, as with arbitrary polyhedral data, you may want to see how VisIt split the original mesh. In this case, use the combo box to change to Honor actual mesh.

Project operator

The Project operator sets all of the Z values in the coordinates of a 3D mesh to zero and reduces the topological dimension of the mesh by 1. The Project operator is, in essence, an operator to make 2D meshes out of 3D meshes. An example of the Project operator is shown in Figure 1.112.

Setting the projection type

The Project operator can project 3D down to 2D using either Cartesian or Cylindrical transforms, which can be performed along the X, Y or Z axis, as shown in (see Figure 1.113). To specify which of these transforms you want to use when using the Project operator, choose the appropriate option from the Projection type combo box. Z-Axis Cartesian is the default option.
Fig. 1.112: Project Operator Example

Fig. 1.113: Project Attributes Window showing available projection types
Choosing how vectors are treated

The Project operator can treat vectors as instantaneous directions, as coordinate displacements or as point coordinates. The Project operator can also ignore the vectors and not transform them at all. To specify how you wish vectors to be treated during the projection transform, choose the appropriate option from the Vector transform method combo box. (see Figure 1.114) The default is Treat as instantaneous directions.

![Project Attributes Window showing available vector treatments](image)

Fig. 1.114: Project Attributes Window showing available vector treatments

Reflect operator

Use the Reflect operator to reflect database geometry across one or more axes. Scientific simulations often rely on symmetry so they only need to simulate part of the problem. When creating a visualization, most users want to see the entire object that was simulated. This often involves reflecting the database geometry to create the full geometry of the simulated object. VisIt’s Reflect operator can be applied to both 2D and 3D databases and can reflect them across one or more plot axes. An example of the Reflect operator is shown in Figure 1.115.

![Reflect operator example](image)

Fig. 1.115: Reflect operator example

Setting the Reflect attribute window’s input mode

The Reflect attributes window, shown in Figure 1.116, has two input modes. One input mode is for 2D data, in which only reflection quadrants are shown, and the second input mode is for 3D data for which the window shows 3D octants. In either input mode, clicking on the brightly colored shapes turns on different reflections and in the 3D input
mode, clicking on the cyan arrow rotates the view so you can more easily get to reflections in the back. To set the input mode, click either the 2D or 3D radio buttons.

![Reflect attributes window](image)

**Fig. 1.116: Reflect attributes window**

### Setting the data octant

The Reflect operator assumes that the database being reflected resides in the +X+Y+Z octant when performing its reflections. Sometimes, due to the orientation of the database geometry, it is convenient to assume the geometry exists in another octant. To change the data octant, make a new selection from the **Original data octant** menu in the **Reflect attributes window**. The **Reflect attributes window** graphically depicts the original data octant as the octant that contains a sphere instead of a cube, which correspond only to reflections.

### Reflecting plots

Once the Reflect operator has been applied to plots, you must usually specify the direction in which the plots should be reflected. To set the plot direction, click on the glyphs below the **Original data octant** menu. The possible reflections are shown by cube and sphere glyphs. When a reflection is set to be on, the glyph in the octant or quadrant will be green or magenta. When a reflection is not on, its glyph is smaller and silver. To turn a reflection on or off, just click on its glyph. If the window is in its 3D input mode and you need to access octants in the back that are obscured by other octants, clicking on the cyan arrow will rotate the glyphs so the octants in the back will be more accessible.

### Reflection limits

Reflection limits determine the axes about which the database geometry is reflected. The Reflect attributes window has three reflection limits controls; one for each dimension. You will usually want to reflect plots using the dataset min value, which you set by clicking the **Use dataset min** radio button. When using the dataset min value to reflect plots, the reflected plots will touch along the reflected edge. You can also specify another axis of reflection. When using a custom axis of reflection, the reflected plots will not necessarily touch. This option, though not normally needed, can
produce interesting effects in animations. To specify a custom axis of reflection, click the Specify X, Specify Y, or Specify Z radio buttons and enter a new X, Y, or Z value into the appropriate text field.

Resample operator

The Resample operator extracts data from any input dataset in a uniform fashion, forming a new 2D or 3D rectilinear grid onto which the original dataset has been mapped. The Resample operator is useful in a variety of contexts such as downsampling a high resolution dataset (shown in Figure 1.117), rendering Constructive Solid Geometry (CSG) meshes, or mapping multiple datasets into a common grid for comparison purposes.

![Fig. 1.117: Resample operator example](image)

Resampling onto a rectilinear grid

Resampling a high resolution dataset onto a rectilinear grid is the most common use case for the Resample operator. When a Resample operator is applied to a plot, the Resample operator clips out any data values that are not within the operator’s bounding box. For the data that remains inside the bounding box, the operator samples it using the user-specified numbers of samples for the X, Y, and Z dimensions. The default for the Resample operator is to use the entire extents of the dataset. If you want to choose a smaller region, unselect the Resample Entire Extents checkbox and enter new bounding box information. The bounding box is specified by entering new start and end values for each dimension. For example, if you want to change the locations sampled in the X dimension then you could type new floating point values into the Start X and End X text fields. The same pattern applies to changing the locations sampled in the Y and Z dimensions. One difference between resampling 2D and 3D datasets is that 3D datasets must have the 3D resampling check box enabled to ensure that VisIt uses the user-specified Z-extents and number of samples in Z.

Samples for which there was no data in the original input dataset are provided with a default value that you can change by typing a new floating point number into the Value for uncovered regions text field.

Using Resample with CSG meshes

Constructive Solid Geometry (CSG) modeling is a method whereby complex models are built by adding and subtracting primitive objects such as spheres, cubes, cones, etc. When you plot a CSG mesh in VisIt, VisIt resamples the CSG mesh into discrete cells that can be processed as an unstructured mesh and plotted. The Resample operator can be used to tell VisIt the granularity at which the CSG mesh should be sampled, overriding the CSG mesh’s default sampling.
Fig. 1.118: Resample attributes window
Naturally, higher numbers of samples in the Resample operator produce a more faithful representation of the original CSG mesh. Figure 1.119 depicts a CSG model that contains a disc within a smooth ring. Note that as the number of samples in the Resample operator increases, the model becomes smoother and jagged edges start to disappear.

Fig. 1.119: The Resample operator can be used to control the resolution of CSG meshes. Resolution is increased from left to right.

**Resampling surfaces projected to 2D**

Sometimes is is useful to project complex surfaces into 2D and resample them onto a 2D mesh so queries and other analysis can be performed.

When you project a complex surface to 2D using the Project operator, all of a plot’s geometry remains and its Z coordinates are set to zero. This results in some areas where the plot is essentially crushed on top of itself, as shown in Figure 1.120. When resampling the plot onto a new 2D grid, these overlapping areas can be treated in three different ways. You can ensure that the top value is taken if you choose the random option by clicking on the random button in the Resolve ties button group. You can use a mask variable to decide ties by clicking on the largest or smallest buttons and by selecting an appropriate variable using the Variable to resolve ties menu.

When used in parallel, the resampled data is distributed across all processors. This can be changed by unselecting the checkbox.

You can also force the output data to be cell centered by selecting the Make output cell centered checkbox.

**Revolve operator**

The Revolve operator is for creating 3D geometry from 2D geometry by revolving the 2D about an axis. The Revolve operator is useful for incorporating 2D simulation data into a visualization along with existing 3D data. An example of the Revolve operator is shown in Figure 1.121.

**Using the Revolve operator**

To use the Revolve operator, the first thing to do is pick an axis of revolution. The axis of revolution is specified as a 3D vector in the Axis of revolution text field (see Figure 1.122) and serves as the axis about which your 2D geometry is revolved. If you want to revolve 2D geometry into 3D geometry without any holes in the middle, be sure to pick an axis of revolution that is incident with an edge of your 2D geometry. If you want 3D geometry where the initial 2D faces do not meet, be sure to specify start and stop angles in degrees in the Start angle and Stop angle text fields. Finally, the number of steps determines how many times the initial 2D geometry is revolved along the way from the
Fig. 1.120: Using the Resample operator to create a 2D projection

This plot shows the above plot with transparency enabled to highlight the darker blue areas where the plot has been crushed on top of itself.

Fig. 1.121: Revolve operator example
start angle to the stop angle. You can specify the number of steps by entering a new value into the **Number of steps** text field.

By default, VisIt will choose the axis of revolution based on mesh type, which is also determined automatically. You can specify the mesh type manually by selecting a radio button other than **Auto**. To specify the axis of revolution manually, uncheck the **Choose axis based on mesh type** checkbox.

![Revolve attributes window](image1.png)

**Fig. 1.122: Revolve attributes window**

**Slice operator**

This operator slices a 3D database with a plane that can have an arbitrary orientation. Plots to which the Slice operator has been applied are turned into 2D planar surfaces that are coplanar with the slice plane. The resulting plot can be left as a 2D slice in 3D space or it can be projected to 2D space where other operations can be done to it. A Pseudocolor plot to which a Slice operator has been applied is shown in Figure 1.123.

![Slice operator example](image2.png)

**Fig. 1.123: Slice operator example**
Positioning the slice plane

You can position the slice plane by setting the origin, normal, and up-axis vectors in the **Slice operator attributes window**, shown in Figure 1.124. The slice plane is specified using the origin-normal form of a plane where all that is needed to specify the plane are two vectors; the origin and the normal. The origin of the plane is a point in the slice plane. The normal vector is a vector that is perpendicular to the slice plane.

![Slice attributes window](image)

**Fig. 1.124: Slice attributes window**

VisIt allows the slice plane normal to be aligned to a specific axis or it can be set to any arbitrary vector. If you want the slice plane to be along any of the three axes, click the **X-Axis**, **Y-Axis**, or **Z-Axis** radio button. If you want to make a slice plane that does not align with the principle axes, click the **Arbitrary** or **Theta-Phi** radio button and then type a direction vector into the text field to the right of the radio button. The vector need not be normalized since VisIt will normalize the vector before using it.

The slice plane’s origin, which specifies the location of the slice plane, can be set five different ways. The middle of the **Slice attributes window**, or **Origin area** (see the Figures below), provides the necessary controls required to set the slice plane origin. The **Origin area** provides five radio buttons: **Point**, **Intercept**, **Percent**, **Zone**, and **Node**. Clicking on one of these radio buttons causes the **Origin area** to display the appropriate controls for setting the slice plane origin. To set the slice plane origin to a specific point, click the **Point** radio button in the **Origin area** and then type a new 3D point into the **Point** text field. To set the slice plane origin to a specific value along the principle slice axis (usually an orthogonal slice), click the **Intercept** radio button and then type a new value into the **Intercept** text field.

If you don’t know a good value to use for the intercept, consider using the percent slice mode. Percent slice mode, which is most often used for an orthogonal slice, allows you to slice along a particular axis using some percentage of the distance along that axis. For example, this allows you to see what the slice plane looks like if its origin is 50% of
the distance along the X-Axis. To set the origin using a percentage of the distance along an axis, click the Percent radio button and then type a new percentage value into the Percent text field or use the Percent slider.

Sometimes it is useful to slice through a particular zone or node. The Slice operator allows you to pick an origin for the slice plane so a specific zone or node lies in the slice plane. To make sure that a particular zone is sliced by the Slice operator, click on the Zone radio button and then enter the zone to be sliced into the Zone text field. Be sure to also enter the domain that contains the zone into the Domain text field if you are slicing a multi-domain database. If you want to make sure that the slice plane’s origin is at a specific node in a mesh, click the Node radio button and enter a new node number into the Node text field. Note that you must also specify a domain if you are slicing a multi-domain database. If the database contains multiple meshes, their will also be Mesh combo box option from which to choose the mesh to use, as seen in the Node example in Figure 1.125.

Use the up-axis vector when you want the slice plane to be projected to 2D. The up-axis vector is a vector that lies in the slice plane and defines a 2D coordinate system within the plane where the up-axis vector corresponds to the Y-axis. To change the up-axis vector, type a new 3D vector into the Direction text field in the Up Axis area of the window.

Positioning the slice plane using the Plane Tool

You can also position the slice plane using VisIt’s interactive plane tool. The plane tool, which is available in the visualization window’s popup menu, allows you to position a slice plane interactively using the mouse. The plane tool is an object in the visualization window that can be moved and rotated. When the plane tool is changed, it gives its new slice plane to the Slice operator if the operator is set to accept information interactively. To make sure that the Slice operator can accept a new slice plane from the plane tool, check the Interactive check box in the Slice attributes window. For more information about the plane tool, read the Interactive Tools chapter.

Projecting the slice to 2D

The Slice operator usually leaves sliced plots in 3D so you can position the slice with the plane tool. However, you might want the plot projected to 2D. When a sliced plot is projected to 2D, any 2D operation, like Lineout, can be applied to the plot. To project a plot to 2D, check the Project 2D check box in the Slice attributes window.
Smooth operator

The Smooth operator smooths a mesh to improve areas plagued by jagged edges or sharp peaks.

![Smooth operator example](image)

Fig. 1.126: Smooth operator example

Using the Smooth operator

The Smooth operator has a number of controls that can be used to tune mesh smoothness. One such control is the number of iterations, which controls the number of times the mesh relaxation algorithm is applied to the input mesh. Larger numbers of iterations will produce smoother meshes but will also take more time to compute. To change the number of iterations, type a new integer value into the **Maximum number of iterations** text field in the **Smooth attributes window** (see Figure 1.127). The relaxation factor is a floating point number in the range [0,1] and it controls how much the mesh is relaxed. Values near 1 produce a mesh that is very smooth relative to the input mesh. To use a new relaxation factor, type a floating point number into the **Relaxation Factor** text field. The **Maintain Features** check box allows you to tell VisIt to preserve sharp peaks in the mesh while still smoothing out most of the mesh. The angle in the **Feature Angle** text field determines which features are kept. Any mesh angles less than the feature angle are preserved while others are smoothed.

SphereSlice operator

The SphereSlice operator slices a 2D or 3D database with an arbitrary sphere. Plots to which the SphereSlice operator have been applied become 2D surfaces that are coincident with the surface of the slicing sphere. The resulting plots remain in 3D space. You can use the SphereSlice operator to slice objects to judge their deviation from being perfectly spherical. An example of the SphereSlice operator is shown in Figure 1.128.

Positioning and resizing the slice sphere

You can position the slice sphere by setting its origin in the **SphereSlice attributes window** shown in Figure 1.129. The slice sphere is specified by a center point and a radius. To change the slice sphere’s center, enter a new point into
Fig. 1.127: Smooth attributes

Fig. 1.128: SphereSlice operator example
the **Origin** text field. The origin is a 3D coordinate that is represented by three space-separated floating point numbers. To resize the sphere, enter a new radius number into the **Radius** text field.

![SphereSlice attributes window](image)

**Fig. 1.129: SphereSlice attributes window**

**Positioning the slice sphere using the Sphere tool**

You can also position the slice sphere using VisIt’s interactive sphere tool. The sphere tool, available in the visualization window’s popup menu, allows you to position and resize a slice sphere interactively using the mouse. The sphere tool is an object in the visualization window that can be moved and resized. When the sphere tool is changed, it gives its new slice sphere to the SphereSlice operator. For more information about the sphere tool, read the *Interactive Tools* chapter.

**ThreeSlice operator**

The ThreeSlice operator slices 3D databases using three axis-aligned slice planes and leaves the resulting planes in 3D where they can all be viewed at the same time. The ThreeSlice operator is meant primarily for quick visual exploration of 3D data where the internal features cannot be readily observed from the outside.

**Moving the ThreeSlice operator**

The ThreeSlice operator is controlled by moving its origin, which is the 3D point where all axis-aligned slice planes intersect. There are two ways to move the ThreeSlice operator’s origin. First, you can directly set the point that you want to use for the origin by entering new x, y, z values into the respective **X**, **Y**, **Z** text fields in the **ThreeSlice operator attributes window**, shown in Figure 1.131. You can also make sure that the **Interactive** toggle is turned on so you can use VisIt’s interactive Point tool to set the ThreeSlice operator’s origin. When you use the Point tool to set the origin for the ThreeSlice operator, the act of moving the Point tool sets the ThreeSlice operator’s origin and causes plots that use the ThreeSlice operator to be recalculated with the new origin. For more information about the point tool, read the *Interactive Tools* chapter.

**Threshold operator**

The Threshold operator extracts cells from 2D and 3D databases where the plotted variable falls into a specified range. The resulting database can be used in other VisIt plots. You might use this operator when searching for cells with certain values. One such example is searching for the cell with the minimum or maximum value for the plotted variable. The Threshold operator removes all cells that do not have values in the specified range, making it easy to spot cells with the desired values. The Threshold operator can also use variables other than the plotted variable, for instance, you might want to see a Pseudocolor plot of pressure while using the Threshold operator to remove all cells.
Fig. 1.130: ThreeSlice operator example

Fig. 1.131: ThreeSlice attributes window
below a certain density. By specifying a different threshold variable, it is possible to visualize different quantities over the subset of cells specified by the threshold variable and range. An example of the Threshold operator is shown in Figure 1.132.

Fig. 1.132: Threshold operator example

Setting the variable range

The Threshold operator uses a range of values to determine which cells from the database should be kept in the visualization. For the Default bounds input, you specify the range of values by lower and upper bounds on the threshold variable. Cells with values below the lower bound or with values above the upper bound are removed from the visualization. To specify a new lower bound, type a new number or the special keyword: min into the Threshold attributes window's (Figure 1.133) Lower bound text field. To specify a new upper bound, type a new number or the special keyword: max into the Upper bound text field.

For the Custom bounds input, you can specify a list of ranges in the Range text field. A colon - ':' defines a range and a comma - ',' defines a logical OR. The range shown in Figure 1.134 has the following meaning:

\[ 1 \leq \text{default} \leq 10 \text{ OR } \text{default} = 17 \text{ OR } 23 \leq \text{default} \leq \text{max} \]

Numbers, commas, and colons are the only valid symbols that can be used in specifying a range list.

When the threshold variable is a nodal quantity, the cell being considered by the Threshold operator has values at each node in the cell. In this case, the Threshold operator provides a control that determines whether or not to keep the cell if some nodes have values in the threshold range or if all nodes have values in the threshold range. More cells are usually removed from the visualization when all nodes must be in the threshold range. Select Part in range from the Show zone if combo box to allow cells where at least one value is in the threshold range into the visualization. Select All in range from the Show zone if combo box to require that all nodal values exist in the threshold range.

Setting the threshold variable

The Threshold operator uses the threshold variable to determine whether cells remain in the visualization. The threshold variable is usually the plotted variable in which case the Variable column displays: default. To specify a threshold variable other than the plotted variable, click on the Add variable variable button and select a new scalar variable from the list of available variables.

You might set the threshold variable when you apply the Threshold operator to plots which do not take scalar variables as input. An example of this is the Mesh plot. When you apply the Threshold operator to a Mesh plot, you must set the threshold variable to a valid scalar variable for cells to be removed from the plot. You can also use the threshold variable to remove cells based on one variable while viewing the plotted variable.
Fig. 1.133: Threshold attributes window - Default
Fig. 1.134: Threshold attributes window - Custom
Setting the output mesh type

The Threshold operator removes all cells that do not meet the threshold criterion, leaving behind a set of cells that are gathered into an unstructured mesh. Sometimes, it can be useful to transform the remaining cells into a point mesh. You can specify the desired output mesh type using the Cells from input and Point mesh radio buttons in the Threshold attributes window.

Transform operator

The Transform operator manipulates a 2D or 3D database’s coordinate field by applying rotation, scaling, and translation transformations. The operator’s transformations are applied in the following order: rotation, scaling, translation. The Transform operator is applied to databases before they are plotted. You might use the Transform operator to rotate database geometry to a more convenient orientation or to scale database geometry to make better use of the visualization window. You can also use the Transform operator to make objects rotate and move around the visualization window during animations. This works well when only one part of the visualization should move while other parts and the view remain fixed. An example of the Transform operator is shown in Figure 1.135.

Rotation

You can use the Transform operator to rotate plots around an arbitrary axis in 3D and around the Z-axis in 2D. To apply the rotation component of the Transform operator, be sure to check the Rotate check box in the Transform attributes window (Figure 1.136). An origin and normal are needed to specify the axis of rotation. The origin serves as a reference point for the object being rotated. The axis of rotation is a 3D vector that, along with the origin, determines the 3D axis that will serve as the axis of rotation. You must supply an origin and an axis vector to specify an axis of rotation. To change the origin, type a new 3D vector into the top Origin text field. To change the 3D axis, type a new 3D vector into the Axis text field. Both the origin and the axis are represented by three space-separated floating point numbers.

When applying the Transform operator to plots, you probably want to make the origin the same as the center of the plot extents which can be found by looking at the axis annotations. When the Transform operator is applied to 3D plots, the axis of rotation can be set to any unit vector. When the Transform operator is applied to 2D plots, the axis of rotation should always be set to the Z-axis (0 0 1).

Once you specify the axis of rotation, you must supply the angle of rotation. The default angle of rotation is zero degrees, which gives no rotation. To change the angle of rotation, enter a number in degrees or radians into the Amount text field and click the Deg radio button for degrees or the Rad radio button if the angle is measured in radians.
Fig. 1.136: Transform attributes window
Scale

You can use the Transform operator to scale plots. Each dimension can be scaled independently by entering a new scale factor into the X, Y, Z text fields. Each scale factor is a multiplier so that a value of 1 scales plots to their original size while a value of 2 scales plots to twice their original size. To apply the scale component of the Transform operator, be sure to check the Scale check box in the Transform attributes window. All dimensions are scaled relative to a scaling origin which can be changed by typing a new origin into the middle lower Origin text field.

Translation

You can use the Transform operator to translate plots. To apply the translation component of the Transform operator, be sure to check the Translate check box in the Transform attributes window. To translate plots in the X dimension, replace the default value of zero in the X translation text field. Translations in the Y and Z dimensions are handled in the same manner.

Coordinate system conversion

In addition to being able to rotate, scale, and translate plots, the Transform operator can also perform coordinate system conversions. A plot’s coordinates can be specified in terms of Cartesian, Cylindrical, or Spherical coordinates (illustrated in Figure 1.137). Ultimately, when a plot is rendered in the visualization window, its coordinates must be specified in terms of Cartesian coordinates due to the implementation of graphics hardware. If you have a database where the coordinates are not specified in terms of Cartesian coordinates, you can apply the Transform operator to perform a coordinate system transformation so the plot is rendered correctly in the visualization window.

![Fig. 1.137: Cartesian, Cylindrical, Spherical coordinate systems](image)

Figure 1.138 shows a model of an airplane that is specified in terms of spherical coordinates. When it is rendered initially, VisIt assumes that the coordinates are Cartesian, which leads to the plot getting stretched and tangled. The Transform operator was then applied to convert the plot’s spherical coordinates into Cartesian coordinates, which allows VisIt to draw the plot as it is intended to look.

The Transform operator allows coordinate system transformations between any of the three supported coordinate systems, shown in Figure 1.139. To pick a coordinate system transformation, you must first pick the coordinate system used for the input geometry. Next, you must pick the desired output coordinate system. In the example shown in Figure 1.138, the input coordinate system was Spherical and the output coordinate system was Cartesian. Note that if you use the Transform operator to perform a coordinate system transformation then you cannot also perform rotation, scaling, or translation. If you must perform any of those operations, add a second Transform operator to your plots.
1.4. Operators

Fig. 1.138: Coordinate system conversion using the Transform operator

Fig. 1.139: Supported coordinate systems
Linear transforms

Linear transforms can be specified via a 4x4 matrix as shown in Figure 1.140. Vectors will be transformed by default, uncheck the transform vectors checkbox if this is not desired. The inverse transform can be applied by selecting Invert linear transform.

![Linear transformation options](image)

Fig. 1.140: Linear transformation options

Tube operator

The Tube operator is an operator that turns line geometry into tubes, making the lines appear fatter and shaded.

Changing tube appearance

The Tube operator provides a few knobs that control the appearance of the generated tubes. First of all, the tube radius can be set by typing a new radius into the Radius text field in the Tube attributes window (Figure 1.142). The specified radius can either be a Fraction of Bounding Box (default) or Absolute by changing the combo box option next to the Radius text box. If you want the radius scaled by a variable instead, check the Scale width by variable? checkbox, and choose a variable from the Variable menu.

The number of polygons used to make up the circumference of the tube can be altered by typing a new number of sides into the Fineness of tube text field. Finally, the ends of tubes can be capped instead of remaining open by turning on the Cap Tubes check box. See Figure 1.143 for result of capping.
Fig. 1.141: Linear transformation example
Fig. 1.142: Tube attributes window

Fig. 1.143: Uncapped and capped tubes
1.5 Saving and Printing

In this chapter, we discuss how to save and print files from within VisIt. The section on saving files is further broken down into four main areas: saving session files, saving images, saving movies, saving Cinema databases, and exporting databases. We first learn about saving session files using the Save Session window. We then learn about saving images of visualizations using the Save Window and then we move on to saving movies and sets of image files using the Save movie wizard. In addition to movies, VisIt provides the Save to Cinema wizard to create Cinema image databases, which surpass movies and allow the user to explore data from different viewpoints. After learning to save images, movies, and Cinema databases, this chapter concentrates on exporting VisIt plots as databases using the Export Database window. Finally, we learn to print images of visualizations using the Printer Window.

1.5.1 Session files

A session file is an XML file that contains all of the necessary information to recreate the plots and visualization windows used in a VisIt session. You can set up complex visualizations, save a session file, and then run a new VisIt session later and be able to pick up exactly where you left off when you saved the session file. If you often look at the same types of plots with the same complex setup then you should save a session file for your visualization once it is set up so you don’t have to do any manual setup in the future.

Saving session

Once you have set up your plots, you can select Save session option in the Main Window’s File menu to open up a Save file dialog. Once the Save file dialog is opened, select the location and filename that you want to use to store the session file. By default, VisIt stores all session files in your .visit directory on UNIX and MacOS X computers and in the directory where VisIt was installed on Windows computers. Once you select the location and filename to use when saving the session file, VisIt writes an XML description of the complete state of all vis windows, plots, and GUI windows into the session file so the next time you come into VisIt, you can completely restore your VisIt session.

Restoring session

Restoring a VisIt session file deletes all plots, closes all databases, etc before VisIt reads the session file to get back to the state described in the session file. After restoring a session file, VisIt will look exactly like it did when the session file was saved. To restore a session file, click the Restore session option from the Main Window’s File menu to open an Open file dialog. Choose a session file to open using the Open file dialog. Once a file is chosen, VisIt restores the session using the selected session file. If you are on the Windows platform, you can double-click session files (.vses files) stored on your computer in order to directly open them with VisIt.

1.5.2 Saving the Visualization Window

VisIt allows you to save the contents of any open visualization window to a variety of file formats. You can save visualizations as images so they can be imported into presentations. Alternatively, you can save the geometry of the plots in the visualization window so it can be imported into other computer modeling and visualization programs.

VisIt currently supports the image files formats: BMP, JPEG, PNG, PPM, Raster Postscript, RGB, and TIFF
VisIt currently supports the geometry file formats: Curve, Alias Wavefront Obj, PL, POV, STL, ULTRA, and VTK

The Curve and ULTRA file formats are specially designed to store the data created from curve plots and can be used with other Lawrence Livermore National Laboratory visualization software. The Alias Wavefront Obj file format is supported so visualizations produced with VisIt can be imported into rendering programs such as Maya. VisIt can save visualizations into STL files, which are used with stereolithographic printers to fabricate three-dimensional parts.
Finally, VisIt can save visualizations into the VTK (Visualization Toolkit) format so they can be read back into VisIt and used in other VTK-based applications.

When saving the geometry of plots in the visualization window into any of the afore-mentioned formats, you are performing a type of database export operation. However, saving geometry in this manner differs from exporting databases using the Export Database Window. Only the external faces of the plots are saved out when saving plot geometry whereas during a database export, 3D cells are preserved in the final exported database. The topic of exporting databases is covered later in this chapter.

The Save Window

You can set the Save window options before saving by selecting Set Save options… from the Main Window’s File menu. The Set save options window contains the controls that allow you to set the options that govern how visualizations are saved.

The Set Save options window, shown in Figure 1.144, contains four basic groups of controls. The first group, File-name, allows you to set the file information. Use the file information controls to set the name and destination. If the Family checkbox is selected, then each time an image is saved with the same name, a number will be appended to the filename that is one more than the current file with the same name. The second group, Format options, allows you to set the file type, compression type, and any optional quality parameters that may exist for the selected file type. Use the third group of controls, Aspect ratio and resolution, to specify the dimensions of the saved image. If Screen capture is checked, the aspect ratio and width/height will be ignored and the current screen image will be saved. The last group, Multi-window save, allows you to set options for each window being saved by clicking on the Window drop-down and selecting the appropriate window. When the save options are set and applied by clicking the Apply button, the active visualization can be saved either through the Save Window option in the Main Window’s File menu, by the keyboard shortcut Ctrl+S, or by clicking the Save button in the Set Save options window.

Selecting the output directory for saved files

On most platforms, VisIt’s default behavior is to save output files to the current directory, which is the directory where VisIt was started. On the Windows platform, VisIt saves images to the "My images" directory, which is a directory under the VisIt installation directory. If you want to specify a special output directory for your output files, you can turn off the Output files to current directory check box and type in the path to the directory where you want VisIt to save your files in the Output directory text field. If you want to browse the file system to find a suitable directory in which to save your images, click on the “…” button to the right of the Output directory text field to bring up a Directory chooser dialog. Once you select a suitable directory using the Directory chooser dialog, the path that you chose is inserted into the Output directory text field.

Setting the save file name

To set the file name that will be used to save files, type a file name into the Filename text field. The file name that you use may contain a path to a directory where you want to write the saved files. If no path is specified, the saved files are written to the directory from which VisIt was launched. A file extension appropriate for the type of file being generated is automatically appended to the file name. For example, a BMP file will have a “.bmp” extension, while a JPEG file will have a “.jpeg” extension, and so on.

The file name that VisIt uses to save visualizations is based on the specified file name, the file format, and also the family toggle setting. The family toggle setting is set by checking the Family check box towards the top right part of the Save Window.

The family toggle setting allows you to save series of files that all have essentially the same name except for a number that is appended to the file name. The number increases by one each time an image is saved. If the family toggle setting is on then a file named “visit” of type TIFF will save out as “visit0000.tif”. If the family toggle setting is off, the file will save as “visit.tif”.

142 Chapter 1. VisIt GUI User Manual
Fig. 1.144: Save Window
Setting the file type

You set the file type by making a selection from the File type menu. You can choose from image file types or geometry file types. Note that some areas of the Save Window become enabled or disabled for certain file types.

Choosing JPEG format files enables the Quality slider and the Progressive check box. These controls allow you to specify the desired degree of quality in the resulting JPEG images. A lower quality setting results in blockier images that fit into smaller files. The progressive setting stores the JPEG images in such a way that they progressively refine as they are downloaded and displayed by Web browsers.

Choosing TIFF format files enables the Compression type combo box. The available compression types are: None, PackBits, JPEG, and Deflate. When compression is enabled for TIFF files, they are smaller than they would be without compression.

Choosing STL or VTK file formats saves visualizations as geometry files instead of images and also enables the Binary check box. The Binary check box tells these formats to write their geometry data as binary data files instead of human-readable ASCII text files. In general, files written with the binary option are smaller and faster to load than their non-binary counterparts.

Saving images with screen capture

The Screen capture check box tells VisIt to grab the image directly off of the computer screen. This means that the saved image will be exactly the same size as the image on the screen. There are advantages and disadvantages to using screen capture. An advantage is that capturing the image from the screen does not require VisIt to redraw the image to an internal buffer before saving, which usually results in a faster save. A disadvantage of screen capture is that any other windows on top of VisIt’s visualization window occlude portions of the image. Screen capture can also be very slow over a sluggish network connection. Finally, using screen capture might not provide images that have enough resolution. Weigh the advantages and disadvantages of using screen capture for your own situation. Screen capture is on by default.

Setting image resolution

You set image resolution using the controls in the Aspect ratio and resolution group. These controls are disabled unless the file being saved is an image format and screen capture is not being used. You specify the image height and width by typing new values into the Height and Width text fields. If the Maintain 1:1 aspect check box is on, VisIt forces the image’s height and width to be the same, yielding a square image. Turn off this setting if you want to save rectangular images. The image resolution is ignored unless you turn off the Screen capture check box.

Saving stereo images

When the Stereo check box is turned on and you save an image, VisIt will save a separate image for the left eye and for the right eye. The cameras used to generate each image are offset such that when the images are played together at high rates, they appear to have more depth. To enable saving of stereo images, click the Stereo check box in the Save Window before you try to save an image.

When Family mode is not enabled, VisIt will prepend left_ and right_ designators to the saved filenames. However, when Family mode is enabled, VisIt saves the two images in sequence without any left/right designation. The left image is saved first followed by the right image. If next available number in the Family is odd, the left will be odd and right will be even. On the other hand, if next available number in the Family is even, the left will be even and right will be odd. However, the notification messages VisIt produces about the saved images may only mention the first (left) saved image filename.
Saving binary geometry files

Some geometry file formats such as STL and VTK have both ASCII and binary versions of the file format. The ASCII file formats are human-readable and are larger and slower for programs to process than binary formats, which are not human-readable but are smaller and quicker for programs to read. When geometry file formats support both ASCII and binary formats, the Binary check box is enabled. By default VisIt writes ASCII geometry files but you can click the Binary check box to make VisIt write binary geometry files.

Selecting pixel data

Normally when saving an image, VisIt will simply save the RGB pixel data into the specified image format. It is possible to request that VisIt saves additional pixel data when saving an image. This may result in additional files being saved alongside the normal image file. These additional images will share the same filename root as the image file but will have suffixes such as “value”, “depth”, or “lum”, depending on their contents. Special file formats such as OpenEXR can contain all of these additional image channels. When OpenEXR is the selected file format, a single “.exr” file will be written containing all pixel data.

The Save options window contains a Pixel data group that lets you request additional image channels. The RGB check box selects RGB pixel data. The Alpha check box tells VisIt to also request transparency information and to not render with a background when saving an image. This lets VisIt save images with a transparent background, which makes compositing such an image in front of other backgrounds far easier (see Figure 1.145). The Depth check box tells VisIt to export the depth buffer (Z-buffer) to a ZLib-compressed binary file containing 32-bit floating point numbers. The Luminance check box tells VisIt to save a luminance image, which shows how much lighting is used in various parts of the scene. The luminance image is saved to the selected image format. The Value check box tells VisIt to produce a rendering of the actual scalar values in the scene in the form of a ZLib-compressed 32-bit floating point buffer (same format as the depth image).

![Figure 1.145: Partially transparent plot saved to PNG with alpha channel](image)
Saving tiled images

A tiled image is a large image that contains the images from all visualization windows that have plots. If you want to save tiled images, make sure to check the **Save tiled** check box in the **Set Save options** window. To get an idea of how VisIt saves your visualization windows into a tiled image, see Figure 1.146 and Figure 1.147.

1.5.3 Saving movies

In addition to allowing you to save images of your visualization window for the current time state, VisIt also allows you to save movies and sets of images for your visualizations that vary over time. There are multiple methods for saving movies with VisIt. This section introduces the Save movie wizard and explains how to use it to create movies from within VisIt’s GUI. The *Animation* chapter explains some auxiliary methods that can be used to create movies.

The **Save movie wizard** (see Figure 1.148) is available in the **Main Window’s Files** menu. The **Save movie wizard**’s purpose is to lead you through a set of simple questions that allow VisIt to gather the information required to create movies of your visualizations. For example, the **Save movie wizard** asks which image and movie formats you want to generate, where you want to store the movies, what you want to call the movies, etc. Each of these questions appears on a separate screen in the **Save movie wizard** and once you answer the question on the current screen, clicking the **Next (Continue for OSX)** button advances you to the next screen. You can cancel saving a movie at any time by clicking on the **Cancel** button. If you advance to the last screen in the **Save movie wizard** then you have successfully provided all of the required information that VisIt needs to make your movie. Clicking the **Finish** button at that point invokes VisIt’s movie-making script to make the movie. If you want to make subsequent movies, you can choose to use the settings for the movies that you just made or you can choose to create a new movie and provide new information.
Fig. 1.147: Saving tiled images example (after)
Fig. 1.148: Save movie wizard (screen 1)
Choosing movie formats

The **Save movie wizard**’s second screen, shown in Figure 1.149, allows you to pick the types of movies that you want to create. You can select as many image and movie formats as you want and you can even specify multiple resolutions of the same movie. VisIt allows you to order multiple versions of your movie because it is often easier to create different versions of the movie all at once as opposed to doing it later once it is discovered that you need a new version to play on a laptop computer or a tiled display wall.

The **Save movie wizard**’s second screen is divided vertically into two main areas. On the left you will find the **Format and resolution** area, which displays the format and resolution for the current movie. On the right, you will find the **Output** area, which lists the formats and resolutions for all of the movies that you have ordered. By default no movie formats are present in the **Output** area’s list of movies. You cannot proceed to the next screen until you add at least one movie format to the list of movies in the **Output** area.

To add a movie format to the list of movies in the **Output** area, first choose the desired movie format from the **Format** combo box in the **Format and resolution** area. Next, choose the movie resolution. The movie resolution can be specified in terms of the visualization window’s current size or it can be specified in absolute pixels. The default movie resolution uses the visualization window’s current size with a scale of 1. You can change the scale to shrink or grow the movie while keeping the visualization window’s current aspect ratio. If you want to specify an absolute pixel size for the movie, click on the **Specify movie size** radio button and type the desired movie width and height into the **Width** and **Height** text fields. Note that if you specify a width and height that causes the movie’s shape to differ from the visualization window’s shape, you might want to double-check that the view used for the visualization window’s plots does not change appreciably.

The **Save movie wizard** allows you to create stereo movies if you check the **Stereo movie** box and select a stereo type from the **Stereo type** drop-down menu. The default is to create non-stereo movies because stereo movies are not widely supported.
The only movie format that VisIt produces that is compatible with stereo movies is the “Streaming movie” format, which is an LLNL format commonly used for tiled displays. The “Streaming movie” format can support stereo movies where the image will flicker between left and right eye versions of the movie, causing a stereo effect if you view the movie using suitable liquid-crystal goggles. The stereo option has no effect when used with other movie formats. However, if you choose to save a stereo movie in any of VisIt’s supported image formats, VisIt will save images for the left eye and images for the right eye. You can then take the left and right images into your favorite stereo movie creation software to create your own stereo movie.

Once you have selected the desired movie format, width, and height, click on the right-arrow button that separates the Format and resolution area from the Output area. Clicking the right-arrow button adds your movie to the list of movies that you want to make. Once you have at least one movie in the Output area, the screen’s Next button will become active. Click the Next button to go to the next screen in the Save movie wizard.

Choosing movie length

![Choose length](image)

Fig. 1.150: Save movie wizard (screen 3)

It is possible to specify the range of time states to use for the movie, as well as specify a stride if you have too many time states saved (see Figure 1.150). The wizard will automatically set the range of time states.

Choosing the movie name

Once you have specified options that tell VisIt what kinds of movies that you want to make, you must provide the base name and location for your movies. By default, movies are saved to the directory in which you started VisIt. If you
Fig. 1.151: Save movie wizard (screen 4)
want to specify an alternate directory, you can either type in a new directory path into the **Output directory** text field (see Figure 1.151) or you can select a directory from the **Choose directory** dialog box activated by clicking on the “...” button.

The base filename for the movie is the name that is prepended to all of the movies that you generate. When generating multiple movies with differing resolutions, the movie resolution is often encoded into the filename. VisIt may generate many different movies with different names but they will all share the same base filename that you provided by typing into the **Base filename** text field.

### Choosing e-mail notification

![Save movie wizard (screen 5)](image)

If you want to be notified by e-mail when the movie creation is complete, then select the **Yes** option and enter the appropriate e-mail address (see Figure 1.152). By default, no e-mail notification is sent once the movie creation is complete.

### Choosing movie generation method

After all movie options are specified, VisIt prompts you how you would like your movie made. At this point, you can click the **Finish/Done** button to make VisIt start generating your movie. You can change how VisIt creates your movie by clicking a different movie generation method on the **Save movie wizard**'s sixth screen, shown in Figure 1.153.

The default option for movie creation allows VisIt to use your current VisIt session to make your movies. This has the advantage that it uses your current compute engine and allocated processors, which makes movie generation start
Fig. 1.153: Save movie wizard (screen 6)
immediately. When you use this movie generation method, VisIt will launch its command line interface (CLI) and execute Python movie-making scripts in order to generate your movie. This means that you have both the VisIt GUI and CLI controlling the viewer. If you use this movie generation method, you will be able to watch your movie as it is generated. You can track the movie’s progress using the Movie progress dialog, shown in Figure 1.154. The downside to using your currently allocated processors is that movie generation takes over your VisIt session until the movie is complete. If you want to regain control over your VisIt session, effectively cancelling the movie generation process, you can click the Movie progress dialog’s Cancel button.

![Movie progress dialog](image)

Fig. 1.154: Movie progress dialog

The second movie generation method will cause VisIt to save out a session file containing every detail about your visualization so it can be recreated by a new instance of VisIt. This method works well if you want to create a movie without sacrificing your current VisIt session but you cannot watch the movie as it is generated and you may have to wait for the second instance’s compute engine to be scheduled to run. The last movie generation option simply makes VisIt display the command that you would have to type at a command prompt in order to make VisIt generate a movie of your current visualizations.

### 1.5.4 Saving Cinema

VisIt lets you save Cinema databases in addition to saving images and movies of your plots. A Cinema database is an image-based proxy for large scale data that lets you explore the data using far fewer computational resources. Where post-processing full data might take a supercomputer, exploring a Cinema database can be done on a tablet. Cinema databases consist of images that are indexed by a JSON file or CSV file. The index file is used by the Cinema viewer (available at [www.cinemascience.org](http://www.cinemascience.org)) to determine a set of parameters that can be changed by the user. These parameters are used to look up corresponding image files for display in the Cinema viewer. For example, Cinema databases typically allow the user to navigate through time using a time parameter. Cinema databases also can be saved using a spherical camera that is described by phi and theta parameters to let the user see the plots from various camera angles. It is possible to create Cinema databases in situ using Libsim so Cinema databases can be created incrementally as a simulation runs. This section introduces the Save Cinema wizard and explains how to create Cinema databases from within VisIt’s GUI.

The Save Cinema wizard (see Figure 1.155) is available in the Main Window’s Files menu. The Save Cinema wizard’s purpose is to let you set the options that are used to take the current visualizations and produce a Cinema database. Progress through the screens using the Next button until the last screen is reached. Clicking Cancel at any time will close the wizard. Clicking the Finish button will tell VisIt to produce a Cinema database with the current settings.
Fig. 1.155: Save Cinema wizard (screen 1)
Choosing filename

Cinema databases are stored as a directory structure containing various nested directories with image files and an index file. When saving a Cinema database, you must pick the name of the top level directory under which all other files will be saved. The Save Cinema wizard contains a File name selection control that lets you select the name of the Cinema “.cdb” directory. The control can accept file names that are typed in and clicking the … button opens a filename selection window that permits a new filename to be selected.

Choosing specification

Cinema databases are described by specifications that dictate the format and allowable contents for the files that they contain. There are currently 3 Cinema specifications in use: A, C, D.

Specification A describes a Cinema database format that contains image files (PNG, TIFF, etc.) that are associated with various user-defined parameters such as time or camera angles in the case of a phi-theta camera. This specification is compatible with any of the VisIt plots since images of the currently set up visualizations are saved. Specification C describes a Cinema database format that adheres to a different directory structure over specification A and can contain composite images. Composite images are comprised of 3 separate files: a PNG file containing a luminance image, a ZLib-compressed file containing the Z-buffer, and a ZLib-compressed file containing a rendering of actual scalar values for the plot. Specification D is similar to specification A except that it uses a CSV file to associate image files with a set of parameters, enabling sparse sets of images.

The Save Cinema wizard contains a set of A, C, D radio buttons to let you choose the most appropriate specification for the type of Cinema database to be created.

Image settings

The Save Cinema wizard contains controls for image settings such as the file format, image width/height, and whether to use screen capture. The File format control lets you select the image file format to be used. Several pixel-based image file formats are available such as BMP, PNG, TIFF, and when available EXR. OpenEXR is a format from ILM that can store various image channels and data in multiple layers that can be composited later. Support for OpenEXR is optionally compiled into VisIt. The Width and Height controls allow the output image width and height to be specified when screen capture is not in use by setting the Use screen capture controls. This permits VisIt to save images in a custom size as opposed to saving images based on the current visualization window’s size. Note that using screen capture is faster for normal images since it does not require VisIt to re-render the visualizations.

Composite images

Specification C Cinema databases support saving composite images which consist of a luminance image, a Z image, and a scalar image. The luminance image is a gray scale image that indicates the lighting used in the scene and it is saved as a PNG image or other pixel format image. The Z image is contains the Z-buffer for the luminance image, stored as a buffer of 32-bit floating point values that have been ZLib-compressed and written to a raw binary file. The scalar image is stored the same as the Z buffer image but it contains float values that correspond to the actual scalars that were rendered in the visualization. The scalar values are used in the Cinema viewer to dynamically recolor the scene at render time. Composite images are most appropriate for surface-based VisIt plots that employ a continuous color table, such as the Pseudocolor plot. Composite images can be enabled by turning on the Create composite images check box in the Save Cinema wizard when specification C is used. When this setting is in effect, each VisIt plot will be saved to a separate “layer” in the Cinema database so it can be composited into the scene at will. Figure 1.157 shows multiple VisIt plots that have been saved as separate layers to a composite image specification C Cinema database that enables layers to be turned on and off at view time.
Fig. 1.156: Cinema viewer with composite layers
Choosing Camera type

Cinema databases support multiple camera types. VisIt’s Cinema export supports static cameras and phi-theta cameras. A static camera corresponds to the view that is currently in effect in the visualization and when it is used, all time states in the Cinema database will be viewed from that camera orientation. A phi-theta camera defines 2 angles, phi and theta, that define the view direction as in a spherical coordinate system. When a phi-theta camera is used, the Cinema export will save the visualization from a multitude of different camera orientations. This allows the user later in the Cinema viewer to interactively rotate around the object much as though the object was live instead of just a collection of image frames. The camera type can be selected using the Camera type control in the Save Cinema wizard and either static or phi-theta cameras can be selected. When a phi-theta camera is selected, the number of camera angles in the phi and theta dimensions can be set using the Phi and Theta controls.

Frame settings

The second tab in the Save Cinema wizard (see Figure 1.157) contains controls that select the range and stride of time states that will be included in the Cinema database. Use the Frame start controls to select the beginning time state for the Cinema database. A value of zero corresponds to the first time state. Use the Frame end controls to set the last time state that will be included in the Cinema database. Finally, use the Frame stride controls to set the stride that will be used between the start and end time states, which is useful when making shorter preview databases that vary over time but do not include all time states.

Fig. 1.157: Save Cinema wizard (screen 2)
Saving Cinema from Libsim

It is possible to use VisIt’s Libsim to directly save Cinema databases in situ from an instrumented simulation. This means that the Cinema database can be generated incrementally as the simulation runs, making it possible to periodically check in on the simulation by viewing the Cinema database. To add Cinema support to a simulation instrumented with Libsim, there are 3 calls that need to be made. First, the simulation must call `VisItBeginCinema`, which passes the parameters that describe the Cinema database format and returns a handle to a Cinema object. Next, the simulation must call `VisItSaveCinema` to make Libsim generate and add the appropriate images to the Cinema database, taking into account the type of camera being used. The `VisItSaveCinema` function can be called repeatedly to add new time states to the Cinema database. It is the simulation’s responsibility to make Libsim calls that set up VisIt plots or restore a session so there are plots when `VisItSaveCinema` is called. Finally, the simulation must call `VisItEndCinema` to close out the Cinema database context and free associated memory. A working example can be found in the batch simulation example in VisIt’s simulation directory. The overall call structure for creating a Cinema database looks something like this:

```c
visit_handle h = VISIT_INVALID_HANDLE;
visit_handle hvar = VISIT_INVALID_HANDLE;
double time_value = 0.;
VisItBeginCinema(&h, "visit.cdb", VISIT_CINEMA_SPEC_A, 0,
                  VISIT_IMAGEFORMAT_PNG, 800, 800,
                  VISIT_CINEMA_CAMERA_PHI_THETA, 12, 7,
                  hvar);

while (1) /* Simulation main loop */
{
    /* Compute... */

    VisItSaveCinema(h, time_value);
}

VisItEndCinema(h);
```

The above code example will generate a Cinema database using the plots that have been set up elsewhere using Libsim. Since Cinema output may sometimes serve as the only simulation data product, it can be useful to save out additional variables. The last argument to `VisItBeginCinema` is a handle to a name list object. When the handle is set to `VISIT_INVALID_HANDLE`, there is no name list and the argument does nothing. If instead, the name list is created and filled with a list of variable names from the simulation, the VisIt plots will have their variables changed to the variables in the name list and Libsim will generate a Cinema database with images for each variable. The variable becomes a parameter in the Cinema viewer. A name list object is created and populated like this:

```c
visit_handle hvar;
VisIt_NameList_alloc(&hvar);
VisIt_NameList_addName(hvar, "pressure");
VisIt_NameList_addName(hvar, "rho");
VisIt_NameList_addName(hvar, "energy");
```

1.5.5 Exporting databases

Plot geometry can be saved to a handful of geometric formats by saving the plots in the window to a format such as VTK. Often saving the plot geometry, which only consists of the visible faces required to draw the plot, is not enough. When interfacing VisIt to other tools you may want to save out the database in a different file format. For instance, you might plot a 3D database and want to export actual 3D cells for the entire database instead of just the externally visible geometry. You might also want to save out additional variables that you did not plot. VisIt allows this kind of data export via the Export Database Window, shown in Figure 1.158.
Fig. 1.158: Export Database Window
You can find the Export Database Window in the Main Window’s File menu. To save a database, you must first have opened a database and created a plot. Note that the data transformations applied by plots or operators will affect the data that you export. This allows you to alter the data using sophisticated chains of operators before you export it for use in another tool.

**Exporting variables**

The Export Database Window allows you to export a subset of the variables for your active plot’s database by letting you specify which variables are to be exported. To choose which variables should be exported, you can type the names of the variables to export into the Variables text field or you can select from the available variables in the Variables menu depicted in Figure 1.159. You can select as many variables as you want from the menu. Each time you select a variable from the Variables menu, VisIt will append it to the list of variables to be exported.

**Choosing an export file format**

The Export Database Window lists the names of the database reader plugins that can also write data back into their native file formats. A small handful of the total number of database plugins currently support this feature but in the future most formats will support this capability more fully, making VisIt not only a powerful visualization tool but a powerful database conversion tool.

You can try to use any of the supported export formats to export your data but some of the file formats may not be able to accept certain types of data. The Silo file format can safely export any type of data that you may want to export. If you want to export data to other applications and the data must be stored in an ASCII file that contains columns of data, you might want to choose the Xmdv file format. If you want to choose a specific database plugin to export your data files, make a selection from the Export to menu shown in Figure 1.160.
Export Options

Some export formats support various options. Those options will be presented in a dialog box when the Export button is pressed in the Export Database Window. For example, shown below are some options for exporting to the Silo database.

If VisIt has been compiled with HDF5 support, Silo’s export options will include the ability to select either the PDB or HDF5 driver. The Checksums check-box indicates where the Silo library should compute checksums on the exported data. In addition, the DBSetCompression() option text box is for specifying a compression string to be used in Silo’s DBSetCompression() method before exporting data.

When the meaning of an export option is not clear, try also pressing the Help button in Export options for XXX writer window to get more information.

1.5.6 Printing

VisIt allows you to print the contents of any visualization window to a network printer or to a PostScript file.

The Printer Window

Open the Printer Window by selecting Print window from the Main Window’s File menu. The Printer Window’s appearance is influenced by the platform on which you are running VisIt so you may find that it looks somewhat different when you use the Windows, Unix, or MacOS X versions of VisIt. The MacOS X version of the Printer Window is shown in Figure 1.162.
1.5. Saving and Printing

Fig. 1.161: Export options example (for Silo)

Fig. 1.162: Printer window
1.6 Visualization Windows

A visualization window, also known as a vis window, is a window that displays plots and allows you to interact with them using the mouse. The vis window not only allows for direct manipulation of plots but it also provides a popup menu and toolbar that allow you to switch window modes, activate interactive tools, and perform commonly used operations. This chapter explains how to manage and use vis windows.

1.6.1 Managing vis windows

VisIt allows you to create up to 16 vis windows and to manage those vis windows, VisIt provides controls to add vis windows, remove vis windows or alter their layout. The controls for managing vis windows are located in the Main Window’s Windows menu (see Figure 1.163), as well as in the vis window’s Toolbars and Popup menu.

Fig. 1.163: Window menu

Adding a new vis window

You can add a new vis window in a few different ways, the first of which is by selecting the New option from the Main Window’s Windows menu. You can also click on the New window icon in the vis window’s Toolbar or you can select the New window option from the Windows submenu in the vis window’s Popup menu to add a new vis window. When you add a new window, it will be sized according to the window layout so if you have only a single, large vis window, the new vis window will also be large. You can change the window layout to shrink the vis windows so that they both fit on the screen. Vis windows are numbered 1 to 16 so the new window will have the first available number for which there is not already a window. If you have windows 1, 2, and 4, vis window 3 would be created by adding a new window. Adding a new window also makes the new window the active window.

A new vis window can also be added by cloning the active window. You can clone the active window by selecting the Clone option from the Main Window’s Windows menu or you can click the Clone window icon in the vis window’s Toolbar. When you clone the active window, VisIt creates a new window as if you had clicked the Add option but it also copies the plots, annotations, and lighting from the active window so that the new window is identical in appearance to the active window. When plots are copied to the new cloned window, they have not yet been generated.
so their plot list entries in the Plot list are green. You can force the plots to be generated by clicking the Draw button in the Main Window.

Deleting a vis window

There are four ways to delete a vis window. The first way is to select the Delete option from the Main Window’s Windows menu. When you delete a window in this manner, the active window gets deleted and VisIt makes the window with the smallest number the new active window. The second way to delete a window is to click on the close window button in the window decorations provided by the windowing system. The window decorations’ appearance varies based on the platform and windowing system used to run VisIt, but the button used to close windows is commonly a button with an X in it. An example of a close window button is shown in Figure 1.164.

![Fig. 1.164: Window decorations with close button](image)

The third way to delete a vis window is to click on the Delete window icon in the vis window’s Toolbar. The fourth way to delete a vis window is to use the Delete option in the vis window’s Popup menu. When you use the Toolbar or the Popup menu to delete a window, the window does not need to be the active window as when other controls are used.

Clearing plots from vis windows

The Main Window’s Windows menu provides a Clear all option that you can use to clear the plots from all vis windows. Selecting this option does not delete the plots from a vis window’s plot list but it does clear the plots so they have to be regenerated by VisIt’s compute engine. You can also clear the plots for just the active window by selecting the Plots option from the Clear submenu in the Main Window’s Windows menu (see Figure 1.165). You might find clearing plots useful when you want to make several changes to plot attributes because, unlike plots that are already generated, setting attributes of cleared plots does not force them to regenerate when you change their attributes.

In addition to clearing plots, you can also clear pick points and reference lines from a vis window. A pick point is a marker that VisIt adds to a vis window when you click on a plot in pick mode. The marker indicates the location of the pick point. A reference line is a line that you draw in a vis window when it is in lineout mode. You can clear a vis window’s pick points or reference lines, by selecting the Pick points or Reference lines options from the Clear submenu in the Main Window’s Windows menu.

Changing window layouts

VisIt uses different window layouts to organize vis windows so they all fit on the screen. Changing the window layout typically resizes all of the vis windows and moves them into a tiled formation. If there are not enough vis windows to complete the desired layout, VisIt creates new vis windows until the layout is complete. You can change the layout selecting a new layout from the Layouts menu located in the Main Window’s Windows menu or you can click on a layout icon in the vis window’s Toolbar.

Setting the active window

VisIt has the concept of an active window that is the window to which new plots are added. You can change the active window by selecting a window number from the Active window menu located near the top of the Main Window. Setting the active window updates the GUI so that it displays the state for the new active window. The Active window menu is shown in Figure 1.166. You can also set the active window using the Active window submenu in the Main Window’s Windows menu or you can click on the Active window icon in the vis window’s Toolbar.
Fig. 1.165: Clear menu

Fig. 1.166: Active window menu
Copying window attributes

VisIt allows you to copy window attributes and plots from one window to another when you have more than one window. This can be useful when you are comparing plots generated from similar databases. The Copy menu, shown in Figure 1.167, contains options to copy the view, lighting, annotations, plots, or everything from other from other vis windows. Under each option, the Copy menu provides a list of available vis windows from which attributes can be copied so, for example, if you have two windows and you want to copy the view from vis window 1 into vis window 2, you can select the Window 2 option from the View from submenu. The list of available windows depends on the vis windows that you have created. You can copy the lighting from one window to another window by using the Lighting from submenu or you can use the Annotations from or Plots from to copy the annotations or plots, respectively. If you make a selection from the Everything from submenu, all attributes and plots are copied into the active vis window.

![Fig. 1.167: Copy menu](image)

Locking vis windows together

When you use VisIt to do side by side comparisons of databases, you may find is useful to lock vis windows together. Vis windows can be locked together in time so that when you change the active database timestep in one database, as when viewing an animation, all vis windows that are locked in time switch to the same database timestep. You can lock vis windows together in time by selecting the Time option from the Lock menu (see Figure 1.168) in the Main Window’s Windows menu. Any number of windows can be locked together in time and you can turn off time locking at any time.

You can also lock interactive tools together so that updating a tool in one window updates the tool in other windows that have enabled tool locking. This can be useful when you have sliced a database using the plane tool in more than one window and you want to be able to change the slice using plane tool in either window and have it affect the other vis windows. You can enable tool locking by selecting the Tools option from the Lock menu.

In addition to locking vis windows in time, or locking their tools together, you can also lock vis windows’ views together so that when you change the view in one vis window, other vis windows get the same view. When you change the view in a vis window that has view locking enabled, the view only effects other vis windows that also have view locking enabled and have plots of the same dimension. That is, when you change the view of a vis window that
contains 3D plots, it will only have an effect on other locked vis windows if they have 3D plots. Vis windows that contain 2D plots are not affected by changing the view of a vis window containing 3D plots and vice-versa. When you enable view locking, the vis window snaps to the view used by other vis windows with locked views or it stays the same if no other vis windows have locked views. To enable view locking, select the **View** option from the **Lock** menu or click on the **Lock view icon** in the vis window’s **Toolbar**.

### 1.6.2 Using vis windows

The first thing to know about using a vis window is how to change window modes. A window mode is a state in which the vis window behaves in a specialized manner. There are four window modes: Navigate, Zoom, Lineout, and Pick. Vis windows are in navigate mode by default. This means that most mouse actions are used to move, rotate, or zoom-in on the plots that the vis window displays. Each vis window has a **Popup menu** that can be activated by clicking the right mouse button while the mouse is inside of the vis window. The **Popup menu** contains options that can put the vis window into other modes and perform other common operations. To put the vis window into another window mode, open the **Popup menu**, select **Mode** and then select one of the four window modes. You can also change the window mode using the vis window’s **Toolbar**, which has buttons to set the window mode. You can find out more about the **Popup menu** and **Toolbar** later in this chapter.

**Navigate mode**

Navigate mode is VisIt lingo for moving and zooming-in on plots. When the vis window is in navigate mode, clicking the left mouse button and dragging with the mouse will perform an action that moves, rotates, or zooms the plot. The mouse motions used to rotate plots are shown in Figure 1.169. You can translate plots by holding down the **Shift** key before left-clicking and dragging the plot. You zoom in on plots by clicking the middle button and moving the mouse up or down. Sometimes the controls are modified based on the interactor settings. For more information, look at the section on **Interactor settings**.
Zoom mode

When the window is in zoom mode, you can draw a box around the area of the vis window that you want drawn larger. Press the left mouse button and move the mouse to sweep out a box that will define the area to be zoomed. Release the mouse button when the zoom box covers the desired area. If you start zooming and decide against it before releasing the left mouse button, clicking one of the other mouse buttons cancels the zoom operation. Changes to the view can be undone by selecting the Undo view option from the popup menu’s View menu. Sometimes the zoom controls can change based on the interactor settings, which are described further on in Interactor settings.

Lineout mode

Lineout mode is only available when the vis window contains 2D plots. A lineout is essentially a slice of a two dimensional dataset that produces a one dimensional curve in another vis window. When a vis window is in lineout mode, pressing the left mouse button in the vis window creates the first endpoint of a line that will be used to create a curve. As you move the mouse around, the line to be created is drawn to indicate where the lineout will be applied. When you release the mouse button, VisIt adds a lineout to the vis window and a curve plot is created in another vis window.

Pick mode

When a vis window is in pick mode, any click with the left mouse button causes VisIt to calculate the value of the plot at the clicked point and place a pick point marker in the vis window to indicate where you clicked. The calculated value is printed to the Output Window and the Pick Window.

1.6.3 Interactor settings

Some window modes such as Zoom mode and Navigate mode have certain interactor properties that you can set. Interactor properties influence how user interaction is fed to the controls in the different window modes. For example, you can set zoom interactor settings that clamp a zoom rectangle to a square or fill the viewport when zooming. VisIt provides the Interactors window so you can set properties for window modes that have interactor properties. The Interactors window is shown in Figure 1.170.
Zoom interactor settings

The zoom interactor settings are mostly used when the vis window is in zoom mode. When the vis window is in zoom mode, clicking in the vis window will anchor a point that becomes one of the corners of a zoom rectangle. When you release the mouse, the point over which the mouse was released becomes the opposite corner of the zoom rectangle. VisIt’s default behavior is to show guidelines that extend from the edges of the zoom rectangle to the edges of the plots’ bounding box when the vis window is in 2D mode. If you want to turn off the guidelines, click off the **Show Guidelines** check box in the **Interactors window**.

When sweeping out a zoom rectangle in zoom mode, VisIt allows you to draw a rectangle of any proportion. The relative shape of the zoom rectangle, in turn, influences the shape of the viewport drawn in the vis window. If you hold down the *Shift* key while sweeping out the zoom rectangle, VisIt will constrain the shape of the zoom rectangle to a square. If you want VisIt to always force a square zoom rectangle so that you don’t have to use the *Shift* key, you can click on the **Clamp to Square** check box, click **Apply** in the **Interactors window** and save your settings.

Using the **Clamp to Square** zoom mode is a good way to maximize the amount of the vis window that is used when you zoom in on plots and when the vis window is in zoom mode. When the vis window is in navigate mode, the middle mouse button also effects a zoom. By default, zooming with the middle mouse button zooms into the plots but keeps the same vis window viewport which may, depending on the aspect ratio of the plots, not make the best use of the vis window’s pixels. Fortunately, you can turn on the **Fill viewport on zoom** check box to force middle mouse zooming to also enlarge the viewport to its largest possible size in order to make better use of the vis window’s pixels.

Navigation styles

When VisIt displays 3D plots, there are a few navigation styles from which you can choose by clicking on the following radio buttons in the **Interactors window**: Trackball, Dolly, and Flythrough. The default navigation style for 3D plots
is: Trackball and it allows you to interactively rotate plots and move around them but it keeps the camera at a fixed distance from the plots and while it can get infinitely close to plots when you zoom in, it can never touch them or go inside of them. The Dolly navigation style behaves like the trackball style except that when the camera zooms, it is actually moved. The Flythrough navigation style moves the camera and allows you to fly into plots and out the other side.

1.6.4 The Popup menu and the Toolbar

Each vis window contains a **Popup menu** and a **Toolbar**, which can be used to perform several categories of operations such as window management, setting the window mode, activating tools, manipulating the view, or playing animations. Options in the **Popup menu** exist in the **Toolbar** and vice-versa. A group of actions that is represented in the **Popup menu** as a menu usually maps to a toolbar in the vis window’s **Toolbar**. To perform an action using the **Toolbar**, you can just click on its buttons. Access the **Popup menu** by pressing the right mouse button in the vis window. Select the desired item, then release the mouse button.

**Hiding toolbars**

The **Popup menu** has a **Customize** menu that lets you customize the vis window’s **Toolbar**. For instance, you can choose to hide all of the toolbars so that they do not take up any of your screen space if you use a small monitor. If you want to hide all toolbars, you can select the **Hide toolbars** option from the **Customize** menu. If you want to show the toolbars again, you can click the **Show toolbars** option in the **Customize** menu. Note that when you select the **Show toolbars** option, VisIt only shows the toolbars that were enabled before they were hidden. If you want to enable or disable individual toolbars, you can select from the **Toolbars** menu under the **Customize** menu so VisIt only shows the toolbars that you routinely need. Once you tell VisIt which toolbars you want to use, you can save your preferences using the **Save settings** option in the **Main Window’s Options** menu so that the next time you run VisIt, it only shows the toolbars that you enabled.

**Moving toolbars**

Each of the vis window **Toolbar**’s smaller toolbars can be moved to other edges of the vis window by clicking the small tab on the left or top side of the toolbar and dragging it to other edges of the vis window.

**Switching window modes**

The **Popup menu** contains a **Mode** menu (see Figure 1.172) that contains the 5 window modes. You can select a window mode from the **Mode** menu to change the vis window’s mode. If you want to move or zoom the plot, choose **navigate** or **zoom** modes. If you want to extract data from the plots in the vis window, choose **lineout** mode or one of the **pick** modes. You can also use the **Mode toolbar** to change the vis window’s window mode.

**Activating tools**

The **Popup menu** contains a **Tools** menu (see Figure 1.173) that lists of all of VisIt’s interactive tools. Each tool shown in the menu has an associated icon that is used to indicate if the tool is enabled and if it is available in the vis window. Some tools are not available if the vis window does not contain plots or if the plots in the vis window are the wrong dimension to be used with the tool. In that event, the tool cannot be activated and the menu and toolbar entries for that tool are disabled. If a tool is available, its icon is bright blue; otherwise the icon is grayed out. If a tool is enabled, its icon has a selection rectangle around it. To activate a tool, choose an inactive tool from the **Tools** menu or click on its...
Fig. 1.171: Customize menu
button in the **Toolbar**. To deactivate a tool, choose the tool that you want to deactivate from the **Tools** menu or click on its button in the **Toolbar**.

**View options**

VisIt’s **Popup menu** and **Toolbar** (see Figure 1.174) have several options that are available for manipulating the view. You can reset the view, recenter the view, undo a view change, toggle perspective viewing, save and reuse useful views, or choose a new center of rotation.

**Resetting the view**

The **Popup menu** has a **Reset view** option (see Figure 1.174) that resets the view used to view the plots in the vis window. The view is typically reset to look down the -Z axis in a right-handed coordinate system. You can reset the view by selecting the Reset view option from the **Popup menu** or by clicking on the **Reset view icon** in the **Toolbar**.

**Recentering the view**

Sometimes adding a plot to a vis window that already contains plots can result in a lop-sided visualization. This happens when the spatial extents of the plots do not match. The **Popup menu** has a **Recenter view** option (see Figure 1.174) to calculate a new center of rotation for the plots so they are drawn in the center of the window. You can
Fig. 1.173: Tool toolbar and menu

Fig. 1.174: View toolbar and menu
also recenter the view by clicking on the **Recenter view icon** in the **Toolbar**. To make sure that the view updates appropriately when new plots are added to the vis window, you may also want to check the **Auto center view** check box that is available in the **View Window**.

### Undo view

The vis window saves the last ten views in a buffer so that you can restore them if you make an unintended change to the view. You can undo a view change, by selecting the **Undo view** option in the **Popup menu’s View** menu or by clicking the **Undo view icon** in the **Toolbar** (see Figure 1.174).

### Changing view perspective

![Perspective examples](image)

*Fig. 1.175: Perspective examples*

When the vis window contains 3D plots, the perspective setting can be used to enhance how 3D the plot looks. In a perspective projection, graphics grow smaller as they recede into the distance which makes them look more realistic. To change the perspective setting, click on the **Perspective** option in the **Popup menu’s View** menu (see Figure 1.174). When the vis window uses a perspective projection, the Popup menu’s Perspective option will have a selection rectangle around its icon. You can also turn perspective on or off by clicking on the **Perspective icon** in the **Toolbar**. The difference in appearance having perspective and not having it is shown in Figure 1.175.

### Locking views

The vis window can lock its view to other vis windows. When this toggle is set, making a change that affects the view in the active vis window will cause other vis windows that have the lock views toggle set to receive the same view as the active window. To lock the view, select the **Lock view** option from the **Popup menu’s View** menu (see Figure 1.174) or click on the **Lock view icon** in the **Toolbar**. Note that you can lock 2D and 3D windows separately.

### Saving and reusing views

Sometimes when analyzing a database, it is useful to be able to toggle between several different views. VisIt allows you to save up to 15 views that you can then use to look at different parts of your visualization. When you navigate to
a view that you like, click the **Save view** icon in the **View** toolbar or click the **Save view** option in the **Popup menu’s View** menu to save the view. When you save a view, VisIt adds a new numbered camera icon to the **View** toolbar and the **Popup menu**. Clicking on a view icon makes VisIt use the view that is associated with the clicked icon so you have one-click access to all of your saved views. You can preserve the saved views across VisIt sessions if you save your settings. If you want to delete the saved views so you can create different saved views, click the **Clear saved views** icon next to the **Save views** icon in the **View** toolbar.

**Fullframe mode**

Some databases yield plots that are so long and skinny that they leave most of the vis window blank when VisIt displays them. VisIt provides Fullframe mode to stretch the plots so they fill more of the vis window so it is easier to see them. It is worth noting that Fullframe mode does not preserve a 1:1 aspect ratio for the displayed plots because they are stretched in each dimension so they fit better in the vis window. To activate Fullframe mode, click on the **Fullframe** option in the **Popup menu’s View** menu.

**Choosing a new center of rotation**

When you are working with a 3D database and you have created plots and zoomed in on them, you should set the center of rotation. The center of rotation is the point about which the plots are rotated when you rotate the plots in navigate mode. Normally, the center of rotation is set to the center of the plots being visualized. When you zoom way in on plots and attempt to rotate them, the default center of rotation often causes plots to whiz off of the screen when you rotate because the center of rotation is not close enough to the geometry that you are actually viewing. To set the center of rotation to something more suitable, VisIt provides the **Choose center** button, which can be accessed in the **Popup menu** or in the **View** toolbar. Once you click the **Choose center** button, VisIt temporarily switches to pick mode so you can click on the part of your visualization that you want to become the new center of rotation. Once you click on a plot, VisIt exits pick mode and uses the picked point as the new center of rotation. After setting the center of rotation, VisIt will make sure that the picked point is visible at all times.

**Animation options**

The animation controls in VisIt’s **Main Window** are not the only controls that are provided for playing animations. Each vis window’s **Popup menu** and **Toolbar** has options for playing and stepping through animations. To play an animation, select the Play option from the **Popup menu’s Animation** menu or click on the **Play icon** in the **Toolbar**, shown in Figure 1.176. To play the animation in reverse, select the **Reverse play option** or click on the **Reverse play icon** in the **Toolbar**. To stop the animation from playing, select the **Stop option** in the **Animation menu** or click on the **Stop icon** in the **Toolbar**. If you want to advance or reverse one frame at a time, use forward or reverse step.

**Window options**

Many window options have previously been explained in this chapter so this section describes some addition options that were not covered. Many of the options in the **Main Window’s Windows** menu are also present in the **Popup menu’s Window** menu and toolbar (see Figure 1.177).
1.6. Visualization Windows

Fig. 1.176: Animation toolbar and menu

Fig. 1.177: Window toolbar and menu
Changing bounding-box mode

The vis window allows a simple wireframe box to be substituted for complex plots when you want to rotate or move them. This is called bounding-box navigation and you can use it during navigate mode for complex plots so you can navigate faster when a vis window contains plots that take a long time to redraw. You can change the bounding-box mode by selecting the Navigate bbox option from the Popup menu’s Window menu shown in Figure 1.177. You can also change the bounding-box mode by clicking on the Bounding-box icon in the Toolbar.

Engaging spin

Spin is a setting that makes plots spin after the user stops rotating them and it provides a nice, easy way to see the entire plot without having to actively rotate it. To spin a 3D plot, turn on the Spin option in the Popup menu’s Windows menu and then rotate the plot as you would in navigate mode. The plot will continue to spin after you release the mouse buttons. You can also engage spin using the Spin option in the Main Window’s Windows menu or by clicking the Spin icon in the vis window’s Toolbar. You can stop plots from spinning by turning off spin.

Inverting the foreground and background colors

Sometimes it is useful to swap the vis window’s foreground and background colors. You can invert the background and foreground colors by clicking on the Windows menu’s Invert background option. Note that this option is disabled when the vis window has a gradient background.

Clear options

The Clear menu (see Figure 1.178) in the Popup menu contains options that cause certain items such as: plots, pick points, and reference lines to be removed from a vis window. The Clear menu also appears in the Main Window’s Windows menu.

Fig. 1.178: Clear menu
Clearing plots from all windows

Sometimes it is useful to clear all plots from the vis window. Clearing plots from the vis window does not delete the plots but instead deletes their computed geometry and returns them to the new state so they appear green in the Plot list. An example of when you might want to clear plots is when you change material interface reconstruction options since changing them requires a plot to be regenerated. Rather than deleting plots that existed before changing the material interface reconstruction parameters, you can clear the plots and force them to be completely regenerated by clearing the plots.

Clearing pick points

Click on the Clear menu’s Clear pick points option if you want to remove all of the pick labels that were added when you picked on the plots in the vis window. Clearing the pick points also removes any pick information related to those pick points in the Pick window.

Clearing reference lines

Click on the Clear menu’s Clear reference lines option if you want to remove all of the reference lines that were added to the vis window when you performed lineouts on the plots in the vis window.

Plot options

The Plot toolbar and Plot menu let you create new plots using variables from the open databases and also let you hide, delete, and draw the plots that correspond to the selected plot entries in VisIt’s Plot list. The Plot menu is always available in the Popup menu but the Plot toolbar is not visible by default. If you want to make the Plot toolbar visible, you can turn it on in the Popup menu’s Customize menu. The Plot menu and toolbar are shown in Figure 1.179.

Adding a plot

The Plot menu and toolbar both provide options for you to add new plots. Each plot has its own menu option or icon that contains the variables that can be plotted from the open database. To add a new plot using the Plot menu, click the Add plot option to activate the list of available plots and then select a variable for the desired plot type. To add a new plot using the Plot toolbar, click on the icon for the desired plot type and select a variable from its variable menu. A new plot will appear in the Main Window’s Plot list and it will be in the new state. To draw the plot, click the Draw button.

Drawing a plot

All plots added using the Plot menu or toolbar are in the new state, indicating that they have not been generated yet. To generate a plot once it has been created, click the Draw plots option in the Plot menu.

Hiding active plots

To hide the active plots, which are the plots that are highlighted in the Main Window’s Plot list, click the Plot menu’s Hide active plots option. Once clicked, the selected plots are made invisible until you hide them again to show them.
Fig. 1.179: Plot toolbar and menu
Deleting active plots

To delete the active plots, which are the plots that are highlighted in the Main Window’s Plot list, click the Plot menu’s Hide active plots option. Once a plot has been deleted, you can’t get it back.

Operator options

The Operator menu and toolbar allow you to add new operators and remove operators from plots. The Operator menu is always available in the Popup menu but the Operator toolbar is not visible by default. If you want to make the Operator toolbar visible, you can turn it on in the Popup menu’s Customize menu. The Operator menu and Operator toolbar are shown in Figure 1.180.

Adding an operator

The Operator menu and toolbar both provide options for you to add new operators. Each operator has its own menu option or icon that adds an operator of that type to the selected plots when you click its menu option or icon.

Removing the last operator

The Operator menu and toolbar both have options for you to remove the last operator from a plot. Each plot has a list of applied operators and clicking the Remove last operator menu option or icon will remove the last operator from each plot that is selected in the Plot list. Plots that have been drawn are regenerated.

Removing all operators

The Operator menu and toolbar both have options for you to remove all operators from a plot. Each plot has a list of applied operators and clicking the Remove all operators menu option or icon will remove all operators from each plot that is selected in the Plot list. Plots that have been drawn are regenerated.

Lock options

The Lock menu and toolbar, both shown in Figure 1.181, allow you to lock certain visualization window attributes so that when you change them, other locked visualization windows also update. Currently, you can lock the view, time and tools. See Locking Windows for more information on how to use the lock options.

1.7 Subsetting

Meshes are frequently composed of a variety of subsets that represent different portions of the mesh. Common examples are domains, groups (of domains), AMR patches and levels, part assemblies, boundary conditions, node sets and zone sets, materials and even material species.
Fig. 1.180: Operator toolbar and menu
Users often find it useful to restrict which subsets are used in any given operation to focus their analyses on only certain regions of interest. This is handled through VisIt’s Subset Window. Here, we describe VisIt’s subsetting functionality and Subset Window in detail.

What is described here is primarily about pre-defined, first-class, named subsets as created by the data producer and supported within VisIt. Nonetheless, it is important to keep in mind that there are other ways that the data producer can organize data within VisIt’s GUI or that users can employ VisIt’s Expressions and Operators to create and manage subsets. However, using these other approaches for the sole purpose of subsetting is often cumbersome through VisIt’s GUI. To understand why as well as read about other issues related to subsetting, please see these developer notes.

1.7.1 What is a subset?

VisIt has first-class support for four different kinds of subsets; Domains, Groups (also called Blocks), Materials and material Species. In particular, as currently designed, any given mesh in VisIt can have only one decomposition into each of these kinds of subsets. That is, a mesh can have only one Domain decomposition, one Group decomposition, one Material decomposition and one material Species decomposition. A fifth kind of subset, Enumerated, is also supported and provides some additional generality but cannot be used in combination with the other four or even with other Enumerated subsets.

Data producers as well as the database plugins that read data into VisIt often have flexibility in deciding how to utilize these various kinds of subsets in representing their data. We describe each of these kinds of subsets and constraints in their use below.

**Domain Subsets**

VisIt’s concept of a Domain subset is fundamental to its parallel programming and execution model. A domain in VisIt represents a chunk of mesh plus its variables that is both stored (in files and in memory) and processed coherently as a single, self-contained unit. Large meshes in VisIt are typically decomposed into Domain subsets for parallel processing. In fact, except in rare cases, the maximum number of MPI tasks VisIt may use is determined by the

Fig. 1.181: Lock toolbar and menu
number of Domain subsets created by the data producer. VisIt’s approach to processing a mesh in parallel is often described as piggy-backing off of the parallel decomposition created by the data producer.

Domain subsets also represent the unit of work VisIt allocates in its load balancing algorithms. If VisIt is running on $M$ processors and reading a mesh of $N$ domains, then if $N<M$, $N-M$ processors will idle for operations involving that mesh. On the other hand, if $N>kM$ (where $k$ is an integer), some processors will be assigned $k$ domains and some $k+1$ domains.

In almost all cases, if a mesh is to be processed in parallel by VisIt, it must have been decomposed into Domain subsets by the data producer prior to reading the data into VisIt. In general, VisIt does not perform any on-the-fly domain decomposition of data it is reading. However, there is one, special case where VisIt can perform on-the-fly domain decomposition of a large, monolithic mesh; a structured mesh stored in a file format that supports hyper-slabbed I/O. In this simple case, VisIt will try to evenly decompose the 2 or 3D mesh into roughly equal sized hyper-slabs whose number is determined by the number of parallel tasks. VisIt will also then utilize the file format’s hyper-slab I/O routines to read into each parallel task only the part(s) of the mesh assigned to that task.

A mesh is required to have domains if it is ever to be processed in parallel by VisIt.

**Group or Block Subsets**

Groups (or Blocks) are just unions of Domains. They are optional. A mesh is not required to have groups. On the other hand, if a mesh has Groups, then every domain in the mesh must be assigned to one and only one Group subset. Groups may be used to represent, for example, the files in which multiple domains are stored or sets of neighboring domains that share a common logical/structured indexing arrangement in an otherwise globally unstructured mesh.

The key constraint about group subsets is that they can represent only unions of the domain subsets. Internally in VisIt, a group subset is implemented as a list of domain subset ids.

**Material Subsets**

Material subsets are used to represent the decomposition of a mesh into various materials. For example, a mesh may be composed of steel, brass, and aluminum materials. If these materials are given integer ids 83 (\texttt{int('S')}), 66 (\texttt{int('B')}) and 65 (\texttt{int('A')}), then each zone (or cell) in the mesh can be assigned a value of 83, 66 or 65 to indicate the zone is composed of steel, brass or aluminum. This would be equivalent to an integer valued (with 3 unique values), zone-centered variable on the mesh.

For material subsets, however, VisIt also supports a notion of mixing where a single zone (or cell) can be composed of multiple materials each occupying some fractional volume of a whole zone (or cell). From a sub-setting perspective, a more formal way of thinking about mixing is that it is way of supporting partial inclusion of a mesh zone (or cell) within a given material subset.

Material subsets are optional. Furthermore, if material subsets are defined additionally supporting mixing is also optional. Only some data producers that involve Material subsets also involve mixing.

When mixing materials are involved, VisIt can employ a variety of sophisticated Material Interface Reconstruction (MIR) algorithms to draw the interfaces between materials based on the volume fractions of the mixing. The main point about MIR is that it represents an additional computational burden when manipulating Material subsets. Manipulating Group or Domain subsets has no such equivalent computational cost.

**Mesh Variables with Material Specific Properties**

For some mesh variables, data producers may have different values of the variable for each of the materials within various zones (or cells) of the mesh where mixing is occurring. When such a variable is being plotted, for example with the Pseudocolor Plot, what value/color should VisIt show for such zones? The fact is, depending on the user’s needs, VisIt is capable of showing either an overall value for the zone or showing the material-specific values in the zone. This can be handled through appropriate use of VisIt’s (MIR) algorithms and Subset Window controls.
Species Subsets

In addition to mixing, another feature Materials subsets support is a notion of Species. For example, there are many different varieties of brass and steel depending on the alloys used. Neither brass nor steel are themselves pure elements on the periodic table. They are instead alloys of other pure metals. Common Yellow Brass is, nominally, a mixture of Copper (Cu) and Zinc (Zn) while Tool Steel is composed primarily of Iron (Fe) but mixed with some Carbon (C) and a variety of other elements.

Let's suppose we are dealing with the following alloys and species compositions...

<table>
<thead>
<tr>
<th>Material</th>
<th>Species composition</th>
</tr>
</thead>
<tbody>
<tr>
<td>Brass</td>
<td>Cu:65%, Zn:35%</td>
</tr>
<tr>
<td>T-1 Steel</td>
<td>Fe:76.3%, W:18%, Cr:4.0%, C:0.7%, V:1%</td>
</tr>
<tr>
<td>O-1 Steel</td>
<td>Fe:96.2%, W:0.5%, Cr:0.5%, C:0.9%, Mn:1.4%, Ni:0.5%</td>
</tr>
</tbody>
</table>

The Materials decomposition would consist of 3 subsets for Brass, T-1 Steel and O-1 Steel. For the Species decomposition, Brass would be further decomposed into 2 Species subsets, T-1 Steel into 5 Species subsets and O-1 Steel, 6 Species subsets.

Alternatively, one could opt to characterize both T-1 Steel and O-1 Steel as a single, non-specific Steel having 7 Species subsets, Fe, W, Cr, C, V, Mn, Ni where for T-1 Steel, the Mn and Ni Species subsets are always empty and for O-1 Steel the V Species subset is always empty. In that case, there would only be 2 Materials subsets for Brass and non-specific Steel.

Species subsets are optional. A mesh does not need to have them defined. However, as currently designed, a data producer cannot define Species subsets without also defining Materials subsets (even if there is only one material subset for the whole mesh).

A final thing to note about Species subsets is that they do not represent spatially distinct parts of the mesh like Domains, Groups, or Materials. Instead, Species, if they are defined are ever present, everywhere in the mesh. Only their relative concentrations vary at any given point in the mesh. But, Species do permit subsetting a particular physical quantity’s value in that, for example the total pressure in a zone can be decomposed into partial pressures on each of the species comprising the materials in the zone. Furthermore, using the Subset Window, VisIt can then control which partial value(s) are used in a particular plot.

Domains, Groups, Materials and Species In Combination

A given mesh may involve any combination of Domain, Group and Material subsets. Furthermore, VisIt’s Subset Window makes it possible to manipulate these four kinds of subset in combination. That is, a user can simultaneously control which domains, which materials and which groups VisIt should process in any given operation. However, manipulating subsets in combination works only for these kinds of subsets. Other kinds of sub-setting, such as Enumerated subsets which are discussed next, are not as well integrated.

Enumerated Subsets

A key constraint of the other kinds of subsets is that any given mesh can have only one decomposition into domains and one decomposition into groups and one decomposition into materials. However, a mesh can be composed of any number of Enumerated subsets. Enumerated subsets are defined by first defining the enumeration class and then creating a bitmap like variable over the mesh to indicate which mesh entities (nodes, edges, faces or volumes) belong to which subsets of the enumeration class.

Within an enumeration class, the sets can be arranged hierarchically so that some sets contain other sets as in a part assembly.
Enumerated subsets do not work in combination with domains, groups or materials or in combination with other classes of Enumerated subsets. On the other hand, for any given mesh, there can be any number of enumeration classes, each defining a collection of related subsets. For example, if a mesh has defined two enumeration classes, one for node sets and one for face sets, then different subsets of nodes can be manipulated simultaneously or different subsets of faces can be manipulated simultaneously but different sets of nodes cannot simultaneously be manipulated in combination with different sets of faces. Finally, manipulating enumerated subsets can also incur small a computational burden due to the work involved in finding the mesh entities within a given subset.

1.7.2 Subset Inclusion Lattice

VisIt relates all possible subsets in a database using what is called a Subset Inclusion Lattice (SIL). Ultimately the subsets in a database are cells that can be grouped into different categories such as material region, domain, patch, refinement level, etc. Each category has some number of possible values when taken together form a collection. A collection lets you group the subsets that have different values but are still part of the same category. For example, the mesh shown in Figure 1.182 is broken down into domain and material categories and there are 3 domain subsets in the domain category. VisIt uses the SIL to remove pieces of a database from a plotted visualization by turning off bottom level subsets that are arrived at through turning off members in various collections or turning off entire collections. When various subsets have been turned off in a SIL, the collective on/off state for each subset is known as a SIL restriction.

Fig. 1.182: Whole mesh divided up into domains and materials
1.7.3 Using the Subset Window

Users can open the Subset Window, shown in Figure 1.183, by clicking on the Subset option in the Main Window’s Controls menu or by clicking on the Subset Venn Diagram-looking icon next to the name of a plot in the Plot list. VisIt’s Subset Window shows the relationships between subsets and provides controls that allow users to turn subsets on and off.

![Subset Window](image)

Fig. 1.183: Subset window

The Subset Window initially has three panels that display the sets associated with mesh of the currently active plot. The window will grow more panels to the right, when necessary as the subset structure of a mesh is browsed. Each successive panel shows the next level of subsets in the mesh. The leftmost panel contains the top level (e.g. whole) set for the whole mesh of the currently active plot. The top level or whole set, which includes all subsets in the mesh, is usually decomposed into the various kinds of subsets described in the section What is a subset?. For example, it can be decomposed by material, processor domain, etc. The various ways in which a database can be decomposed are called subset categories. The subset categories will vary depending on how the data producer(s) create the database(s).

Browsing subsets

To browse the subsets for a database, users must first have created a plot. Once a plot is created and selected, open the Subset Window. The left panel in the Subset Window contains the database’s top level set and may also list some subset categories. Some simple databases don’t include any subset and so VisIt will not show any subsets for them. To start browsing the available subsets, users can click on one of the subset categories to display the subsets in that category. For instance, clicking on a “Material” subset category will list all of the mesh’s materials in the next panel to the right. The materials are subsets of the top level set. Double clicking on a set or clicking on an expand arrow lists any subset categories that can be used to further break down the set.
Changing a SIL restriction

Each set in the Subset Window has a small check box next to it that allows users to turn the set on or off. The check box not only displays whether a set is on or off, but it also displays whether or not a set is partially on. When a set is partially on, it means that at least one (but not all) of the subsets it contains is turned on. When a set is partially on, its check box shows a small slash instead of a check or an empty box. Uncheck the check box next to a set name to turn the set off.

Suppose a user has a database that contains 4 domains, numbered 1 through 4. If the user wants to turn off the subset named “domain1”, first click on the “domains” category to list the subsets in that category. Next, click the check box next to the subset name “domain1” and click the Apply button. The result of this operation, shown in Figure 1.184, removes the “domain1” subset from the visualization. Note that the Subset Window “domain1” set’s check box is unchecked and the top level set’s check box has a slash through it to show that some subsets are turned off.

Creating complex subsets

When visualizing a database, it is often useful to look at combinations of subsets. Suppose a user has a database that has two subset categories: “Materials”, and “Domains” and that the user wants to turn off the “domain1” subset but also wants to turn off a material in the “domain4” subset. Users can do this by clicking on the “Domains” category and then unchecking the “domain1” check box in the second panel. Now, to turn off a material in the “domain4” subset, the user clicks on the “domains” category in the left panel. Next, double-click on the “domain4” subset in the second panel. Select the “Materials” subset category in the second panel to make the third panel list the materials that can be removed from the “domain4” subset. Turning off a couple materials from the list in the third panel will only affect
Fig. 1.185: Subset window with one subset removed.
the “domain4” subset. An example of a complex subset selection is shown in Figure 1.186 and the state of the Subset window is shown in Figure 1.187.

Turning multiple sets on and off

When databases contain large numbers of subsets, it is convenient to turn many of them on and off at the same time. Users can select ranges of subsets by clicking on the name of a subset using the left mouse button and dragging the mouse up or down to other subsets in the list while still holding down the left mouse button. Alternatively, users can click on a subset to select it and then click on another subset while holding down the Shift key to select all of the subsets in the middle. Finally, users can select a group of multiple nonconsecutive subsets by holding down the Ctrl key while clicking on the subsets to be selected.

Once a group of subsets has been selected, the buttons at the bottom of the pane can be used to adjust the selection in various ways. The top button applies an action to all of the sets in the pane regardless of how they have been selected. Each action button has three possible actions: Turn on, Turn off, and Reverse. Users can change the action for an action button by clicking on the down-arrow button to its right and selecting one of the Turn on, Turn off, and Reverse menu options. When the Turn on action is used, the appropriate subsets will be turned on. When the Turn off action is used, they will be turned off. When the Reverse action is used, the on/off state of the sets will be reversed (or toggled).

1.7.4 Material Interface Reconstruction

Many data producers create meshes with material subsets. In some cases, materials include mixing where multiple materials exist within each mesh zone and in other cases materials are clean in each zone (e.g. no mixing).

The materials are often used to break meshes into subsets that correspond to physical parts of a model. Materials are commonly stored out as a list of materials and material volume fractions for each cell in the database. If a cell has only one material then it is a clean cell. If a cell has more than one material, it has some fraction of each of the materials and it is known as a mixed cell. The fraction of the material in a cell if accounted for by the material volume fraction. Since only the volume fractions are known, and not any information about how the materials are distributed in the cell, VisIt must make a guess at the location of the boundaries between materials.

Material interface reconstruction (MIR) is the process of constructing the boundaries between materials, in cells with mixed materials, from the material volume fraction information stored in the database. MIR is not usually needed when
Fig. 1.187: Subset window for complex subset example.
you visualize the entire database but when you start to subset the database by removing materials, VisIt must perform MIR to remove only the parts of the database that contain the material to be removed. Without MIR, visualizations containing mixed materials would be very blocky when materials are removed. VisIt’s MIR algorithms have several settings, which you can change using the controls in the Material Reconstruction Options Window (see Figure 1.188), that influence the appearance of the final plot. To open the Material Reconstruction Options Window, click on the Materials option in the Main Window’s Controls menu.

![Material Reconstruction Options Window](image)

Fig. 1.188: Material Reconstruction Options Window

### Choosing a MIR algorithm

VisIt currently provides three MIR algorithms: Tetrahedral, Zoo-based, and Isovolume. Each MIR algorithm reconstructs the interfaces between materials using a different method and one method may work better or worse than another based on the complexities of the input data. You can select your preferred MIR algorithm by choosing from the Algorithm combo box in the Material Reconstruction Options Window. Note that if you have plots that have already been generated, the new material options will not take effect for those plots unless you clear the plots and redraw them.

The Tetrahedral algorithm breaks up each mixed cell into tetrahedra and computes the interfaces through the original cell by recursively subdividing the tetrahedra until the approximate volume fractions, which determine the amount of material in a cell, are reached. The Tetrahedral MIR algorithm results in a high cell count so it is not often used.

The Zoo-based MIR algorithm breaks up mixed cells into elements based on supported finite elements (tetrahedra, prisms, pyramids, wedges, cubes). The resulting reconstruction results in far fewer cells than other methods while also producing superior material boundaries. The Zoo-based algorithm is the default because of the quality of the material boundaries and because the zoo-based cell representation saves memory and ultimately leads to faster pipeline execution due to the smaller cell count.

The Isovolume algorithm computes an isovolume containing portions of cells that contain a user-specified fraction of materials. The Isovolume approach to MIR does not generally produce very good looking results since there are gaps where several materials join. However, the Isovolume algorithm does do a better job than the other two algorithms when it comes to finding cells that contain very small fractions of a certain material when the cells are heavily mixed. If you use the Isovolume MIR algorithm, you can specify the amount of material required to be present before VisIt creates a material interface for a material. The amount of material is specified as a volume fraction in the range \([0,1]\).
Specifying smaller values in the **Volume Fraction for Isovolume** text field will find materials that may be omitted by other MIR algorithms.

### Finding materials with low volume fractions

When mixed cells contain several materials, the Zoo-based MIR algorithm will often omit materials with very small volume fractions, leaving only the materials in the mixed cell that had the highest volume fractions. If you want to plot materials in mixed cells where the volume fraction is very small then you can try using the Isovolume MIR algorithm since it can be used to find materials whose volume fractions are above a user-specified threshold. Figure 1.191 shows an example of a dataset containing five mixed materials where the first four mixed materials are roughly equal in the amount of area that they occupy. The fifth material has a volume fraction that never exceeds 0.08 so it is omitted by the Zoo-based MIR algorithm due to its comparatively low volume fraction. To ensure that VisIt plots the fifth material, the Isosurface MIR algorithm is used with a **Volume Fraction for Isovolume** setting of 0.02. Using the Isovolume MIR algorithm with a low **Volume Fraction for Isovolume** value can find materials that have been distributed into many heavily mixed cells.

### Simplifying heavily mixed cells

VisIt provides the **Simplify heavily mixed cells** check box in the **Material Reconstruction Options Window** so you can tell VisIt to throw away information materials that have low volume fractions. When you tell VisIt to omit these materials, VisIt will use less memory and will also finish MIR faster because fewer materials have to be considered. The **Simplify heavily mixed cells** check box is especially useful for databases where most of the cells are mixed or where there are many cells that contain tens of materials. When you tell VisIt to simplify heavily mixed cells, you can tell VisIt how many of the top materials to keep from each cell by entering a new number of materials into the **Maximum materials per zone** text field. By keeping the N top materials, VisIt will be sure to preserve the features that are contributed by the most dominant materials.
Fig. 1.190: Zoo-based MIR vs. Isovolume MIR

Fig. 1.191: Materials with low volume fractions can be found with the isosurface MIR algorithm
**Smooother material boundary interfaces**

VisIt’s material interface reconstruction algorithm sometimes produces small, pointy outcroppings on reconstructed material boundaries next to where clean cells are located. Since these are often distracting features when looking at a visualization, VisIt provides an interface smoothing option that allows materials to bleed a little bit into clean cells to improve how they look when their material boundary is reconstructed. Figure 1.192 shows a plot that has not been smoothed next to a plot that has been smoothed. To enable interface smoothing, check the **Enable interface smoothing** check box. Note that changing this setting will not affect plots that have already been generated. If you want to make your current plots regenerate with smoother interfaces, you must also clear them out of the visualization window by choosing the **Plots** option from the **Clear** submenu located in the **Main Window’s Windows** menu.

![Fig. 1.192: Effect of material interface smoothing](image)

**Forcing material interface reconstruction**

VisIt tries to minimize the amount of work that it must do to generate a plot so that it can be done quickly. Sometimes databases have variable information for each material in a cell instead of just having a single value for each cell or node. Because the variable is defined for each material in the cell, these variables are known as mixed variables. VisIt tends to just plot the value for the entire cell since it is more work to go through the material interface reconstruction (MIR) stage, which is usually only done when removing material subsets but is required to plot mixed variables correctly. You can force VisIt to always do MIR by checking the **Force interface reconstruction** check box. This will make mixed variables plot correctly even when you are not removing any material subsets.

**Mixed variables**

Some simulations write out multiple scalar values for cells that contain mixed materials so each material in the cell can have its own scalar value. Once a cell has undergone MIR, it is split into multiple cells if the original cell contained more than one material. Each split cell gets its corresponding scalar value from the original mixed variable data. The resulting plot can then display each split cell’s actual value, taking into account the material boundaries. Suppose you are simulating the interaction between hot lava and ice and you have a material interface that happens to cross in the middle of a cell. Obviously each material in the cell has its own temperature. Plotting mixed variables allows the visualization to more faithfully depict the material boundaries while preserving the actual data so the multiple mix
values do not have to be averaged in the cell (see Figure 1.193). Note that VisIt does not use mixed variable values for variables that have them unless the Force interface reconstruction check box is enabled because most scalar fields are not mixed variables and automatically performing MIR can be expensive. If your scalars are mixed variables and you want to visualize them as such, be sure to enable the Force interface reconstruction check box.

![Fig. 1.193: Mixed variables can improve a visualization](image)

### 1.7.5 Species

VisIt adds species, which are components of materials, to the SIL when they are defined by the data producer. Air is a common material in simulations since many things in the real world are surrounded by air. The chemical composition of air on Earth is roughly 78% Nitrogen, 21% oxygen, 1% Argon. One can say that if air is a material then it has species: Nitrogen, Oxygen, and Argon with mass fractions 78%, 21%, 2%, respectively. Suppose one of the calculated quantities in a database with the aforementioned air material is atmospheric temperature. Now suppose that we are examining one cell that contains only the air material from the database and its atmospheric temperature is 100 degrees Fahrenheit. If we wanted to know how much the Nitrogen contributed to the atmospheric temperature, we could multiply its concentration of 78% times the 100 degrees Fahrenheit to yield: 78 degrees Fahrenheit. Species are often used to track chemical composition of materials and their effects on various calculated quantities.

When species are defined, VisIt creates a scalar variable called Species and it is available in the variable menus for each plot that can accept scalar variables. The Species variable is a cell-centered scalar field defined over the whole mesh. When all species are turned on, the Species variable has the value of 1.0 over the entire mesh. When species are turned off, the Species variable is set to 1.0 minus the mass fraction of the species that was turned off. Using the previous example, if we plotted the Species variable and then turned off the air material’s Nitrogen species, we would be left with only Oxygen’s 21% and Argon’s 1% so the species variable would be reduced to 22% or 0.22. When species are turned off, the amount of mass left to be multiplied by the plotted variable drops so the plotted variable’s value in turn drops.

VisIt adds species to the SIL as a category that contains the various chemical constituents for all materials that have species. Since species are handled using the SIL, one can use VisIt’s Subset Window to turn off species. Turning off species has quite a different effect than turning off entire materials. When materials are turned off, they no longer appear in the visualization. When species are turned off, no parts of the visualization disappear but the plotted data values may change due to drops in the Species variable.
Fig. 1.194: Species variable
**Plotting species**

VisIt provides the Species scalar variable so users can plot or create expressions that involve species. If the user creates a Pseudocolor plot of the Species variable, the resulting plot will have a constant value of 1.0 over the entire mesh because when no species have been removed, they all sum to 1.0. Once species are removed by turning off species subsets in the Subset Window, the plotted value of Species changes, causing plots that use it to also change. If all but one species are removed, the plots that use the Species variable will show zero for all areas that do not contain the one selected species (see Figure 1.195). For example, if a user had air for a material and then removed every species except for oxygen, the plots that use the Species variable would show zero for every place that had no oxygen.

![Figure 1.195: Plots of materials and species](image)

**Turning off species**

VisIt adds species information to the SIL as new subsets under a category called: Species. Since species are part of the SIL, users can use the Subset Window (see Figure 1.196) to turn off species. To access the list of species, select the Species category under the whole mesh. Once the Species category is clicked, the secondpane in the Subset Window is populated with the species for all materials. Users can turn off the species that are not needed to look at by clicking off the check box next to the name of the species subset. When the user applies these changes, the values for the Species variable are recalculated to include only the mass fractions for the species that are still turned on.

**1.8 Quantitative Analysis**

Simulation data must often be compared to experimental data so VisIt provides a number of features that allow quantitative information to be extracted from simulation databases. This chapter explains how to visualize derived variables created with expressions and query information about a database. This chapter also explains VisIt’s Pick, Query and Lineout capabilities which allow users to compute highly sophisticated quantitative, as opposed to visual, results.

**1.8.1 Expressions**

**Danger:** Confirm the text here adequately characterizes that an expression has value everywhere over the whole mesh it is defined on. Its a field.

Scientific simulations often keep track of several dozen variables as they run. However, only a small subset of those variables are usually written to a simulation database to save disk space. Sometimes variables can be derived from
Fig. 1.196: Turning off species in the Subset Window
other variables using a variable expression. **VisIt** provides variable expressions to allow scientists to create derived variables using variables that are stored in the database. Expressions are extremely powerful because they allow users to analyze new data without necessarily having to rerun a simulation. Variables created using expressions behave just like variables stored in a database; they appear in menus where database variables appear and can be visualized like any other database variable.

### Expression Window

**VisIt** provides an **Expression Window**, shown in Figure 1.197, that allows users to create new variables that can be used in visualizations. Users can open the **Expression Window** by clicking on the **Expressions** option in the **Main Window’s Controls** menu. The **Expression Window** is divided vertically into two main areas with the **Expression list** on the left and the **Definition** area on the right. The **Expression list** contains the list of expressions. The **Definition** area displays the definition of the expression that is highlighted in the **Expression list** and provides controls to edit the expression definition.

![Expression Window](image)

**Fig. 1.197: Expression Window**

Expressions in **VisIt** are created either manually by the user or automatically by various means including...

- Preferences
  - Mesh quality expressions
  - Time derivative expressions
  - Vector magnitude expressions
- GUI wizards
By default, the **Expression list** will display only those expressions created manually by the user. A check box near the bottom of the **Expression list** controls the display of automatically created expressions. When this box is checked, the **Expression list** will also include expressions created automatically by **Preferences** and **Databases** but not expressions created automatically by **GUI wizards** or **Operators**.

### Creating a new expression

Users can create a new expression by clicking on the **Expression Window’s New** button. When the user clicks on the **New** button, **VisIt** adds a new expression and shows its new, empty definition in the **Definitions** area. The initial name for a new expression is “unnamed” followed by some integer suffix. As the user types a new name for the expression into the **Name** text field, the expression’s name in the **Expression list** will update.

Each expression also has a **Type** that specifies the type of variable the expression produces. The available types are:

- Scalar
- Vector
- Tensor
- Symmetric Tensor
- Array
- Curve

Users must be sure to select the appropriate type for any expression they create. The selected type determines the menu in which the variable appears and subsequently the plots that can operate on the variable.

To edit an expression’s definition, users can type a new expression comprised of constants, variable names, and even other **VisIt** expressions into the **Definition** text field. The expression definition can span multiple lines as the **VisIt** expression parser ignores whitespace. For a complete list of **VisIt**’s built-in expressions, refer to the table in section **Built-in expressions**. Users can also use the **Insert Function…** menu, shown in Figure 1.198, to insert any of **VisIt**’s built-in expressions directly into the expression definition. The list of built-in expressions is divided into certain categories as shown by the structure of the **Insert Function…** menu.

In the example shown in Figure 1.198, the **Insert Function…** operation inserted a sort of *template* for the function giving some indication of the argument(s) to the function and their meanings. Users can then simply edit those parts of the function template that need to be specified.

In addition to the **Insert Function…** menu, which lets users insert built-in functions into the expression definition, **VisIt**’s **Expression Window** provides an **Insert Variable…** menu that allows users to insert variables from the active database into the expression definition. The **Insert Variable…** menu, shown in Figure 1.199, is broken up into Scalars, Vectors, Meshes, etc. and has the available variables under the appropriate heading so they are easy to find.

Some variables can only be expressed as very complex expressions containing several intermediate subexpressions that are only used to simplify the overall expression definition. These types of subexpressions are seldom visualized on their own. If users want to prevent them from being added to the **Plot** menu, turn off the **Show variable in plot menus** check box.

### Deleting an expression

Users can delete an expression by clicking on it in the **Expression list** and then clicking on the **Delete** button. Deleting an expression removes it from the list of defined expressions and will cause unresolved references for any other
Fig. 1.198: Expression Window’s Insert Function… menu
Fig. 1.199: Expression Window’s Insert Variable… menu
expressions that use the deleted expression. If a plot uses an expression with unresolved references, VisIt will not be able to generate it until the user resolves the reference.

Expression grammar

VisIt allows expressions to be written using a host of unary and binary math operators as well as built-in and user-defined functions. VisIt’s expressions follow C-language syntax, although there are a few differences. The following paragraphs detail the syntax of VisIt expressions.

Math operators

These include use of +, -, *, /, ^ as addition, subtraction, multiplication, division, and exponentiation as infix operators, as well as the unary minus, in their normal precedence and associativity. Parentheses may be used as well to force a desired associativity.

Examples: a+b^-c (a+b)*c

Constants

Scalar constants include floating point numbers and integers, as well as booleans (true, false, on, off) and strings.

Examples: 3e4 10 “mauve” true false

Vectors

Expressions can be grouped into two or three dimensional vector variables using curly braces.

Examples: {xc, yc} {0,0,1}

Lists

Lists are used to specify multiple items or ranges, using colons to create ranges of integers, possibly with strides, or using comma-separated lists of integers, integer ranges, floating points numbers, or strings.

Examples: [1,3,2] [1:2, 10:20:5, 22] [silver, gold] [1.1, 2.5, 3.9] [level1, level2]

Identifiers

Identifiers include function names, defined variable and function names, and file variable names. They may include alphabetic characters, numeric characters, and underscores in any order. Identifiers should have at least one non-numeric character so that they are not confused with integers, and they should not look identical to floating point numbers such as 1e6.

Examples: density x y z 3d_mesh
Functions

These are used for built in functions, but they may also be used for functions/macros defined by the user. They take specific types and numbers of arguments within the parentheses, separated by commas. Some functions may accept named arguments in the form identifier=value.

Examples: \( \sin\left(\frac{\pi}{2}\right) \) cross(vec1, \{0,0,1\}) my_xform(mesh1) subselect(materials=[a,b])

Database variables

These are like identifiers, but may also include periods, plus, and minus characters. A normal identifier will map to a file variable when it is not defined as another expression. To force variables that look like integers or floating point numbers to be interpreted as variable names, or to force variable names which are defined by another expression to map to a variable in a file, they should be enclosed with < and >, the left and right carats/angle brackets. Note that quotation marks will cause them to be interpreted as string constants, not variable names. In addition, variables in files may be in directories within a file, so they may include slashes in a path when in angle brackets.

Examples: density <pressure> <a.001> <a.002> <domain1/density>

Databases

A database specification looks similar to a database variable contained in angle brackets, but it is followed by a colon before the closing angle bracket, and it may also contain extra information. A database specification includes a file specification possibly followed a machine name, a time specification by itself, or a file/machine specification followed by a time specification. A file specification is just a file name with a path if needed. A machine specification is an at-sign @ followed by a host name. A time specification looks much like a list in that it contains integer numbers or ranges, or floating point numbers, separated by commas and enclosed in square brackets. However, it may also be followed by a letter c, t, or i to specify if the time specification refers to cycles, times, or indices, respectively. If no letter is specified, then the parser guesses that integers refer to cycles, floating point numbers refer to times. There is also an alternative to force indices which is the pound sign # after the opening square bracket.

Examples: </dir/file:> <file@host.gov:> <\[#0\]:> <file[1.234]:> <file[000, 023, 047]:> <file[10]c:>

Qualified file variables

Just like variables may be in directories within a file, they may also be in other timesteps within the same database, within other databases, and even within databases on other machines. To specify where a variable is located, use the angle brackets again, and prefix the variable name with a database specification, using the colon after the database specification as a delimiter.

Examples: <file:var> </dir/file:/domain/var> <file@192.168.1.1:/var> <\[#0\]:zerocyclevar>

Built-in expressions

**Danger:** Add examples for some of the more complicated cases.

The following table lists built-in expressions that can be used to create more advanced expressions. Unless otherwise noted in the description, each expression takes scalar variables as its arguments.
Arithmetic Operator Expressions (Math Expressions)

In binary arithmetic operator expressions, each operand must evaluate to the same type field. For example, both must evaluate to a scalar field or both must evaluate to a vector field.

In addition, if the two expressions differ in centering (e.g. one is zone or cell centered or piecewise-constant over mesh cells while the other is node or point centered or piecewise-linear over mesh cells), VisIt will recenter any node-centered fields to zone centering to compute the expression. This may not always be desirable. When it is not, the recenter() may be used to explicitly control the centering of specific operands in an expression.

Sum Operator (+) \([exprL + exprR]\) Creates a new expression which is the sum of the exprL and exprR expressions.

Difference Operator (−) \([exprL - exprR]\) Creates a new expression which is the difference of the exprL and exprR expressions.

Product Operator (*) \([exprL * exprR]\) Creates a new expression which is the product of the exprL and exprR expressions.

Division Operator (/) \([exprL / exprR]\) Creates a new expression which is the quotient after dividing the exprL expression by the exprR expression.

Division Operator \([\text{divide}(\text{val}_{\text{numerator}}, \text{val}_{\text{denominator}}, \{\text{div}_{\text{by}}\text{zero}_{\text{value}}, \text{tolerance}\})]\) Creates a new expression which is the quotient after dividing the \text{val}_{\text{numerator}} expression by the \text{val}_{\text{denominator}} expression. The div_{by_zero}_{value} is used wherever the \text{val}_{\text{denominator}} is within tolerance of zero.

Exponent Operator (^) \([exprL \^ exprR]\) Creates a new expression which is the product after multiplying the exprL expression by itself exprR times.

Logical AND Operator (&) \([exprL \& exprR]\) Creates a new expression which is the logical AND of the exprL and exprR expressions treating each value as a binary bit field. It is probably most useful for expressions involving integer data but can be applied to expressions involving any type.

Associative Operator () \([\text{expr}0 \text{ OP expr}1]\) Parenthesis, () are used to explicitly group partial results of sub expressions and control evaluation order.

For example, the expression \((a + b) / c\) first computes the sum, \(a+b\) and then divides by \(c\).

Absolute Value Function (abs()) \([\text{abs}(\text{expr}0)]\) Creates a new expression which is everywhere the absolute value if its argument.

Ceiling Function (ceil()) \([\text{ceil}(\text{expr}0)]\) Creates a new expression which is everywhere the ceiling (smallest integer greater than or equal to) of its argument.

Exponent Function (exp()) \([\text{exp}(\text{expr}0)]\) Creates a new expression which is everywhere \(e\) (base of the natural logarithm) raised to the power of its argument.

Floor Function (floor()) \([\text{floor}(\text{expr}0)]\) Creates a new expression which is everywhere the floor (greatest integer less than or equal to) of its argument.

Natural Logarithm Function (ln()) \([\text{ln}(\text{expr}0)]\) Creates a new expression which is everywhere the natural logarithm of its argument.

Base 10 Logarithm Function (log10()) \([\text{log10}(\text{expr}0)]\) Creates a new expression which is everywhere the base 10 logarithm of its argument.

Max Function (max()) \([\text{max}(\text{expr}0, \text{expr}1 [, \ldots])]\) Creates a new expression which is everywhere the maximum among all input variables.

Min Function (min()) \([\text{min}(\text{expr}0, \text{expr}1 [, \ldots])]\) Creates a new expression which is everywhere the minimum among all input variables.
Modulo Function \( \text{mod} () \) \[ \text{mod}(\text{expr0}, \text{expr1}) \] Creates a new expression which is everywhere the first argument, \( \text{expr0} \), modulo the second argument, \( \text{expr1} \).

Random Function \( \text{random} () \) \[ \text{random}(\text{expr0}) \] Creates a new expression which is everywhere a random floating point number between 0 and 1, as computed by \((\text{rand}() \% 1024) / 1024\) where \(\text{rand}()\) is the standard C library \(\text{rand}()\) random number generator. The argument, \(\text{expr0}\), must be a mesh variable. The seed used on each block of the mesh is the absolute domain number.

Round Function \( \text{round} () \) \[ \text{round}(\text{expr0}) \] Creates a new expression which is everywhere the result of rounding its argument.

Square Function \( \text{sqr} () \) \[ \text{sqr}(\text{expr0}) \] Creates a new expression which is everywhere the result of squaring its argument.

Square Root Function \( \text{sqrt} () \) \[ \text{sqrt}(\text{expr0}) \] Creates a new expression which is everywhere the square root of its argument.

Relational, Conditional and Logical Expressions

The \( \text{if}() \) conditional expression is designed to be used in concert with relation and logical expressions. Together, these expressions can be used to build up more complex expressions in which very different evaluations are performed depending on the outcome of other evaluations. For example, the \( \text{if}() \) conditional expression can be used together with one or more relational expressions to create a new expression which evaluates to a dot-product on part of a mesh and to the magnitude of a divergence operator on another part of a mesh.

If Function \( \text{if} () \) \[ \text{if}(\text{exprCondition}, \text{exprTrue}, \text{exprFalse}) \] Creates a new expression which is equal to \(\text{exprTrue}\) wherever the condition, \(\text{exprCondition}\) is non-zero and which is equal to \(\text{exprFalse}\) wherever \(\text{exprCondition}\) is zero.

For example, the expression \( \text{if}(\text{and}(\text{gt}(\text{pressure}, 2.0), \text{lt}(\text{pressure}, 4.0)), \text{pressure}, 0.0) \) combines the \( \text{if} \) expression with the \( \text{gt} \) and \( \text{lt} \) expressions to create a new expression that is equal to \( \text{pressure} \) wherever it is between 2.0 and 4.0 and 0 otherwise.

**Danger:** Confirm relational and logical expressions produce new, boolean valued expression variables which are themselves plottable in VisIt. Their original intent may have been only to be used as args in the IF expression and not so much be plottable outputs in their own right.

Equal Function \( \text{eq} () \) \[ \text{eq}(\text{exprL}, \text{exprR}) \] Creates a new expression which is everywhere a boolean value (1 or 0) indicating whether its two arguments are equal. A value of 1 is produced everywhere the arguments are equal and 0 otherwise.

Greater Than Function \( \text{gt} () \) \[ \text{gt}(\text{exprL}, \text{exprR}) \] Creates a new expression which is everywhere a boolean value (1 or 0) indicating whether \(\text{exprL}\) is greater than \(\text{exprR}\). A value of 1 is produced everywhere \(\text{exprL}\) is greater than \(\text{exprR}\) and 0 otherwise.

Greater Than or Equal Function \( \text{ge} () \) \[ \text{ge}(\text{exprL}, \text{exprR}) \] Creates a new expression which is everywhere a boolean value (1 or 0) indicating whether \(\text{exprL}\) is greater than or equal to \(\text{exprR}\). A value of 1 is produced everywhere \(\text{exprL}\) is greater than or equal to \(\text{exprR}\) and 0 otherwise.

Less Than Function \( \text{lt} () \) \[ \text{lt}(\text{exprL}, \text{exprR}) \] Creates a new expression which is everywhere a boolean value (1 or 0) indicating whether \(\text{exprL}\) is less than \(\text{exprR}\). A value of 1 is produced everywhere \(\text{exprL}\) is less than \(\text{exprR}\) and 0 otherwise.

Less Than or Equal Function \( \text{le} () \) \[ \text{le}(\text{exprL}, \text{exprR}) \] Creates a new expression which is everywhere a boolean value (1 or 0) indicating whether \(\text{exprL}\) is less than or equal to \(\text{exprR}\). A value of 1 is produced everywhere \(\text{exprL}\) is less than or equal to \(\text{exprR}\) and 0 otherwise.

1.8. Quantitative Analysis 207
Equal Function \( \text{ne()} \) \[ \text{ne(exprL, exprR)} \] Creates a new expression which is everywhere a boolean value (1 or 0) indicating whether its two arguments are not equal. A value of 1 is produced everywhere the arguments are not equal and 0 otherwise.

Logical And Function \( \text{and()} \) \[ \text{and(exprL, exprR)} \] Creates a new expression which is everywhere the logical and of its two arguments. Non-zero values are treated as true whereas zero values are treated as false.

Logical Or Function \( \text{or()} \) \[ \text{or(exprL, exprR)} \] Creates a new expression which is everywhere the logical or of its two arguments. Non-zero values are treated as true whereas zero values are treated as false.

Logical Not Function \( \text{not()} \) \[ \text{not(expr0)} \] Creates a new expression which is everywhere the logical not of its argument. Non-zero values are treated as true whereas zero values are treated as false.

Trigonometric Expressions

Arc Cosine Function \( \text{acos()} \) \[ \text{acos(expr0)} \] Creates a new expression which is everywhere the arc cosine of its argument. The returned value is in radians.

Arc Sine Function \( \text{asin()} \) \[ \text{asin(expr0)} \] Creates a new expression which is everywhere the arc sine of its argument. The returned value is in radians.

Arc Tangent Function \( \text{atan()} \) \[ \text{atan(expr0)} \] Creates a new expression which is everywhere the arc tangent of its argument. The returned value is in radians.

Cosine Function \( \text{cos()} \) \[ \text{cos(expr0)} \] Creates a new expression which is everywhere the cosine of its argument. The argument is treated as in units of radians.

Hyperbolic Cosine Function \( \text{cosh()} \) \[ \text{cosh(expr0)} \] Creates a new expression which is everywhere the hyperbolic cosine of its argument. The argument is the hyperbolic angle.

Sine Function \( \text{sin()} \) \[ \text{sin(expr0)} \] Creates a new expression which is everywhere the sine of its argument. The argument is treated as in units of radians.

Hyperbolic Sine Function \( \text{sinh()} \) \[ \text{sinh(expr0)} \] Creates a new expression which is everywhere the hyperbolic sine of its argument. The argument is the hyperbolic angle.

Tangent Function \( \text{tan()} \) \[ \text{tan(expr0)} \] Creates a new expression which is everywhere the tangent of its argument. The argument is treated as in units of radians.

Hyperbolic Tangent Function \( \text{tanh()} \) \[ \text{tanh(expr0)} \] Creates a new expression which is everywhere the hyperbolic tangent of its argument. The argument is the hyperbolic angle.

Degrees To Radians Conversion Function \( \text{deg2rad()} \) \[ \text{deg2rad(expr0)} \] Creates a new expression which is everywhere the conversion from degrees to radians of its argument. The argument should be a variable defined in units of degrees.

Radians To Degrees Conversion Function \( \text{rad2deg()} \) \[ \text{rad2deg(expr0)} \] Creates a new expression which is everywhere the conversion from radians to degrees of its argument. The argument should be a variable defined in units of radians.

Vector and Color Expressions

Vector Compose Operator \( \{\} \) \[\{expr0, expr1, \ldots, exprN-1\}\] Curly braces, \( \{\} \) are used to create a new expression of higher tensor rank from 2 or more expression of lower tensor rank. A common use is to compose several tensor rank 0 expressions (e.g. scalar expressions) into a tensor rank 1 expression (e.g. a vector expression). The component expressions, \( expr0, expr1, \ldots \), must all be the same tensor rank and expression type. For example, they must all be rank 0 (e.g. scalar expressions) or they must all be rank 1 (e.g. vector expressions) of the same number of components. If they are all scalars, the result is a tensor of rank 1 (e.g. a
vector). If they are all vectors, the result is a tensor of rank 2 (e.g. a tensor). The vector compose operator is also used to compose array expressions.

For example, the expression \( \{u, v, w\} \) takes three scalar mesh variables named \( u, v \) and \( w \) and creates a vector mesh variable.

**Vector Component Operator ([])** \([\text{expr}[I]]\) Square brackets, \([I]\), are used to create a new expression of lower tensor rank by extracting a component from an expression of higher tensor rank. Components are indexed starting from 0. If \( \text{expr} \) is a tensor of rank 2, the result will be a tensor of rank 1 (e.g. a vector). If \( \text{expr} \) is a tensor of rank 1, the result will be a tensor of rank 0 (e.g. a scalar). To obtain the \( J \)-th component of the \( I \)-th row of a tensor of rank 2, the expression would be \( \text{expr}[I][J] \).

**Color Function (color())** \([\text{color}(\text{exprR}, \text{exprG}, \text{exprB})]\) Creates a new, RGB vector, expression which defines a color vector where \( \text{exprR} \) defines the red component, \( \text{exprG} \) defines the green component and \( \text{exprB} \) defines the blue component of the color vector. The resulting expression is suitable for plotting with the True-color Plot. The arguments are used to define color values in the range 0...255. Values outside that range are clamped. No normalization is performed. If the arguments have much smaller or larger range than \([0...255]\), it may be appropriate to select a suitable multiplicative scale factor.

**Color4 Function (color4())** \([\text{color4}(\text{exprR}, \text{exprG}, \text{exprB}, \text{exprA})]\) See color(). This function is similar to the color() function but also supports alpha-transparency as the fourth argument, again in the range 0...255.

**Color lookup Function (colorlookup())** \([\text{colorlookup}(\text{expr0}, \text{tabname}, \text{scalmode}, \text{skewfac})]\) Creates a new vector expression that is the color that each value in \( \text{expr0} \) maps to. The \( \text{tabname} \) argument is the name of the color table. The \( \text{expr0} \) and \( \text{tabname} \) arguments are required. The \( \text{scalmode} \) and \( \text{skewfac} \) arguments are optional. Possible values for \( \text{scalmode} \) are 0 (for linear scaling mode), 1 (for log scaling mode) and 2 (for skew scaling mode). The \( \text{skewfac} \) argument is required only for a \( \text{scalmode} \) of 2.

**Cross Product Function (cross())** \([\text{cross}(\text{exprV0}, \text{exprV1})]\) Creates a new vector expression which is the vector cross product created by crossing \( \text{exprV0} \) into \( \text{exprV1} \). Both arguments must be vector expression.

**Dot Product Function (dot())** \([\text{dot}(\text{exprV0}, \text{exprV1})]\) Creates a new scalar expression which is the vector dot product of \( \text{exprV0} \) with \( \text{exprV1} \).

**HSV Color Function (hsvcolor())** \([\text{hsvcolor}(\text{exprH}, \text{exprS}, \text{exprV})]\) See color(). This function is similar to the color() function but takes Hue, Saturation and Value (Lightness) arguments as inputs and produces an RGB vector expression.

**Magnitude Function (magnitude())** \([\text{magnitude}(\text{exprV0})]\) Creates a new scalar expression which is everywhere the magnitude of the \( \text{exprV0} \).

**Normalize Function (normalize())** \([\text{normalize}(\text{exprV0})]\) Creates a new vector expression which is everywhere a normalized vector (e.g. same direction but unit magnitude) of \( \text{exprV0} \).

**Curl Function: curl()** \([\text{curl}(\text{expr0})]\) Creates a new vector expression which is everywhere the curl of its input argument, which must be vector valued. In a 3D context, the result is also a vector. However, in a 2D context the result vector would always be \([0, 0, V]\) so expression instead returns only the \( V \).

**Divergence Function: divergence()** \([\text{divergence}(\text{expr0})]\) Creates a new scalar expression which is everywhere the divergence of its input argument, which must be vector valued.

**Gradient Function: gradient()** \([\text{gradient}(\text{expr0})]\) Creates a new vector expression which is everywhere the gradient of its input argument, which must be scalar. The method of calculation varies depending on the type of mesh upon which the input is defined. See also \( \text{ij_gradient()} \) and \( \text{ijk_gradient()} \).

**IJ_Gradient Function: ij_gradient()** \([\text{ij_gradient}(\text{expr0})]\) No description available.

**IJK_Gradient Function: ijk_gradient()** \([\text{ijk_gradient}(\text{expr0})]\) No description available.

**Surface Normal Function: surface_normal()** \([\text{surface_normal}(\text{expr0})]\) This function is an alias for \( \text{cell_surface_normal()} \).
Point Surface Normal Function: point_surface_normal() [point_surface_normal(expr0)] Like cell_surface_normal() except that after computing face normals, they are averaged to the nodes.

Cell Surface Normal Function: cell_surface_normal() [cell_surface_normal(<Mesh>)] Computes a vector variable which is the normal to a surface. The input argument is a Mesh variable. In addition, this function cannot be used in isolation. It must be used in combination the external surface, first, and the defer expression, second, operators.

Edge Normal Function: edge_normal() [edge_normal(expr0)] No description available.

Point Edge Normal Function: point_edge_normal() [point_edge_normal(expr0)] No description available.

Cell Edge Normal Function: cell_edge_normal() [cell_edge_normal(expr0)] No description available.

Tensor Expressions

Contraction Function: contraction() [contraction(expr0)] Creates a scalar expression which is everywhere the contraction of expr0 which must be a tensor valued expression. The contraction is the sum of pairwise dot-products of each of the column vectors of the tensor with itself as shown in the code snip-it below.

Show/Hide Code for contraction()

```c
// Conceptually it is like as doting each column vector with
// itself and adding the column results

```

Determinant Function: determinant() [determinant(expr0)] Creates a scalar expression which is everywhere the determinant of expr0 which must be tensor valued.

Effective Tensor Function: effective_tensor() [effective_tensor(expr0)] Creates a scalar expression which is everywhere the square root of three times the second principal invariant of the stress deviator tensor, \( \sqrt{3} \cdot J_2 \), where \( J_2 \) is the second principal invariant of the stress deviator tensor. This is also known as the von Mises stress or the Huber-Mises stress or the Mises effective stress.

Show/Hide Code for effective_tensor()

```c
double s11 = vals[0], s12 = vals[1], s13 = vals[2];
double s21 = vals[3], s22 = vals[4], s23 = vals[5];
double s31 = vals[6], s32 = vals[7], s33 = vals[8];

// First invariant of the stress tensor
// aka "pressure" of incompressible fluid in motion
// aka "mean effective stress"
double trace = (s11 + s22 + s33) / 3.;

// components of the deviatoric stress
double dev0 = s11 - trace;
double dev1 = s22 - trace;
double dev2 = s33 - trace;

// The second invariant of the stress deviator
// aka "J2"
```

(continues on next page)
double out2 = 0.5*(dev0*dev0 + dev1*dev1 + dev2*dev2) +
        s12*s12 + s13*s13 + s23*s23;

    // stress deviator
    out2 = sqrt(3.*out2);

Eigenvalue Function: `eigenvalue()` [eigenvalue(expr0)] The `expr0` argument must evaluate to a 3x3 symmetric tensor. The eigenvalue expression returns the eigenvalues of the 3x3 symmetric matrix argument as a vector valued expression where each eigenvalue is a component of the vector. Use the vector component operator, [], to access individual eigenvalues. If a non-symmetric tensor is supplied, results are indeterminate.

Eigenvector Function: `eigenvector()` [eigenvector(expr0)] The `expr0` argument must evaluate to a 3x3 symmetric tensor. The eigenvector expression returns the eigenvectors of the 3x3 matrix argument as a tensor (3x3 matrix) valued expression where each column in the tensor is one of the eigenvectors.

In order to use the vector component operator [], to access individual eigenvectors, the result must be transposed with the `transpose()`, expression function.

For example, if `evecs = transpose(eigenvector(tensor))`, the expression `evecs[1]` will return the second eigenvector.

Inverse Function: `inverse()` [inverse(expr0)] Creates a new tensor expression which is everywhere the inverse of its input argument, which must also be a tensor.

Principal Deviatoric Tensor Function: `principal_deviatoric_tensor()` [principal_deviatoric_tensor(expr0)] Deviatoric stress is the stress tensor which results after subtracting the hydrostatic stress tensor. Hydrostatic stress is a scalar quantity also often referred to as average pressure or just pressure. However, it is often characterized in tensor form by multiplying it through a 3x3 identity matrix.

The `principal_deviatoric_tensor()` expression function creates a new vector expression which is everywhere the principal components of the deviatoric stress tensor computed from the symmetric tensor argument `expr0`. In other words, the eigenvalues of the deviatoric stress tensor.

Potentially, it would be more appropriate to create a new tensor field here with all zeros for off-diagonal elements and the eigenvalues on the main diagonal.

This expression can also be computed by using a combination of the `trace()` and `principal_tensor()` expression functions. The `trace()` (divided by 3) would be used to subtract out hydrostatic stress and the result could be used in the `principal_tensor()` expression to arrive at the same result.

Show/Hide Code for `principal_deviatoric_tensor()`

```c
    double pressure = -(vals[0] + vals[4] + vals[8]) / 3.;
    double dev0 = vals[0] + pressure;
    double dev1 = vals[4] + pressure;
    double dev2 = vals[8] + pressure;

    // double invariant0 = dev0 + dev1 + dev2;
    double invariant1 = 0.5*(dev0*dev0 + dev1*dev1 + dev2*dev2);
    double invariant2 = -dev0*dev1*dev2;
    invariant2 += -2.0 *vals[1]*vals[2]*vals[5];
    invariant2 += dev0*vals[5]*vals[5];
    invariant2 += dev1*vals[2]*vals[2];
    invariant2 += dev2*vals[1]*vals[1];

    double princ0 = 0.;
```
double princ1 = 0.;
double princ2 = 0.;
if (invariant1 >= 1e-100)
{
    double alpha = -0.5*sqrt(27./invariant1)
                    *invariant2/invariant1;
    if (alpha < 0.)
        alpha = (alpha < -1. ? -1 : alpha);
    if (alpha > 0.)
        alpha = (alpha > +1. ? +1 : alpha);
    double angle = acos((double)alpha) / 3.;
    double value = 2.0 * sqrt(invariant1 / 3.);
    princ0 = value*cos(angle);
    angle = angle - 2.0*vtkMath::Pi()/3.;
    princ1 = value*cos(angle);
    angle = angle + 4.0*vtkMath::Pi()/3.;
    princ2 = value*cos(angle);
}

double out3[3];
    out3[0] = princ0;
    out3[1] = princ1;
    out3[2] = princ2;

Principal Tensor Function: principal_tensor() [principal_tensor(expr0)] Creates a new vector expression which is everywhere the principal stress components of the input argument, which must be a symmetric tensor. The principal stress components are the eigenvalues of the stress tensor. So, the vector expression computed here is the same as eigenvalue().

Potentially, it would be more appropriate to create a new tensor field here with all zeros for off-diagonal elements and the eigenvalues on the main diagonal.

Transpose Function: transpose() [transpose(expr0)] Creates a new tensor expression which is everywhere the transpose of its input argument which must also be a tensor. The first row vector in the input becomes the first column vector in the output, etc.

Tensor Maximum Shear Function: tensor_maximum_shear() [tensor_maximum_shear(expr0)] Creates a new Scalar expression which is everywhere the maximum shear stress as defined in J.C. Ugural and S.K. Fenster “Advanced Strength and Applied Elasticity”, Prentice Hall 4th Edition, page 81. the specific mathematical operations of which are shown in the code snip-it below.

Show/Hide Code for tensor_maximum_shear()
double *vals = in->GetTuple9(i);
double s11 = vals[0], s12 = vals[1], s13 = vals[2];
double s21 = vals[3], s22 = vals[4], s23 = vals[5];
double s31 = vals[6], s32 = vals[7], s33 = vals[8];

// Hydro-static component
double pressure = (s11 + s22 + s33) / 3.;

// Deviatoric stress components
double dev0 = s11 - pressure;
double dev1 = s22 - pressure;
double dev2 = s33 - pressure;

// double invariant0 = dev0 + dev1 + dev2;
// Second invariant of stress deviator
double invariant1 = 0.5*(dev0*dev0 + dev1*dev1 + dev2*dev2);
invariant1 += s12*s12 + s13*s13 + s23*s23;

// Third invariant of stress deviator
double invariant2 = -dev0*dev1-dev2;
invariant2 += -2.0*s12*s13+s23;
invariant2 += dev0*s23+s23;

(continues on next page)
invariant2 += dev1*s13*s13;
invariant2 += dev2*s12*s12;

// Cubic roots of the characteristic equation
// http://mathworld.wolfram.com/CubicFormula.html
double princ0 = 0.;
double princ2 = 0.;
if (invariant1 >= 1e-100)
{
    double alpha = -0.5*sqrt(27./invariant1)
    *invariant2/invariant1;
    if (alpha < 0.)
        alpha = (alpha < -1. ? -1 : alpha);
    if (alpha > 0.)
        alpha = (alpha > +1. ? +1 : alpha);
    double angle = acos((double)alpha) / 3.;
    double value = 2.0 * sqrt(invariant1 / 3.);
    princ0 = value*cos(angle);
    // Displace the angle for princ1 (which we don't calculate)
    angle = angle - 2.0*vtkMath::Pi()/3.;
    // Now displace for princ2
    angle = angle + 4.0*vtkMath::Pi()/3.;
    princ2 = value*cos(angle);
}

// set the output value

Trace Function: trace() [trace(expr0)] Creates a new scalar expression which is everywhere the trace of expr0 which must be a 3x3 tensor. The trace is the sum of the diagonal elements.

Viscous Stress Function: viscous_stress() [viscous_stress(expr0)] Creates a new tensor expression which is everywhere the viscous stress. The key difference between viscous stress and elastic stress (which is the kind of stress many of the other functions here deal with) is that viscous stress is related to the rate of change of deformation whereas elastic stress is related to the amount of deformation. These two are related in the same way velocity and distance are related.

The argument here, expr0 is a vector valued velocity. In addition, the current implementation of this function works only for 2D, structured gridded meshes.

Show/Hide Code for viscous_stress()

\[
\begin{array}{l}
\text{div} = 1.0 / (dx[0] *dy[1] - dx[1] *dy[0] + tiny); \\
\text{dvx}[0] = \text{div} * (du[0] *dy[1] - du[1] *dy[0]); \\
\end{array}
\]

// create the tensor

// if rz mesh include extra divergence term
if (rz_mesh)
{
}

// diag terms
vstress[0] = 1/3.0 * (2.0 * dvx[0] - dvy[1] - cyl_term);
vstress[8] = 0.0;

// other terms
vstress[1] = 0.5 * (dvy[0] + dvx[1]);
vstress[2] = 0.0;
vstress[5] = 0.0;

// use symm to fill out remaining terms
vstress[3] = vstress[1];
vstress[6] = vstress[2];
vstress[7] = vstress[5];
}

Array Expressions

Array Compose Function: array_compose() [array_compose(expr0, expr1, ..., exprN-1)]
Create a new array expression variable which is everywhere the array composition of its arguments, which all must be scalar type. An array mesh variable is useful when using the label plot or when doing picks and wanting pick values to always return a certain selected set of mesh variables. But, all an array mesh variable really is is a convenient container to hold a group of individual scalar mesh variables. Each argument to the array_compose expression must evaluate to a scalar expression and all of the input expressions must have the same centering. Array variables are collections of scalar variables that are commonly used with certain plots to display the contents of multiple variables simultaneously. For example, the Label plot can display the values in an array variable.

Array Compose With Bins Function: array_compose_with_bins() [array_compose_with_bins(expr0, ..., exprN-1, b0, ..., bn-1)] This expression combines two related concepts. One is the array concept where a group of individual scalar mesh variables are grouped into an array variable. The other is a set of coordinate values (you can kinda think of as bin boundaries), that should be used by VisIt for certain kinds of operations involving the array variable. If there are N variables in the array, expr0, expr1, and so on, there are N+1 coordinate values (or bin boundaries), b0, b1. When such a variable is picked using one of VisIt’s pick operations, VisIt can display a bar-graph. Each bar in the bar-graph has a height determined by the associated scalar mesh variable (at the picked point) and a width determined by the associated bin-boundaries.

For example, suppose a user had an array variable, foo, composed of 5 scalar mesh variables, a1, a2, a3, a4, and a5 like so...
array_compose_with_bins(a1, a2, a3, a4, a5, 0, 3.5, 10.1, 10.7, 12, 22)

For any given point on a plot, when the user picked foo, there are 5 values returned, the value of each of the 5
scalar variable members of foo. If the user arranged for a pick to return a bar-graph of the variable using the bin-boundaries, the result might look like…

![Bar graph](image)

Fig. 1.200: Bar graph created from picking an array variable created with `array-compose-with-bins()`

**Array Decompose Function:** `array_decompose()`  
`array_decompose(Arr, Idx)` Creates a new *scalar* expression which is everywhere the scalar member of the *array* input argument at index `Idx` (numbered starting from zero).

**Array Decompose 2D Function:** `array_decompose2d()`  
`array_decompose2d(expr)` No description available.

**Array Component-wise Division Function:** `array_componentwise_division()`  
`array_componentwise_division(<Array>,<Divisor>)` Returns a new *array* variable which is the old input `<Array>` variable with each of its components divided by the `<Divisor>`.

**Array Component-wise Product Function:** `array_componentwise_product()`  
`array_componentwise_product(<Array>,<Multiplier>)` Returns a new *array* variable which is the old input `<Array>` variable with each of its components multiplied by the `<Multiplier>`.

**Array Sum Function:** `array_sum()`  
`array_sum(<Array>)` Returns a new *scalar* variable which is the sum of the `<Array>` components.

**Material Expressions**

**Dominant Material Function:** `dominant_mat()`  
`dominant_mat(<Mesh>)` Creates a new *scalar* expression which is for every mesh cell/zone the material having the largest volume fraction.

**Material Error Function:** `materror()`  
`materror(<Mat>,[Const,Const...])` Creates a new *scalar* expression which is everywhere the difference in volume fractions as stored in the database and as computed by
VisIt’s material interface reconstruction (MIR) algorithm. The `<Mat>` argument is a *material variable* from a database and the `Const` argument is one of the material names as an quoted string or a material number as an integer. If multiple materials are to be selected from the *material variable*, enclose them in square brackets as a list.

Examples...

```python
materror(materials, 1)
materror(materials, [1,3])
materror(materials, "copper")
materror(materials, ["copper", "steel")
```

Material Volume Fractions Function: **matvf()**  
```
matvf(<Mat>, [Const, Const, ...])
```

Creates a new scalar expression which is everywhere the sum of the volume fraction of the specified materials within the specified material variable. The `<Mat>` argument is a *material variable* from a database and the `Const` argument(s) identify one or more materials within the *material variable*.

Examples...

```python
matvf(materials, 1)
matvf(materials, [1,3])
matvf(materials, "copper")
matvf(materials, ["copper", "steel")
```

**NMats Function:**  
```
mats(<Mat>)
```

Creates a new scalar expression which for each mesh cell/zone is the number of materials in the cell/zone. The `<Mat>` argument is a *material variable* from a database.

**Specmf Function:**  
```
specmf(<Spec>, <MConst>, [Const, Const, ...])
```

Performs the analogous operation to `matvf` for species mass fractions. The `<Spec>` argument is a *species variable* from a database. The `<MConst>` argument is a specific material within the *species variable*. The `Const` argument(s) identify which species within the *species variable* to select.

Examples:

```python
specmf(species, 1, 1)
specmf(species, "copper", 1)
specmf(species, "copper", [1,3])
```

**Value For Material Function:**  
```
value_for_material(<Var>, <Const>)
```

Creates a new scalar expression which is everywhere the material-specific value of the variable specified by `<Var>` for the material specified by `<Const>`. If variable specified by `<Var>` has no material specific values, the values returned from this function will be just the variable’s values.

### Mesh Expressions

**Area Function:**  
```
area(<Mesh>)
```

See the Verdict Manual.

**cylindrical Function:**  
```
cylindrical(<Mesh>)
```

Creates a new vector variable on the mesh which is the cylindrical coordinate tuple (R,theta,Z) of each mesh node.

**Cylindrical Radius**  
```
cylindrical_radius(<Mesh>)
```

Creates a scalar new variable on the mesh which is the radius component of the cylindrical coordinate (from the Z axis) of each mesh node.

**cylindrical theta Function:**  
```
cylindrical_theta(<Mesh>)
```

Creates a new scalar variable on the mesh which is the angle component of the cylindrical coordinate (around the Z axis from the +X axis) of each mesh node.

**polar radius Function:**  
```
polar_radius(<Mesh>)
```

Creates a new scalar variable on the mesh which is the radius component of the polar coordinate of each mesh node.
polar theta Function: `polar_theta()` Creates a new scalar variable on the mesh which is the theta component of the polar coordinate of each mesh node.

polar phi Function: `polar_phi()` Creates a new scalar variable on the mesh which is the phi component of the polar coordinate of each mesh node.

min coord Function: `min_coord()` No description available.

max coord Function: `max_coord()` No description available.

external node Function: `external_node()` No description available.

external cell Function: `external_cell()` No description available.

Zoneid Function: `zoneid()` Return a `zone-centered scalar` variable where the value for each zone/cell is local index of a zone, starting from zero, within its domain.

Global Zoneid Function: `global_zoneid()` If global zone ids are specified by the input database, return a `zone-centered scalar` variable where the value for each zone/cell is the `global` index of a zone, as specified by the data producer.

Nodeid Function: `nodeid()` Return a `node-centered scalar` variable where the value for each node/vertex/point is local index of a node, starting from zero, within its domain.

Global Nodeid Function: `global_nodeid()` If global node ids are specified by the input database, return a `node-centered scalar` variable where the value for each node/vertex/point is the `global` index of a node, as specified by the data producer.

Volume Function: `volume()` No description available.

Volume2 Function: `volume2()` No description available.

Revolved Volume Function: `revolved_volume()` No description available.

Revolved Surface Area Function: `revolved_surface_area()` No description available.

Zone Type Function: `zonetype()` Return a `zone centered`, character valued variable which indicates the `shape type` of each zone suitable for being used within the `label` plot. Upper case characters generally denote 3D shapes (e.g. `T` for `tet`) while lower case characters denote 2D shapes (e.g. `t` for `triangle`).

Zone Type Rank Function: `zonetype_rank()` Return a `zone centered`, integer valued variable which indicates the `VTK shape type` of each zone. This expression is often useful with the threshold operator to select only certain shapes within the mesh to be displayed.

Mesh Quality Expressions

VisIt employs the `Verdict Mesh Quality Library` to support a number of expressions related to computing cell-by-cell mesh quality metrics. The specific definitions of the various mesh quality metrics defined by the `Verdict Mesh Quality Library` are amply explained in the `Verdict Manual`. Below, we simply list all the mesh quality metrics and describe in detail only those that are not part of the `Verdict Mesh Quality Library`.

In all cases in the `Mesh Quality Expressions`, the input argument is a `mesh variable` from a database and the output is a `scalar` expression.

Neighbor Function: `neighbor()` See the `Verdict Manual`

Node Degree Function: `node_degree()` See the `Verdict Manual`

degree Function: `degree()` No description available.
Aspect Function: \texttt{aspect()} [aspect(<Mesh>)] See the Verdict Manual
Skew Function: \texttt{skew()} [skew(<Mesh>)] See the Verdict Manual
Taper Function: \texttt{taper()} [taper(<Mesh>)] See the Verdict Manual
Minimum Corner Angle Function: \texttt{min\_corner\_angle()} [min\_corner\_angle(<Mesh>)] See the Verdict Manual
Maximum Corner Angle Function: \texttt{max\_corner\_angle()} [max\_corner\_angle(<Mesh>)] See the Verdict Manual
Minimum Edge Length Function: \texttt{min\_edge\_length()} [min\_edge\_length(<Mesh>)] See the Verdict Manual
Maximum Edge Length Function: \texttt{max\_edge\_length()} [max\_edge\_length(<Mesh>)] See the Verdict Manual
Minimum Side Volume Function: \texttt{min\_side\_volume()} [min\_side\_volume(<Mesh>)] See the Verdict Manual
Maximum Side Volume Function: \texttt{max\_side\_volume()} [max\_side\_volume(<Mesh>)] See the Verdict Manual
Stretch Function: \texttt{stretch()} [stretch(<Mesh>)] See the Verdict Manual
Diagonal Ratio Function: \texttt{diagonal\_ratio()} [diagonal\_ratio(<Mesh>)] See the Verdict Manual
Maximum Diagonal Function: \texttt{max\_diagonal()} [max\_diagonal(<Mesh>)] See the Verdict Manual
Minimum Diagonal Function: \texttt{min\_diagonal()} [min\_diagonal(<Mesh>)] See the Verdict Manual
Dimension Function: \texttt{dimension()} [dimension(<Mesh>)] See the Verdict Manual
Oddy Function: \texttt{oddy()} [oddy(<Mesh>)] See the Verdict Manual
Condition Function: \texttt{condition()} [condition(<Mesh>)] See the Verdict Manual
Jacobian Function: \texttt{jacobian()} [jacobian(<Mesh>)] See the Verdict Manual
Scaled Jacobian Function: \texttt{scaled\_jacobian()} [scaled\_jacobian(<Mesh>)] See the Verdict Manual
Shear Function: \texttt{shear()} [shear(<Mesh>)] See the Verdict Manual
Shape Function: \texttt{shape()} [shape(<Mesh>)] See the Verdict Manual
Relative Size Function: \texttt{relative\_size()} [relative\_size(<Mesh>)] See the Verdict Manual
Shape and Size Function: \texttt{shape\_and\_size()} [shape\_and\_size(<Mesh>)] See the Verdict Manual
Aspect Gamma Function: \texttt{aspect\_gamma()} [aspect\_gamma(<Mesh>)] See the Verdict Manual
Warpage Function: \texttt{warpage()} [warpage(<Mesh>)] See the Verdict Manual
Maximum Angle Function: \texttt{maximum\_angle()} [maximum\_angle(<Mesh>)] See the Verdict Manual
Minimum Angle Function: \texttt{minimum\_angle()} [minimum\_angle(<Mesh>)] See the Verdict Manual
Minimum Corner Area Function: \texttt{min\_corner\_area()} [min\_corner\_area(<Mesh>)] See the Verdict Manual
Minimum Sin Corner Function: \texttt{min\_sin\_corner()} [min\_sin\_corner(<Mesh>)] See the Verdict Manual
Minimum Sin Corner CW Function: \texttt{min\_sin\_corner\_cw()} [min\_sin\_corner\_cw(<Mesh>)] See the Verdict Manual

1.8. Quantitative Analysis
Face Planarity Function: \texttt{face\_planarity()} \([\texttt{face\_planarity(<Mesh>)}] \) Creates a new expression which is everywhere a measure of how close to planar all the points comprising a face are. This is computed for each face of a cell and the maximum over all faces is selected for each cell. Planarity is measured as the maximum distance from an arbitrary plane defined by the first 3 points of a face of the remaining points of the face. Values closer to zero are better. A triangle face will always have a planarity measure of zero. This mesh quality expression is not part of the Verdict library.

Relative Face Planarity Function: \texttt{relative\_face\_planarity()} \([\texttt{relative\_face\_planarity(<Mesh>)}] \) Performs the same computation as the \texttt{face\_planarity()}, except where each face’s value is normalized by the average edge length of the face.

Comparison Expressions

Comparing variables defined on the same mesh is often as simple as taking their difference. What about comparing variables when they are defined on different meshes? A common example is taking the difference between results from two runs of the same simulation application. Even if the two runs operate on computationally identical meshes, the fact that each run involves its own instance of that mesh means that as far as VisIt is concerned, they are different meshes.

In order to compose an expression involving variables on different meshes, the first step is to map the variables onto a common mesh. The position-based CMFE function and its friend, the connectivity-based CMFE function, \texttt{conn\_cmfe()} are the work-horse methods needed when working with variables from different meshes in the same expression. CMFE is an abbreviation for cross-mesh field evaluation.

The syntax for specifying CMFE expressions can be complicated. Therefore, the GUI supports a wizard to help create them. See the Data-Level Comparisons Wizard for more information. Here, we describe the details of creating CMFE expressions manually.

All of the comparison expressions involve the concepts of a donor variable and a target mesh. The donor variable (e.g. pressure) is the variable to be mapped. The target mesh is the mesh onto which the donor variable is to be mapped. In addition, the term donor mesh refers to the mesh upon which the donor variable is defined.

Position-Based CMFE Function: \texttt{pos\_cmfe()} \([\texttt{pos\_cmfe(<Donor Variable>,<Target Mesh>,<Fill>)} ] \) The \texttt{pos\_cmfe()} function performs the mapping assuming the two meshes, that is the <Target Mesh> and the mesh upon which the <Donor Variable> (e.g. the donor mesh) is defined, share only a common spatial (positional) extent. Its friend, the \texttt{conn\_cmfe()} function is optimized to perform the mapping when the two meshes are also topologically identical. In other words, their coordinate and connectivity arrays are 1:1. In this case, the mapping can be performed with more efficiency and numerical accuracy. Therefore, when it is possible and makes sense to do so, it is always best to use \texttt{conn\_cmfe()}.

We’ll describe the arguments to \texttt{pos\_cmfe()} working backwards from the last.

The last, <Fill> argument is a numerical constant that VisIt will use to determine the value of the result in places on the target mesh that do not spatially overlap with the mesh of the donor variable. Note that if a value is chosen within the range of the donor variable, it may be difficult to distinguish regions VisIt deemed were non-overlapping. On the other hand, if a value outside the range is chosen, it will effect the range of the mapped variable. A common practice is to choose a value that is an extremum of the donor variable’s range. Another practice is to choose a value that is easily distinguishable and then apply a threshold operator to remove those portions of the result. If the Fill argument is not specified, zero is assumed.

Working backwards, the next argument, is the <Target Mesh>. The <Target Mesh> argument in \texttt{pos\_cmfe()} is always interpreted as a mesh within the currently active database. The CMFE expressions are always mapping data from other meshes, possibly in other databases onto the <Target Mesh> which is understood to be in the currently active database. When mapping data between meshes in different databases, the additional information necessary to specify the other database is encoded with a special syntax prepending the Donor Variable argument.
The Donor Variable argument is a string argument of the form:

```
<PATH-TO-DATABASE-FROM-CWD[SSS]MM:VARNAME>
```

consisting of the donor variable’s name and up to three pre-pending sub-strings which may be optionally needed to specify...

1. …the Database (PATH-TO-DATABASE-FROM-CWD) in which the donor variable resides,
2. …the State Id ([SSS]) from which to take the donor variable,
3. …the Modality (MM) by which states are identified in the State Id sub-string.

Depending on circumstances, specifying the Donor-Variable argument to the CMFE functions can get cumbersome. For this reason, CMFE expressions are typically created using the Data-Level Comparisons Wizard under the Controls menu. Nonetheless, here we describe the syntax and provide examples for a number of cases of increasing complexity in specifying where the Donor Variable resides.

When the donor variable is in the same database and state as the target mesh, then only the variable’s name is needed. The optional substrings are not. See case A in the examples below.

When the donor variable is in a different database and the databases do not have multiple time states, then only sub-string 1, above, is needed to specify the path to the database in the file system. The path to the database can be specified using either absolute or relative paths. Relative paths are interpreted relative to the current working directory in which the VisIt session was started. See cases B and C in the examples below.

When the donor variable is in a different database and the databases have multiple states, then all 3 sub-strings, above, are required. The State Id substring is a square-bracket enclosed number used to identify which state from which to take the donor variable. The Modality substring is a one- or two-character moniker. The first character indicates whether the number in the the State Id substring is a cycle (c), a time (t), or an index (i). The second character, if present, is a d character to indicate the cycle, time or index is relative (e.g. a delta) to the current state. For example, the substring [200]c means to treat the 200 as a cycle number in the donor database whereas the the substring [-10]id means to treat the -10 as an (i) index (d) delta. So, [200]c would map the donor at cycle 200 to the current cycle of the target and [-10]id would map the donor at the current index minus 10 to the current index of the target. In particular, the string [0]id is needed to create a CMFE that keeps donor and target in lock step. Note that in cases where the donor database does not have an exact match for the specified cycle or time, VisIt will chose the state with the cycle or time which is closest in absolute distance. For the index modality, if there is no exact match for the specified index, an error results. See cases D-I in the examples below.

Note that the relative form of specifying the State Id is needed even when working with different states within the same database. In particular, to create an expression representing a time derivative of a variable in a database, the key insight is to realize it involves mapping a donor variable from one state in the database onto a mesh at another state. In addition, the value in using the relative form of specifying the State Id of the donor variable is that as the current time is changed, the expression properly identifies the different states of the donor variable instead of always mapping a fixed state.

Examples.

```
# Case A: Donor variable, "pressure" in same database as mesh, "ucdmesh"
# Note that due to a limitation in Expression parsing, the '"0id:' is
# currently required in the donor variable name as a substitute for
# specifying a file system path to a database file. The syntax '"0id:'
# means a state index delta of zero within the active database.
pos_cmfe(["0id:pressure"],<ucdmesh>,1e+15)

# Case B: Donor variable in a different database using absolute path
```
pos_cmfe(/var/tmp/foo.silo:pressure>,<ucdmesh>,le+15)

# Case C: Donor variable in a different database using relative path
pos_cmfe(<foo/bar.silo:pressure>,<ucdmesh>,le+15)

# Case D: Map "p" from wave.visit at state index=7 onto "mesh"
pos_cmfe(<./wave.visit[7]:p>, mesh, le+15)

# Case E: Map "p" from wave.visit at state index current-1 onto "mesh"
pos_cmfe(<./wave.visit[-1]:p>, mesh, le+15)

# Case F: Map "p" from wave.visit at cycle=200 onto "mesh"
pos_cmfe(<./wave.visit[200]:p>, mesh, le+15)

# Case G: Map "p" from wave.visit at cycle-cycle(current)-200 onto "mesh"
pos_cmfe(<./wave.visit[-200]:p>, mesh, le+15)

# Case H: Map "p" from wave.visit at time~1.4 onto "mesh"
pos_cmfe(<./wave.visit[1.4]:p>, mesh, le+15)

# Case I: Map "p" from wave.visit at time~time(current)-0.8 onto "mesh"
pos_cmfe(<./wave.visit[-0.8]:p>, mesh, le+15)

Connectivity-Based CMFE Function: conn_cmfe() [conn_cmfe(<Donor Variable>,<Target Mesh>)] The connectivity-based CMFE is an optimized version of pos_cmfe() for cases where the Target Mesh and the mesh of the Donor Variable are topologically and geometrically identical. In such cases, there is no opportunity for the two meshes to fail to overlap perfectly. Thus, there is no need for the third, <Fill> argument. In all other respects, conn_cmfe() performs the same function as pos_cmfe() except that conn_cmfe() assumes that any differences in the coordinates of the two meshes are numerically insignificant to the resulting mapped variable. In other words, differences in the coordinate fields, if they exist, are not incorporated into the resulting mapping.

Curve CMFE Function: curve_cmfe() [curve_cmfe(<Donor Curve>,<Target Curve>)] The curve-based CMFE performs the same function as pos_cmfe() except for curves. The arguments specify the Target Curve and Donor Curve and the same syntax rules apply for specifying the Donor Curve as for the Donor Variable in pos_cmfe(). However, if curves represent different spatial extents or different numbers of samples or sample spacing, no attempt is made to unify them.

Symmetric Difference By Point Function: symm_point() [symm_point(<Scalar>,<Fill>,[Px,Py,Pz])] Return a new scalar variable which is the symmetric difference of <Scalar> reflected about the point [Px, Py, Pz]. In 2D, Pz is still required but ignored. The <Fill> argument is a numerical constant that VisIt will use to determine the value of the result in places symmetry about the point doesn’t overlap with the donor mesh. This operation involves both the reflection about the point and taking the difference. If the input <Scalar> is indeed symmetric about the point, the result will be a constant valued variable of zero.

Symmetric Difference By Plane Function: symm_plane() [symm_plane(<Scalar>,<Fill>,[Nx,Ny,Nz,Px,Py,Pz])] Return a new scalar variable which is the symmetric difference of <Scalar> reflected about the plane defined by the point [Px, Py, Pz] and normal [Nx, Ny, Nz]. In 2D, the Nz and Pz arguments are still required but ignored. The <Fill> argument is a numerical constant that VisIt will use to determine the value of the result in places symmetry about the plane doesn’t overlap with the donor mesh. This operation involves both the reflection about the plane and taking the difference. If the input <Scalar> is indeed symmetric about the plane, the result will be a constant valued variable of zero.

Symmetric Difference By Transform Function: symm_transform() [symm_transform(<Scalar>,<Fill>,[T00,T01,T02,...,T22])] Return a new scalar variable which is the symmetric difference of <Scalar> reflected through the 3x3 transformation where each point, [Px,Py,Pz], in the mesh supporting
<Scalar> is transformed by the transform coefficients, \([T_{00}, T_{01}, \ldots, T_{22}]\) as shown below. In 2D, all 9 transform coefficients are still required but the last row and column are ignored. The <Fill> argument is a numerical constant that VisIt will use to determine the value of the result in places symmetry through the transform doesn’t overlap with the donor mesh. This operation involves both the transform and taking the difference. If the input <Scalar> is indeed symmetric through the transform, the result will be a constant valued variable of zero.

\[
\begin{bmatrix}
T_{00} & T_{01} & T_{02} \\
T_{10} & T_{11} & T_{12} \\
T_{20} & T_{21} & T_{22}
\end{bmatrix}
\begin{bmatrix}
P_x \\
P_y \\
P_z
\end{bmatrix}
= 
\begin{bmatrix}
T_{00} \cdot P_x + T_{01} \cdot P_y + T_{02} \cdot P_z \\
T_{10} \cdot P_x + T_{11} \cdot P_y + T_{12} \cdot P_z \\
T_{20} \cdot P_x + T_{21} \cdot P_y + T_{22} \cdot P_z
\end{bmatrix}
\]

Evaluate Point Function: eval_point() [eval_point(<Scalar>,<Fill>,[Px,Py,Pz])] Performs only the reflection half of the symm_point() operation. That is, it computes a new scalar variable which is the input <Scalar> reflected through the symmetric point. It does not then take the difference between with the input <Scalar> as symm_point() does.

Evaluate Plane Function: eval_plane() [eval_plane(<Scalar>,<Fill>,[Nx,Ny,Nz,Px,Py,Pz])] Performs only the reflection half of the symm_plane() operation. That is, it computes a new scalar variable which is the input <Scalar> reflected through the symmetric plane. It does not then take the difference between with the input <Scalar> as symm_plane() does.

Evaluate Transform Function: eval_transform() [eval_transform(expr0,<Fill>,[T00,T01, T02...T22])] Performs only the transform half of the symm_transform() operation. That is, it computes a new scalar variable which is the input <Scalar> mapped through the transform. It does not then take the difference between with the input <Scalar> as symm_transform() does.

Image Processing Expressions

conservative smoothing Function: conservative_smoothing() [conservative_smoothing(expr0)] No description available.

Mean Filter Function: mean_filter() [mean_filter(<Scalar>,<Int>)] Return a filtered version of the input scalar variable using the mean filter of width specified by <Int> argument. By default, the filter width is 3 (3x3). The input scalar must be defined on a structured mesh.

Danger: It is not clear how filtering is handled across different domain boundaries.

Median Filter Function: median_filter() [median_filter(expr0)] Return a filtered version of the input scalar variable using a 3x3 median filter. The input scalar must be defined on a structured mesh.

Abel Inversion Function: abel_inversion() [abel_inversion(expr0)] No description available.

Miscellaneous Expressions

Zonal Constant Function: zonal_constant() [zonal_constant(expr0)] Return a scalar, zone-centered field that is everywhere on <Mesh> the constant value <Const>.

Zone Constant Function: zone_constant() [zone_constant(<Mesh>,<Const>)] An alias for zonal_constant()

Cell Constant Function: cell_constant() [cell_constant(expr0)] An alias for zonal_constant()

Nodal Constant Function: nodal_constant() [nodal_constant(<Mesh>,<Const>)] Return a scalar, node-centered field that is everywhere on <Mesh> the constant value <Const>.
Node Constant Function: \texttt{node\_constant()} \hspace{1em} \texttt{[node\_constant(expr0)]} An alias for \texttt{nodal\_constant()}

Point Constant Function: \texttt{point\_constant()} \hspace{1em} \texttt{[point\_constant(expr0)]} An alias for \texttt{nodal\_constant()}

Time Function: \texttt{time()} \hspace{1em} \texttt{[time(expr0)]} Return a \texttt{constant scalar} variable which is everywhere the time of the associated input argument within its time-series.

Cycle Function: \texttt{cycle()} \hspace{1em} \texttt{[cycle(expr0)]} Return an integer \texttt{constant scalar} variable which is everywhere the cycle of the associated input argument within its time-series.

Timestep Function: \texttt{timestep()} \hspace{1em} \texttt{[timestep(expr0)]} Return an integer \texttt{constant scalar} variable which is everywhere the index of the associated input argument within its time-series.

curve domain Function: \texttt{curve\_domain()} \hspace{1em} \texttt{[curve\_domain(expr0)]} No description available.

curve integrate Function: \texttt{curve\_integrate()} \hspace{1em} \texttt{[curve\_integrate(expr0)]} No description available.

curve swapxy Function: \texttt{curve\_swapxy()} \hspace{1em} \texttt{[curve\_swapxy(expr0)]} No description available.

curve Function: \texttt{curve()} \hspace{1em} \texttt{[curve(expr0)]} No description available.

Enumerate Function: \texttt{enumerate()} \hspace{1em} \texttt{[enumerate(<Int-Scalar>,<[Int-List]>)]} Map an integer valued \texttt{scalar} variable to a new set of integer values. If \( K \) is the maximum value in the \texttt{Int-Scalar} input argument, the \texttt{[Int-List]} argument must be a square bracketed list of \( K+i \) integer values. Value \( i \) in the \texttt{Int-Scalar} input argument is used to index the \( i \)th entry in the \texttt{[Int-List]} to produce mapped value.

Map Function: \texttt{map()} \hspace{1em} \texttt{[map(<Scalar>,<[Input-Value-List]>,<[Output-Value-List]>)]} A more general form of \texttt{enumerate()} which supports non-integer input \texttt{scalar} variables and input and output maps which are not required to include all values in the input \texttt{scalar} variable. The \texttt{[Input-Value-List]} and \texttt{[Output-Value-List]} must have the same number of entries. A value in the input \texttt{scalar} variable that matches the \( i \)th entry in the \texttt{[Input-Value-List]} is mapped to the new value at the \( i \)th entry in the \texttt{[Output-Value-List]}. Values that do not match any entry in the \texttt{[Input-Value-List]} are mapped to \( -1 \).

Resample Function: \texttt{resample()} \hspace{1em} \texttt{[resample(<Var>,Nx,Ny,Nz)]} Resample \texttt{<Var>} onto a regular grid defined by taking the \( X, Y \) and for 3D, \( Z \) spatial extents of the mesh \texttt{<Var>} is defined on and taking \( Nx \) samples over the spatial extents in \( X, Ny \) samples over the spatial extents in \( Y \), and, for 3D, \( Nz \) samples over the spatial extents in \( Z \). Any samples that \texttt{miss} the mesh \texttt{<Var>} is defined on are assigned the value \( -\text{FLT\_MAX} \). For 2D, the \( Nz \) argument is still required but ignored.

Recenter Expression Function \texttt{[recenter(expr, ["nodal", "zonal", "toggle"])]} This function can be used to recenter \texttt{expr}. The second argument is optional and defaults to \texttt{"toggle"} if it is not specified. A value of \texttt{"toggle"} for the second argument means that if \texttt{expr} is \texttt{node} centered, it is recentered to \texttt{zone} centering and if \texttt{expr} is \texttt{zone} centered, it is recentered to \texttt{node} centering. Note that the quotes are required for the second argument. This function is typically used to force a specific centering among the operands of some other expression.

Process Id Function: \texttt{procid()} \hspace{1em} \texttt{[procid(<Var>)]} Return an integer \texttt{scalar} variable which is everywhere the MPI rank associated with each of the blocks of the possibly parallel decomposed mesh upon which \texttt{<Var>} is defined. For serial execution or for parallel execution of a single-block mesh, this will produce a constant zero variable. Otherwise, the values will vary block by block.

Thread Id Function: \texttt{threadid()} \hspace{1em} \texttt{[threadid(expr0)]} Return an integer \texttt{scalar} variable which is everywhere the local thread id associated with each of the blocks of the possibly parallel decomposed mesh upon which \texttt{<Var>} is defined. For non-threaded execution, this will produce a constant zero variable. Otherwise, the values will vary block by block.

isnan Function: \texttt{isnan()} \hspace{1em} \texttt{[isnan(expr0)]} No description available.

q criterion Function: \texttt{q\_criterion()} \hspace{1em} \texttt{[q\_criterion(<gradient(velocity[0])>, <gradient(velocity[1])>, <gradient(velocity[2])>)]} Generates the Q-criterion value developed by Hunt et. al. It is based on the observation that, in regions where the Q-criterion is greater than
zero, rotation exceeds strain and, in conjunction with a pressure min, indicates the presence of a vortex. The three arguments to the function are gradient vectors of the x-, y-, and z-velocity. The gradient function (see \texttt{gradient()}) can be used to create the gradient vectors.

**lambda2 Function:** \texttt{lambda2()} \[
\text{lambda2}\left(\text{gradient(velocity}[0])\right), \text{gradient(velocity}[1]\right), \text{gradient(velocity}[2]\right))
\] Generates the Lambda-2 criterion. It is based on the observation that, in regions where Lambda-2 is less than zero, rotation exceeds strain and, in conjunction with a pressure min, indicates the presence of a vortex. The three arguments to the function are gradient vectors of the x-, y-, and z-velocity. The gradient function (see \texttt{gradient()}) can be used to create the gradient vectors.

**mean curvature Function:** \texttt{mean_curvature()} No description available.

**Gauss Curvature Function:** \texttt{gauss_curvature()} No description available.

**agrad Function:** \texttt{agrad()} No description available.

**key aggregate Function:** \texttt{key_aggregate()} No description available.

**Laplacian Function:** \texttt{laplacian()} No description available.

**rectilinear Laplacian Function:** \texttt{rectilinear_laplacian()} No description available.

**conn components Function:** \texttt{conn_components()} No description available.

**resrad Function:** \texttt{resrad()} No description available.

---

### Time Iteration Expressions

**Average Over Time Function:** \texttt{average_over_time()} \[
\text{average_over_time}\left(<\text{Scalar}, <\text{Start}, <\text{Stop}, <\text{Stride}>\right)\]
Return a new \textit{scalar} variable in which each zonal or nodal value is the average over the times indicated by \texttt{Start}, \texttt{Stop} and \texttt{Stride}.

**Min Over Time Function:** \texttt{min_over_time()} \[
\text{min_over_time}\left(<\text{Scalar}, <\text{Start}, <\text{Stop}, <\text{Stride}>\right)\]
Return a new \textit{scalar} variable in which each zonal or nodal value is the minimum value the variable, \texttt{<Scalar>}, attained over the times indicated by \texttt{Start}, \texttt{Stop} and \texttt{Stride}.

**Max Over Time Function:** \texttt{max_over_time()} \[
\text{max_over_time}\left(<\text{Scalar}, <\text{Start}, <\text{Stop}, <\text{Stride}>\right)\]
Return a new \textit{scalar} variable in which each zonal or nodal value is the maximum value the variable, \texttt{<Scalar>}, attains over the times indicated by \texttt{Start}, \texttt{Stop} and \texttt{Stride}.

**Sum Over Time Function:** \texttt{sum_over_time()} \[
\text{sum_over_time}\left(<\text{Scalar}, <\text{Start}, <\text{Stop}, <\text{Stride}>\right)\]
Return a new \textit{scalar} variable in which each zonal or nodal value is the sum of the values the variable, \texttt{<Scalar>}, attains over the times indicated by \texttt{Start}, \texttt{Stop} and \texttt{Stride}.

**First Time When Condition Is True Function:** \texttt{first_time_when_condition_is_true()} \[
\text{first_time_when_condition_is_true}\left(<\text{Cond}, <\text{Fill}, <\text{Start}, <\text{Stop}, <\text{Stride}>\right)\]
Return a new \textit{scalar} variable in which each zonal or nodal value is the first time (not cycle and not time-index, but floating point time) at which the true/false condition, \texttt{<Cond>} is true. The \texttt{<Fill>} value is used if there is no first time the condition is true.

**Last Time When Condition Is True Function:** \texttt{last_time_when_condition_is_true()} \[
\text{last_time_when_condition_is_true}\left(<\text{Cond}, <\text{Fill}, <\text{Start}, <\text{Stop}, <\text{Stride}>\right)\]
Return a new \textit{scalar} variable in which each zonal or nodal value is the last time (not cycle and not time-index,
but floating point time) at which the true/false condition, \(<\text{Cond}>\) is true. The \(<\text{Fill}>\) value is used if there is no last time the condition is true.

**First Cycle When Condition Is True Function:** \texttt{first\_cycle\_when\_condition\_is\_true()}

\[
\texttt{first\_cycle\_when\_condition\_is\_true(<\text{Cond}>,<\text{Fill}>,<\text{Start}>,<\text{Stop}>,<\text{Stride}>)}
\]

Return a new integer valued \textit{scalar} variable in which each zonal or nodal value is the first cycle (not time and not time-index, but integer cycle) at which the true/false condition, \(<\text{Cond}>\) is true. The \(<\text{Fill}>\) value is used if there is no first cycle the condition is true.

**Last Cycle When Condition Is True Function:** \texttt{last\_cycle\_when\_condition\_is\_true()}

\[
\texttt{last\_cycle\_when\_condition\_is\_true(<\text{Cond}>,<\text{Fill}>,<\text{Start}>,<\text{Stop}>,<\text{Stride}>)}
\]

Return a new integer valued \textit{scalar} variable in which each zonal or nodal value is the last cycle (not time and not time-index, but integer cycle) at which the true/false condition, \(<\text{Cond}>\) is true. The \(<\text{Fill}>\) value is used if there is no last cycle the condition is true.

**First Time Index When Condition Is True Function:** \texttt{first\_time\_index\_when\_condition\_is\_true()}

\[
\texttt{first\_time\_index\_when\_condition\_is\_true(<\text{Cond}>,<\text{Fill}>,<\text{Start}>,<\text{Stop}>,<\text{Stride}>)}
\]

Return a new integer valued \textit{scalar} variable in which each zonal or nodal value is the first time index (not cycle and not time, but integer time-index) at which the true/false condition, \(<\text{Cond}>\) is true. The \(<\text{Fill}>\) value is used if there is no first time-index the condition is true.

**Last Time Index When Condition Is True Function:** \texttt{last\_time\_index\_when\_condition\_is\_true()}

\[
\texttt{last\_time\_index\_when\_condition\_is\_true(<\text{Cond}>,<\text{Fill}>,<\text{Start}>,<\text{Stop}>,<\text{Stride}>)}
\]

Return a new integer valued \textit{scalar} variable in which each zonal or nodal value is the last time index (not cycle and not time, but integer time-index) at which the true/false condition, \(<\text{Cond}>\) is true. The \(<\text{Fill}>\) value is used if there is no last time-index the condition is true.

**var when condition is first true Function:** \texttt{var\_when\_condition\_is\_first\_true()}

\[
\texttt{var\_when\_condition\_is\_first\_true(expr0)}
\]

No description available.

**var when condition is last true Function:** \texttt{var\_when\_condition\_is\_last\_true()}

\[
\texttt{var\_when\_condition\_is\_last\_true(expr0)}
\]

No description available.

**time at minimum Function:** \texttt{time\_at\_minimum()}

\[
\texttt{time\_at\_minimum(expr0)}
\]

No description available.

**cycle at minimum Function:** \texttt{cycle\_at\_minimum()}

\[
\texttt{cycle\_at\_minimum(expr0)}
\]

No description available.

**time index at minimum Function:** \texttt{time\_index\_at\_minimum()}

\[
\texttt{time\_index\_at\_minimum(expr0)}
\]

No description available.

**value at minimum Function:** \texttt{value\_at\_minimum()}

\[
\texttt{value\_at\_minimum(expr0)}
\]

No description available.

**time at maximum Function:** \texttt{time\_at\_maximum()}

\[
\texttt{time\_at\_maximum(expr0)}
\]

No description available.

**cycle at maximum Function:** \texttt{cycle\_at\_maximum()}

\[
\texttt{cycle\_at\_maximum(expr0)}
\]

No description available.

**time index at maximum Function:** \texttt{time\_index\_at\_maximum()}

\[
\texttt{time\_index\_at\_maximum(expr0)}
\]

No description available.

**value at maximum Function:** \texttt{value\_at\_maximum()}

\[
\texttt{value\_at\_maximum(expr0)}
\]

No description available.

**localized compactness Function:** \texttt{localized\_compactness()}

\[
\texttt{localized\_compactness(expr0)}
\]

No description available.

**merge tree Function:** \texttt{merge\_tree()}

\[
\texttt{merge\_tree(expr0)}
\]

No description available.

**split tree Function:** \texttt{split\_tree()}

\[
\texttt{split\_tree(expr0)}
\]

No description available.

**local threshold Function:** \texttt{local\_threshold()}

\[
\texttt{local\_threshold(expr0)}
\]

No description available.
Expression Compatibility Gotchas

*VisIt* will allow you to define expressions that it winds up determining to be invalid later when it attempts to execute those expressions. Some common issues are the mixing of incompatible mesh variables in the same expression. *VisIt* will allow users to define such an expression. But, when it is plotted, the plot will fail.

As an aside, as the user goes back and forth between the Expressions window creating and/or adjusting expression definitions, *VisIt* makes no attempt to keep track of all the changes made in expressions and automatically update plots as expressions change. Users have to manually clear or delete plots to force *VisIt* to re-draw plots in which the expressions changed.

If what is really intended was a scalar mesh variable, then users must use one of the expression functions that converts a vector to a scalar such as the magnitude() built-in expression or the array de-reference operator.

Centering Compatibility

Some variables are zone centered and some are node centered. What happens if a user combines these in an expression? *VisIt* will default to zone centering for the result. If this is not the desired result, the *recenter()* expression function should be used, where appropriate, to adjust centering of some of the terms in the expression. Note that ordering of operations will probably be important. For example
both achieve a node-centered result. But, each expression is subtly (and numerically) different. The first recenter’s zone_var to the nodes and then performs the summation operator at each node. In the second, there is an implied recentering of node_var to the zones first. Then, the summation operator is applied at each zone center and finally the results are recentered back to the nodes. In all likelihood this creates in a numerically lower quality result. The moral is that in a complex series of expressions be sure to take care where you want recentering to occur.

Mesh Compatibility

In many cases, especially in Silo databases, all the available variables in a database are not always defined on the same mesh. This can complicate matters involving expressions in variables from different meshes.

Just as in the previous two examples of incompatible variables where the solution was to apply some function to make the variables compatible, we have to do the same thing when variables from different meshes are combined in an expression. The key expression functions which enable this are called Cross Mesh Field Evaluation or CMFE expression functions. We will only briefly touch on these here. CMFEs will be discussed in much greater detail elsewhere.

Just as for centering, we have two options when dealing with variables from two different meshes. Each of which involves mapping one of the variables onto the other variable’s mesh using one of the CMFE expression functions.

Automatic expressions

1.8.2 Query

VisIt allows you to gather quantitative information from the database being visualized through the use of queries. A query is a type of calculation that can either return values from the database or values that are calculated from data in the database. For example, VisIt’s Pick and Lineout capabilities (described later in this chapter) are specialized point and line queries that print out the values of variables in the database at points or along lines. In addition to point and line queries, VisIt provides database queries that return values that are based on all of the data in a database.

Some queries can even be executed for all of the time states in a database to yield a Curve plot of the query’s behavior over time. This feature will be covered in more detail a little later.

VisIt’s queries are available in the Query Window (shown in Figure 1.201), which you can open by clicking the Query option in the Main Window’s Control menu. The Query Window consists of upper and lower areas where the upper area allows you to select a query and set its query parameters. The controls for setting a query’s parameters change as required and some queries have no parameters and thus have no controls for setting parameters. The bottom area of the window displays the results of the query once VisIt has finished processing it. The results for new queries are appended to the output from previous queries until you clear the Query results by clicking the Clear results button.

Query types

VisIt’s queries can be divided into three types: database queries, point queries, and line queries. Database queries usually calculate information for the database as a whole instead of concentrating on a single zone or node but some Pick-related database queries do concentrate on cells and nodes. Point queries calculate information for a point in the database and several types of variable picking queries fall into this category. Line queries calculate information along a line. Each type of query has different controls in the Query parameters area (see Figure 1.202) and as you highlight different queries, the controls in the Query parameters area may change.
Fig. 1.201: Query window
Database queries provide a few different interfaces depending on the query. Many database queries require no additional input so they have no controls except for the Query button. Other database queries ask whether the query is to be performed with respect to the original data or the actual data, which is that data that is left in the plot after subsets have been removed and operators have transformed the data. Finally, some database queries ask for a specific domain number and zone or node number.

Point queries provide interfaces in the Query parameters area that allow you to enter a 3D point or a screen space point to use as the point for the query. Line queries provide an interface that lets you specify the start and end positions of the line as well as the number of sample points to consider along the length of the line. Nearly all query types allow you to provide additional variables to query in a Variables text field.

**Built-in queries**

**Database Queries**

**2D area** The 2D area query calculates the area of the 2D plot highlighted in the Plot list and prints the result to the Query results. VisIt can produce a Curve plot of this query with respect to time.
3D surface area  The 3D surface area calculates the area of the plot highlighted in the **Plot list** and prints the result to the **Query results**. VisIt can produce a Curve plot of this query with respect to time.

Area Between Curves  The Area Between Curves query calculates the area between 2 curve plots. The plots that will serve as input to this query must both be highlighted in the **Plot list** or VisIt will issue an error message. Once the area has been calculated, the result is printed to the **Query results**.

Centroid  This query will calculate the centroid of a dataset. The contribution of each cell is calculated assuming its mass all lies at the center of the cell. If the query is performed on a Pseudocolor plot, the plot’s variable will be assumed to be density. If the query is performed on a plot such as a Mesh plot or FilledBoundary plot, uniform density will be used. The results are print to the **Query results**.

Chord Length Distribution  The Chord Length Distribution query calculates a probability density function of chord length over a two or three dimensional object. Axially symmetric objects (RZ-meshes) are treated as 3D meshes and chords are calculated over the revolved, 3D object. A statistical approach, casting uniform density, random lines, is used. The result of this query is a curve, which is outputted as a separate file. This curve is a probability density function over length scale. The name of the resulting file is printed to the **Query results**.

Compactness  The Compactness query calculates mesh metrics and prints them in the **Query results**.

Cycle  The Cycle query prints the cycle for the plot that is highlighted in the **Plot list** to the **Query results**.

Distance from Boundary  The Distance From Boundary query calculates how much mass is at a given distance away from the boundary of a shape. An important distinction for this query is that distance from the boundary (for a given point) is not defined as the shortest distance to the boundary, but simultaneously as all surrounding distances. Axially symmetric objects (RZ-meshes) are treated as 3D meshes and length scales are calculated over the revolved, 3D object. The implementation employs a statistical approach, with the casting of uniform density, random lines. The result of this query is a curve, which is outputted as a separate file. This curve contains the amount of mass as a function of length scale. Integrating the curve between P0 and P1 will give the total mass at distance between P0 and P1 (given the interpretation above). The name of the resulting file is printed to the **Query results**.

Eulerian  The Eulerian query calculates the Eulerian number for the mesh that is used by the highlighted plot in the Plot list. The results are printed to the **Query results**.

Expected Value  The Expected Value query calculates the integral of \( xf(x)dx \) for some curve \( f(x) \). The curve should be highlighted in the **Plot list** and prints the result to the **Query results**. This query is intended for distribution functions.

Grid Information  The Grid Information query prints information for each domain in a multi-domain mesh. The mesh type is printed as well as the mesh sizes. For structured meshes the size information contains the logical mesh dimensions (IJK sizes) and for unstructured meshes the size information contains the number of nodes and number of cells in the mesh. The query can optionally accept a **get_extents** parameter that will cause the spatial extents for each domain to be obtained. The query also accepts an optional **get_ghosttype** parameter that causes the ghost zone information for each domain to be obtained. Both the numerical value and list of or’d values for ghost values are obtained. All query outputs are printed to the **Query results**.

Integrate  The Integrate query calculates the area under the Curve plot that is highlighted in the Plot list and prints the result to the **Query results**.

Kurtosis  The Kurtosis query calculates the kurtosis of a normalized distribution function. The normalized distribution function must be represented as a Curve plot in VisIt. Kurtosis measures the variability of a distribution by comparing the ratios of the fourth and second central moments. The results are print to the **Query results**.

L2Norm  The L2Norm query calculates the L2Norm, or square of the integrated area, of a Curve plot. The Curve plot must be highlighted in the **Plot list**. The results are printed to the **Query results**.

L2Norm Between Curves  The L2Norm query takes two Curve plots as input and calculates the L2Norm between the 2 curves. Both Curve plots must be highlighted in the **Plot list** or VisIt will issue an error message. The results are printed to the **Query results**.
Min  The Min query calculates the minimum value for the variable used by the highlighted plot in the Plot list and prints the value and the logical and physical coordinates where the minimum value was found to the Query results.

Mass Distribution  The Mass Distribution query calculates how much mass occurs at different length scales over a two or three dimensional object. Axially symmetric objects (RZ-meshes) are treated as 3D meshes and length scales are calculated over the revolved, 3D object. The implementation employs a statistical approach, with the casting of uniform density, random lines. The result of this query is a curve, which is outputted as a separate file. This curve contains the amount of mass as a function of length scale. Integrating the curve between P0 and P1 will give the total mass between length scale P0 and length scale P1. The name of the resulting file is printed to the Query results.

Max  The Max query calculates the maximum value for the variable used by the highlighted plot in the Plot list and prints the value and the logical and physical coordinates where the maximum value was found to the Query results.

MinMax  The MinMax query calculates the minimum and maximum values for the variable used by the highlighted plot in the Plot list and prints the values and their logical and physical coordinates in the Query results.

Moment of inertia  This query will calculate the moment of inertia tensor for each cell in a three-dimensional dataset. The contribution of each cell is calculated assuming its mass all lies at the center of the cell. If the query is performed on a Pseudocolor plot, the plot’s variable will be assumed to be density. If the query is performed on a plot such as a mesh plot or FilledBoundary plot, uniform density will be used. The results are printed to the Query results.

NodeCoords  The NodeCoords query prints the node coordinates for the specified node and prints the values in the Query results.

NumNodes  The NumNodes query prints the number of nodes for the mesh used by the highlighted plot in the Plot list to the Query results.

NumZones  The NumZones query prints the number of zones for the mesh used by the highlighted plot in the Plot list to the Query results.

Revolved surface area  The Revolved surface area query revolves the mesh used by the highlighted plot in the Plot list about the X-axis and prints the plot’s revolved surface area to the Query results.

Revolved volume  The Revolved volume area query revolves the mesh used by the highlighted plot in the Plot list about the X-axis and print’s the plot’s volume to the Query results.

Skewness  The Skewness query calculates the skewness of a normalized distribution function. The normalized distribution function must be represented as a Curve plot in VisIt. Skewness measures the symmetry of a distribution using its second and third central moments. The results are print to the Query results.

Spatial Extents  The Spatial Extents query calculates the original or actual spatial extents for the plot that is highlighted in the Plot list. Whether the original or actual extents are calculated is determined by setting the options in the Query parameters area. The spatial extents are printed to the Query results when the query has finished.

Spherical compactness factor  This query attempts to measure how spherical a three dimensional shape is. The query first determines what the volume of a shape is. It then constructs a sphere that has that same volume. Finally, the query positions the sphere so that the maximum amount of the original shape is within the sphere. The query returns the percentage of the original shape that is contained within the sphere. The results are print to the Query results. VisIt can produce a Curve plot of this query with respect to time.

Time  The Time query prints the time for the plot that is highlighted in the Plot list to the Query results.

Variable Sum  The Variable Sum query adds up the variable values for all cells using the plot highlighted in the Plot list and prints the results to the Query results. VisIt can produce a Curve plot of this query with respect to time.

Volume  The Volume query calculates the volume of the mesh used by the plot highlighted in the Plot list and prints the value to the Query results. VisIt can use this query to produce a Curve plot of volume with respect to time.
Watertight  The Watertight query determines if a three-dimensional surface mesh, of the plot highlighted in the Plot list, is “watertight”, meaning that it is a closed volume with mesh connectivity such that every edge is incident to exactly two faces. This means that no edge can have a duplicate in the exact same position. The result of the query is printed in the Query results.

Weighted Variable Sum  The Weighted Variable Sum query adds up the variable values, weighted by cell size, for all cells using the plot highlighted in the Plot list and prints the results to the Query results. VisIt can produce a Curve plot of this query with respect to time.

ZoneCenter  The ZoneCenter query calculates the zone center for a certain cell in the database used by the highlighted plot in the Plot list. The cell center is printed to the Query results and the Pick Window.

Point Queries

NodePick  The NodePick query performs node picking at the specified world coordinate which, if used in 3D, need not be on the surface of a 3D dataset. The plot to be picked must be highlighted in the Plot list. Information about the picked node, if there is one, is printed to the Query results and the Pick Window.

Pick  The Pick query performs zone picking at the specified world coordinate which, if used in 3D, need not be on the surface of a 3D dataset. The plot to be picked must be highlighted in the Plot list. Information about the picked node, if there is one, is printed to the Query results and the Pick Window.

PickByNode  The PickByNode query performs node pick using the highlighted plot in the Plot list and specified domain and node values. You can give a global node number if you turn on the Use Global Node check box. A pick point is added to the vis window and the query results appear in the Query results and the Pick Window. Note: this is the query to use if you want to query the database for the value of a variable at a certain node. VisIt can produce a Curve plot of this query with respect to time.

PickByZone  The PickByZone query performs zone pick using the highlighted plot in the Plot list and specified domain and zone values. You can give a global node number if you turn on the Use Global Zone check box. A pick point is added to the vis window and the query results appear in the Query results and the Pick Window. Note: this is the query to use if you want to query the database for the value of a variable at a certain cell. VisIt can produce a Curve plot of this query with respect to time.

Line Queries

Lineout  The Lineout query creates a new instance of the highlighted plot in the Plot list, applies a Lineout operator, and copies the plot to another vis window. The properties of the Lineout operator such as the start and end points are set using the controls in the Query parameters area of the Query Window. Creating Lineouts in this manner instead of using VisIt’s interactive lineout allows you to create 1D Curve plots from 3D databases.

Executing a query

VisIt has many queries from which to choose. You can choose the type of query to execute by clicking on the name of the query in the Queries list. The Queries list usually displays the names of all of the queries that VisIt knows how to execute. If you instead want to view a subset of the queries, grouped by function, you can make a selection from the Display as combo box. Once you have clicked on a query in the Query list, the Query parameters area updates to show the controls that you need to edit the parameters for the query. In the case of a point query like Pick, the only parameters you need to specify are the 3D point where VisIt will extract values and the names of the variables that you want to examine. Once you specify the query parameters, click the Query button to tell VisIt to process the query. Once VisIt has fulfilled your request, the query results are displayed in the Query results at the bottom of the Query Window.
Querying over time

Many of VisIt’s queries can be executed for every time state in the database used by the queried plot. The results from a query over time is a Curve plot that plots the query results with respect to time. The Query parameters area contains a Time Curve button when the selected query can be plotted over time. Clicking the Time Curve button executes the selected query for each time state in the database used by the plot highlighted in the Plot list. VisIt then creates a new Curve plot in a new vis window and uses the query results versus time as the curve data.

![Query Over Time Window](image)

Fig. 1.203: Query Over Time Window

By default, querying over time will force VisIt to execute the selected query on every time state in the relevant database. If you want to restrict the number of time states used when querying over time or if you want to set some general options that also affect how time curves are created, you can set additional options in the Query Over Time Window (see Figure 1.203). If you want to open the Query Over Time Window, click on the Query over time option in the Controls menu in VisIt’s Main Window.

Querying over a time range

You can restrict the range of time states that are considered when VisIt is performing a query over time if you specify a start or end time state in the Query Over Time Window. To set a starting time state, click the Starting timestep check box and enter a new time state into the adjacent text field. To set an ending time state, click the Ending timestep check box and enter a new ending time state into the adjacent text field.
In addition to setting the starting and ending time states, you can also specify a stride so VisIt can skip frames in the middle and consider every Nth frame instead of every frame. If you want to specify a stride, enter a new stride into the \textit{Stride} text field in the \textbf{Query Over Time Window} and click the \textbf{Apply} button.

**Setting the axis title**

When VisIt creates a new Curve plot, after having calculated a query over time, the horizontal axis label is labeled with the database cycles. If you prefer to think about time in terms of time state or simulation time then you can change the axis label by clicking one of the following radio buttons in the \textbf{Query Over Time Window}: \textit{Cycle}, \textit{Time}, \textit{Timestep}.

**Setting the time curve’s destination window**

When VisIt creates a Curve plot using the results of a query over time, the Curve plot is placed in a vis window designated for Curve plots. If there is no vis window into which the Curve plot can be added, VisIt creates a new vis window to contain the Curve plot. If you want VisIt to always place the new Curve plot in a specific window, turn off the \textit{Use 1st unused window or create new one} check box and enter a new window number into the \textit{Window\#} text field. After setting these options, subsequent Curve plots created by querying over time will be added to the specified vis window.

1.8.3 \textbf{Pick}

VisIt provides a way to interactively pick values from the visualized data using the visualization window’s Zone Pick and Node Pick modes. When a visualization window is in one of those pick modes, each mouse click in the visualization window causes VisIt to find the location and values of selected variables at the pick point. When VisIt is in Zone pick mode, it finds the variable values for the zones that you click on. When VisIt is in node pick mode, similar information is returned but instead of returning information about the zone that you clicked on, VisIt returns information about the node closest to the point that you clicked. Pick is an essential tool for performing data analysis because it can extract exact information from the database about a point in the visualization.

**Pick mode**

You can put the visualization window into one of VisIt’s pick modes by selecting \textit{Zone Pick} or \textit{Node Pick} from the \textbf{Popup menu’s Mode} submenu. After the visualization window is in pick mode, each mouse click in the visualization window causes VisIt to determine the values of selected variables for the zone that contains the picked point or the node closest to the picked point. Each picked point is marked with an alphabetic label which starts at A, cycles through the alphabet and repeats. The pick marker is added to the visualization window to indicate where pick points have been added in the past. To clear pick points from the visualization window, select the \textit{Pick points} option from the \textbf{Clear} menu in the \textbf{Main Window’s Window} menu. The dimension of the plots in the visualization does not matter when using pick mode. Both 2D and 3D plots can be picked for values. However, when using pick mode with 3D plots, only the surface of the plots can be picked for values. If you want to obtain interior values then you should use one of the Pick queries or apply operators that expose the interiors of 3D plots before using pick. An example of the visualization window with pick points is shown in Figure 1.204 and an example of node pick and zone pick markers is shown in Figure 1.205.

**Pick Window**

Each time a new pick point is added to the visualization window by clicking on a plot, VisIt extracts information about the pick point from the plot’s database and displays it in the \textbf{Pick Window} (Figure 1.206) and the \textbf{Output Window}. If the \textbf{Pick Window} does not automatically open after picking, you can open the \textbf{Pick Window} by selecting the \textbf{Pick} option from the \textbf{Main Window’s Controls} menu.
Fig. 1.204: Visualization window with pick points
Fig. 1.205: Zone pick marker L and node pick markers M, N, O, P
Fig. 1.206: Pick Window

1.8. Quantitative Analysis
The **Pick Window** mainly consists of a group of tabs, each of which displays the values from a pick point. The tab label A, B, C, etc. corresponds to the pick point label in the visualization window. Since there is a fixed number of tabs in the **Pick Window**, tabs are recycled as the number of pick points increases. When a pick point is added, the next available tab, which is usually the tab to the right of the last unused tab, is populated with the pick information. If the rightmost tab already contains pick information, the leftmost tab is recycled and the process repeats. To see a complete list of picked points, open the **Output Window**.

The information displayed in each tab consists of the database name and timestep, the coordinates of the pick point, the zone/cell that contains the pick point, the nodes that make up the cell containing the pick point, and the picked variables. The rest of the **Pick Window** is devoted to setting options that format the pick output.

### Setting the pick variable

The **Pick Window** contains a **Variables** text field that allows you to specify pick variables. Most of the time, the value in the text field is the word “default” which tells VisIt to use the plotted variables as the pick variables. You can replace the default pick variable by typing one or more valid variable names, separated by spaces, into the **Variables** text field. You can also select additional pick variables by selecting a new variable name from the **Variables** variable button to the left of the **Variables** text field. When more than one variable is picked, multiple variables appear in the pick information displayed in the information tabs.

### Concise pick output

Pick returns a lot of information when you pick on a plot. The **Pick Window** usually displays the pick output one item per line, which can end up taking a lot of vertical space. If you want to condense the information into a smaller area, click the **Concise output** check box. Sometimes, not all of the information is relevant for your analysis so VisIt provides options to hide certain items in the pick output. If you don’t want VisIt to display the name of the picked mesh, turn off the **Show Mesh Name** check box. If you don’t want VisIt to show the time state, turn off the **Show timestep** check box.

### Turning off incident nodes and cells in pick output

When VisIt performs a pick, the default behavior is to show a lot of information about the cell or node that was picked. This information usually includes the nodes or cells that were incident to the node or cell that was picked. The incident nodes and cells are included to give more information about the neighborhood occupied by the cell or node. If you want to turn off incident nodes and cells in the pick output, click off the **Display incident nodes/zones** check box.

### Displaying global node and cell numbers

Many large meshes are decomposed into smaller meshes called domains that, when added together, make up the whole mesh. Each domain typically has its own range of cell numbers that begin at 0 or 1, depending on the mesh’s cell origin. Any global cell numbering scheme that may have been in place before the original mesh was decomposed into domains is often lost. However, some meshes have auxiliary information that allows VisIt to use the original global node and cell numbers for the domains. If you want the pick output to contain global node and cell numbers if they are available, click on the **Display global nodes/zones** check box.

### Turning off pick markers for new pick points

Some queries that perform picks create pick markers by default, as do VisIt’s regular pick modes. If you want to prevent pick queries from creating pick markers, click off the **Pick Window’s Display reference pick letter** check box.
Returning node information

In addition to printing the values of the pick variables, pick can also display information about the nodes or cells over which the pick variables are defined. By default, VisIt only returns the integer node indices of the nodes contained by the picked cell. You can make VisIt return the node coordinates in other formats by checking the **Id** check box in the **Display for Nodes** area. The node coordinates can be displayed 4 different ways: Node indices, physical coordinates, domain-logical coordinates, or block-logical coordinates. Click the check boxes in the **Display for Nodes** area that correspond to the types of node information that you want to examine.

Returning zone information

The **Pick Window** has controls in its **Display for Zones** area that allow you to specify how you want VisIt to display zone information. Click the check boxes that correspond to the types of information that you want to examine.

Automatically showing the Pick Window

When you pick on a plot, VisIt automatically opens the **Pick Window** to display the results of the pick operation. You can prevent VisIt from automatically showing the **Pick Window** after a pick operation by turning off the **Automatically show window** check box in the **Pick Window**. If the **Pick Window** does not automatically appear after picking then you can turn on the **Automatically show window** check box.

Picking over time

Querying over time is normally done using the controls in the **Query Window** but you can also pick over time to generate curves that show the behavior of a picked zone or node over time. To pick over time, you must click the **Create time curve with next pick** check box in the **Pick Window**. Once that check box is turned on, each pick operation will result in a new Curve plot that shows the behavior of the most recently picked zone or node over time.

1.8.4 Lineout

One-dimensional curves, created using data from 2D or 3D plots, are popular for analyzing data because they are simple to compare. VisIt's visualization windows can be put into a mode that allows you to draw lines, along which data are extracted, in the visualization window. The extracted data are turned into a Curve plot in another visualization window. If no other visualization window exists, VisIt creates one and adds the Curve plot to it. Curve plots are often more useful than 2D Pseudocolor plots because they allow the data along a line to be seen spatially as a 1D curve instead of relying on differences in color to convey information. Furthermore, the curve data can be exported to curve files that allow the data to be imported into other Lawrence Livermore National Laboratory curve analysis software such as *Ultra*.

Lineout mode

You can put the visualization window into lineout mode by selecting the **Lineout** icon (Figure 1.207) in the visualization window’s Toolbar or from the **Popup menu’s Mode** submenu. Note that lineout mode is only available with 2D plots in this version though you can create 3D lineouts using the Lineout query in the **Query Window**. After the visualization window is in lineout mode, you can draw reference lines in the window. Each reference line causes VisIt to extract data from the database along the prescribed path and draw the data as a Curve plot in another visualization window. Each reference line is drawn in a color that matches the initial color of the Curve plot so the reference lines, which may not have labels, can be easily associated with their corresponding Curve plots. To clear the reference lines...
from the visualization window, select the **Clear reference lines** option from **Popup menu’s Clear** submenu. An example of the visualization window with reference lines and Curve plots is shown in **Figure 1.208**.

![Figure 1.207: Lineout mode toolbar icon](image)

**Fig. 1.207: Lineout mode toolbar icon**

![Figure 1.208: Visualization windows with reference line and Curve plots](image)

**Fig. 1.208: Visualization windows with reference line and Curve plots**

**Curve plot**

Curve plots are created by drawing reference lines. The visualization window must be in lineout mode before reference lines can be created. You can create a reference line by positioning the mouse over the first point of interest, clicking the left mouse button and then moving the mouse, while pressing the left mouse button, and releasing the mouse over the second endpoint. Releasing the mouse button creates a reference line along the path that was drawn with the mouse. When you draw a reference line, you cause a Curve plot of the data along the reference line to appear in another visualization window. If another visualization window is not available, VisIt opens a new one before creating the Curve plot. The Curve plot in the second window can be modified by setting the active window to the visualization window that contains the Curve plots.

See **Curve Plot** for information on changing the Curve plot’s appearance.

**Saving curves**

Once a curve has been generated, it can be saved to a curve file. A curve file is an ASCII text file that contains the X-Y pairs that make up the curve and it is useful for exporting curve data to other curve analysis programs. To save a curve, make sure you first set the active window to the visualization window that contains the curve. Next, save the window using the **curve** file format. All of the curves in the visualization window are saved to the specified curve file.
Lineout Operator

The Curve plot uses the Lineout operator to extract data from a database along a linear path. The Lineout operator is not generally available since curves are created only through reference lines and not the Plot menu. Still, once a curve has been created using the Lineout operator, certain attributes of the Lineout can be modified. Note that when you modify the Lineout attributes, it is best to turn off the Apply operators to all plots check box in the Main Window so that all curves do not get the same set of Lineout operator attributes.

Setting lineout endpoints

You can modify the line endpoints by typing new coordinates into the Point 1 or Point 2 text fields of the Lineout attributes window (Figure 1.209). Each endpoint is a 3D coordinate that is specified by three space-separated floating point numbers. If you are performing a Lineout operation on 2D data, you can set the value for the Z coordinate to zero.

![Fig. 1.209: Lineout attributes window](image)

Setting the number of lineout samples

The Lineout operator works by extracting sample points along a line. The sample points are then used to create Curve plots. The Lineout operator’s default sampling scheme is to sample data values at the intersections between the sampling line and the cell boundaries encountered along the way. This method gives rise to jagged Curve plots favored by many VisIt users. See Figure 1.210 for a comparison between the sampling methods. If you instead want to smoothly sample the cells along the sampling line with some number of evenly spaced sample points, you can make the Lineout operator use evenly spaced sampling by clicking on the Override Global Lineout Settings check box in the Lineout attributes window. Then click on the Use Sampling check box and enter a number of sample points. The number of sample points taken along the line determine the fidelity of the Curve plot. Generally, it is best to set the number of sample points such that each cell is sampled at least once. To set the number of sample points, type a new number into the Samples text field.
Interactive mode

When the Interactive check box is checked, changes to the Lineout operator can be made by using the Line tool available from the originating plot’s visualization window Toolbar or Popup menu. Interactive mode does not apply to lineouts created via the Curve plot’s variable menu.

To utilize the line tool to modify a Lineout curve, make the visualization window with the originating plot the active window. Choose the Line tool. It should be initialized with the endpoints of the reference line. Moving the tool will change the lineout. (Note: Due to a current bug, the tool must be activated, deactivated, then activated a second time in order to be properly initialized with the Lineout’s endpoint values.) See Interactive Tools for more information on tool utilization.

Reference line labels

You can make the reference lines in the window that caused Curve plots to be generated to have labels by checking the Lineout operator’s Refline Labels check box.

Lineout query

Performing a Lineout query requires an existing non-hidden plot in the active window. Choose Lineout from the Query window (available from the GUI’s Controls dropdown menu). Set start and end points (similar to Setting lineout endpoints). Lineout query is the only Lineout method that allows you to create curves for multiple variables. Simply select the desired variables from the Variables dropdown menu. Default means the variable as plotted in the currently active plot. A lineout curve will be generated for each variable, plotted along the same reference line. Each curve will have its own color. The Use Sampling and Sample Points option is the same as before.

Lineout via Curve plot variable menu

With this method, Lineout is considered one of the Operators that Generate New Variables. That means you can use it without first generating a plot of the data from which you wish to extract the lineout. To create a Lineout in this manner, open your database, select Curve plot, then choose operators/Lineout/<var-name> from the Curve plot’s variable menu as shown in Figure 1.212.

It is highly recommended that you modify the Lineout’s endpoints before clicking draw, as the defaults will probably not be appropriate for your data.
Fig. 1.211: Lineout query’s parameters window

Fig. 1.212: Choosing lineout from the Curve plot’s variable menu
Global lineout options

The Lineout Options Window, available by selecting Lineout from the Controls menu in the Main Window contains global lineout options. They are global in the sense that they will apply to all future lineouts. The Lineout Options Window has controls for choosing the destination window of the lineout curve plots, as well as settings for how changes to the originating plot affect the lineout curve plot. Modifying these options will only apply to future lineouts, not lineouts already created.

Lineout destination window

By default, VisIt will place all lineout curves in the same window. It will use the first unused open window or create one if one does not yet exist. You can override this behavior for future lineouts by unchecking the Use 1st unused window checkbox, and typing a window number into the Window # text box.

Freeze In Time

If the plot that originated the Lineout curve was from a time-varying database, the curve can be advanced in time using the animation controls for the window containing the lineout curve. If you would rather the lineout be frozen at the timestep from which it was taken, check the Freeze in Time option. This will also disable the ability to synchronize the lineout curve with its originating plot.
Synchronous lineout

Normally when you perform a lineout operation, the Curve plot that results from the lineout operation is in no way connected to the plots in the window that originated the Curve plot. If you want variable or time state changes made to the originating plots to also affect the Curve plots that were created via lineout, click the Synchronize with originating plot check box in the Lineout Options Window (see Figure 1.213).

With this option selected, any change to the variable in the plot that originated the lineout, will update the lineout to reflect the new variable’s data. When you change time states for the plot that originated the lineout, the lineout will update to reflect the data at the new time state.

To make VisIt create a new Curve plot for the lineout instead of updating when you change time states in the originating plot, change the Time change behavior in the Lineout Options Window from updates curve to creates new curve. VisIt will then put a new curve in the lineout destination window each time you advance to a new time state, resulting in many Curve plots (see Figure 1.214). By default, VisIt will make all of the related Curve plots be the same color. You can override this behavior by selecting creates new color instead of repeats color from the New curve combo box.

Synchronization does not apply to lineout curves created via the Curve plot variable menu, as this type of lineout does not have an originating plot.

Sampling and Refline labels

These options are the same as described for individual lineouts. Use these options when you want your choices to apply to all lineouts.

1.8.5 Data-Level Comparisons Wizard

The data-level comparisons wizard facilitates creation of expressions that can be used when comparing fields on different meshes and/or in different databases. Such expressions are also known as Cross-Mesh Field Evaluation.
(CMFE) expressions because they effectively take a field defined on one mesh and evaluate it (e.g. map it) onto a new mesh. The data-level comparisons wizard is a very helpful alternative to entering CMFE expressions directly into the expression system manually.

These expressions involve the concepts of a donor variable and a target mesh. The donor variable is the variable to be mapped onto a new mesh. The target mesh is the mesh onto which the donor variable is to be mapped. In addition, the term donor mesh refers to the mesh upon which the donor variable is defined. Also, the target mesh is always interpreted as a mesh in the currently active database. Data-level comparison expressions (CMFEs) are always mapping data from other meshes, possibly in other databases onto a target mesh which is understood to be in the currently active database.

To start the wizard, go to Controls->Data-Level Comparisons... as shown in Figure 1.215.

This will open the the initial window where the user is asked to choose between a few basic varieties of CMFE expressions. These differ in the relative locations (e.g. which database) of the donor variable and target mesh.
1. Donor variable and target mesh are in the same database.
2. Donor variable and target mesh are from different time states of the same database.
3. Donor variable and target mesh are in wholly different databases.

**Note:** if you wish to create a CMFE that works properly across a time series with wholly different databases (3rd case above), the data-level comparisons wizard does not directly support that. However, you can use wizard to construct an initial CMFE expression and then edit it manually in the Expression Window to adjust it for a time series following the documentation on donor variable syntax.

![Data-Level Comparison Setup](image)

Fig. 1.216: Selecting among varieties of CMFE expressions

If the user is unsure, selecting the last option is usually fine. There are some simplifications and maybe some small performance optimizations in the creation and evaluation of the expressions that can be made for the other cases. But, **VisIt** will operate fine even if those are not chosen. In the description that follows, we demonstrate only this selection but describe variations where necessary.

After selecting the variety of CMFE expression to create, the user is presented with the next wizard window to specify the target mesh and donor variables to be used in the expression.

The target mesh selection will present the user with a pull-down list of currently opened databases with the currently active database in the list selected. If another database is desired, the user may either select it from among the pull-down list of currently open databases or, if the database is not yet open, press the ellipsis (3 dots) button next to the database selection list to open a file browser and navigate to the desired database in the file system as shown in Figure 1.218.

Once the database of the target mesh is specified, the target mesh within that database is specified with the **Target Mesh:** pull down list.

A similar sequence of steps is followed for specifying the donor variable. The example in Figure 1.219 demonstrates the selection of a specific donor variable from the donor database with the **Donor Variable:** pull down list.
Next, the user is presented with a window to specify the manner in which the CMFE expression is to be evaluated. The choices are either 

*connectivity-based* or *position-based*. A position-based CMFE is a more general evaluation at the likely expense of lower performance. When in doubt, it is best to use this option. Connectivity-based evaluation is applicable *only* when donor and target meshes are one-for-one *both* topologically and geometrically. In this case, VisIt can optimize the evaluation and avoid having to deal with cases where the donor and target meshes do not wholly overlap.

For a position-based CMFE, the user is required to also specify what VisIt should do for those positions on the target mesh that do not overlap with the mesh of the donor variable. The user can choose either a constant numerical value (e.g. a *fill value*) or can specify a variable already defined on the target mesh. It is possible for the user to make a choice that either enhances or inhibits one’s ability to distinguish between values in the result that come from the donor and values that come from the selected *fill* choice. A common practice is to choose a constant value that is an extremum of the donor variable’s range. For example, if the donor variable has a maximum value of 25.7, then selecting this as the constant to use for non-overlapping regions in the CMFE has the benefit of not altering the variable’s range but then also being indistinguishable from real data. Another practice is to choose a value that is easily distinguishable and later apply a threshold operator to remove those portions of the result.

The final step in the wizard is to give the result variable a name and then decide what to do with the result variable. In Figure 1.221, we have given the result variable the name `hardyglobal_onto_mesh1_from_globe`.

Often, it is sufficient to have VisIt just compute the mapped variable and then allow the user to use the result variable in other expressions. However, for convenience, the wizard also offers a number of options common to the work of comparing the mapped variable to another variable. This last window in the wizard allows the user to select from among several common methods for comparing the mapped variable to another variable on the target mesh. By selecting the *Expression with* option, the user is then offered the ability to select a variable already defined on the target mesh from the pull down list. Then, the user can select from one of several common methods for comparing the two variables. For example, the *Absolute value of difference* choice will have the effect of creating a single expression.
Fig. 1.218: Setting up the target mesh and donor variables
Fig. 1.219: Selecting a specific variable from a database

Fig. 1.220: Selecting the mode of evaluation
At any point during the steps in the wizard, the user can hit the Go Back button to go back and make different choices. The user completes the wizard by hitting the Done button. There is no way to go back after hitting the Done button. Upon completion of the wizard, a new expression is created according to user’s selections. This new expression can be edited in the expression window, like any other expression as illustrated in Figure 1.222.

In addition, this new expression can be used in other expressions. Finally, if for some reason the resulting expression is problematic, it can be deleted from the Expression system and the Data-Level Comparisons wizard can be run again to re-create it as desired.

1.9 Making it pretty

Now that you know how to visualize databases, it is time to learn how to make presentation quality visualizations. This chapter explains what options are available for making professional looking visualizations and introduces new windows that allow you to control annotations, colors, lighting, and the view.

1.9.1 Annotations

Annotations are objects in the visualization window that convey information about the plots. Annotations can be global objects that show information such as the database name, or they can be objects like plot legends that are directly tied to plots. Annotations are an essential component of a good visualization because they make it clear what is being visualized and make the visualization appear more polished.

VisIt supports several different annotation types that can be used to enhance visualizations. The first category of annotations includes general annotations like the database name, the user name, and plot legends. These annotations convey a good deal of information about what is being visualized, what values are in the plots, and who created the visualization. The second category of annotations include the plot axes and labels. This group of annotations comes in three groups: 2D, 3D and Array. The attributes for these groups can be set independently. Colors can
Fig. 1.222: New can be manipulated in the Expression window
greatly enhance the look of a visualization so VisIt provides controls to set the colors used for annotations and the visualization window that contains them. The third and final category includes annotation objects that can be added to the visualization window. You can add as many annotation objects as you want to a visualization window. The currently supported annotation objects are: 2D text, 3D text, time slider, 2D line, 3D line, and image annotations.

Annotation Window

The Annotation Window (Figure 1.223) contains controls for the various annotations that can appear in a visualization window. You can open the window choosing the Annotation option from the Main Window’s Controls menu. The Annotation Window has a tabbed interface which groups the different categories of annotations together.

Fig. 1.223: The Annotation window
General Annotations

VisIt has a few general annotations that describe the visualization and are independent of the type of database in the visualization. General annotations encompass the user name, the database name, and plot legends. The general annotation controls are located in the General tab. Figure 1.224 shows common locations for some general annotations.

Turning plot legends off globally

Plot legends are special annotations that are added by plots. An example of a plot legend is the color bar and title that the Pseudocolor plot adds to the visualization window. Normally, plot legends are turned on or off by a check box in
a plot attribute window but VisIt also provides a check box in the General tab that can turn off the plot legends for all the plots in the visualization window. You can use the Legend check box at the top of the General tab to turn plot legends off if they are present.

Displaying database information

When plots are displayed in the visualization window, the name of the database used in the plots is shown in the visualization window’s upper left corner. You can turn the database information on or off using the Database check box in the General tab.

The Path Expansion selection box controls the display of the filename text. File causes just the name of the file to be displayed. Directory causes the directory name of the file to be displayed. Full causes the full path of the file to be displayed. Smart uses simulation code specific conventions to display the file name in an optimal fashion. Smart Directory uses simulation code specific conventions to display the directory name in an optimal fashion.

The Time check box controls the display of the time associated with the current database. If Time is enabled then the Time scale factor and Time offset controls become active, allowing you to scale as well as apply an offset to the time associated with a database when displaying it.

Displaying user information

When you add plots to the visualization window, your username is shown in the lower right corner. The user information annotation is turned on or off using the User information check box. You may want to turn off user information when you are generating images for presentations.

2D Annotations

VisIt has a number of controls in the Annotation Window to control 2D annotations on the 2D tab (Figure 1.225). The 2D annotation settings are primarily concerned with the appearance of the 2D axes that frame plots of 2D databases. Figure 1.224 shows a plot with various annotations.

The Show axes check box turns on and off the display of the 2D axes.

General 2D axis properties

Auto scale label values causes the labels to be multiplied by a factor of 10 to a multiple of 3 power such that the labels are in the range 0.001 to 999. It then displays the multiplier in the axis title. An example is shown in Figure 1.226. The X-Axis range is 0 to 100,000, which causes the labels to be in the range 0 to 100, with a (x10^3) added to the X-Axis and Y-Axis labels to indicate that the true range is actually 0 to 100x10^3 or 100,000.

The tick marks are small lines that are drawn along the edges of the 2D viewport. Tick marks can be drawn on a variety of axes by selecting a new option from the Show tick marks menu. Tick marks can also be drawn on the inside, outside, or both sides of the plot viewport by selecting a new option from the Tick mark locations menu.

Tick mark spacing is usually changed to best suite the plots in the visualization window but you can explicitly set the tick mark spacing by first unchecking the Auto set ticks check box and then typing new tick spacing values into the Major minimum, Major maximum, Major spacing, and Minor spacing text fields in the X-Axis and Y-Axis tabs.

Setting the X-Axis and Y-Axis properties

There are tabs for separately controlling the properties of the X and Y axes. The tab for setting the X-Axis properties is shown in Figure 1.227.
Fig. 1.225: The general 2D properties
Fig. 1.226: 2D plot with axes labels being scaled by $10^3$
Fig. 1.227: The 2D axes properties
The axis titles are the names that are drawn along each axis, indicating the meaning of the values shown along the axis. Normally, the names used for the axis titles come from the database being plotted so the axis titles are relevant for the displayed plots. Many of VisIt’s database readers plugins read file formats that have no support for storing axis titles so VisIt uses default values such as: “X-Axis”, “Y-Axis”. VisIt provides options that allow you to override the defaults or the axis titles that come from the file. You can control the display of the axis titles by enabling and disabling the Title check box. If you want to override the axis titles that VisIt uses for 2D visualizations, turn on the Custom title check box and type the new axis title into the adjacent text field.

In addition to overriding the names of the axis titles, you can also override the units that are displayed next to the axis titles. Units are displayed only when they are available in the file format and like axis titles, they are not always stored in the file being plotted. If you want to specify units for the axes, turn on the Custom Units check box and type new units into the adjacent text field.

The axis labels are the labels that appear along the 2D plot viewport. By default, the axis labels are enabled and set to appear. You can turn the labels off by unchecking the Labels check box. You can change the label scale factor by changing the Scaling (x10^?) text field.

Tick mark spacing is usually changed to best suite the plots in the visualization window but you can explicitly set the tick mark spacing by first unchecking the Auto set ticks check box on the General 2D tab and then typing new tick spacing values into the Major minimum, Major maximum, Major spacing, and Minor spacing text fields.

The 2D grid lines are a set of lines that make a grid over the 2D viewport. The grid lines are disabled by default but you can enable them by checking the Show grid check box. The grid lines correspond to the major tick marks.

### 3D Annotations

VisIt has a number of controls, located on the 3D tab in the Annotation Window for controlling annotations that are used when the visualization window contains 3D plots. Like the 2D controls, these controls focus mainly on the axes that are drawn around plots. Figure 1.228 shows an example 3D plot with the 3D annotations. Figure 1.229 and Figure 1.230 shows the Annotation Window’s 3D tab.

The Show axes check box turns on and off the display of the 3D axes.

The Show triad check box turns on and off the display of the triad annotation. The triad annotation consists of a small set of axes and is displayed in the lower left corner of the visualization window and help you get your bearings in 3D.

The Show bounding box check box turns on or off the display of the bounding box. The bounding box annotation displays the edges of a box that contains all the data.

### General 3D axis properties

Auto scale label values causes the labels to be multiplied by a factor of 10 to a multiple of 3 power such that the labels are in the range 0.001 to 999. It then displays the multiplier in the axis title. A 2D example is shown in Figure 1.226. The X-Axis range is 0 to 100,000, which causes the labels to be in the range 0 to 100, with a (x10^3) added to the X-Axis and Y-Axis labels to indicate that the true range is actually 0 to 100x10^3 or 100,000.

The tick marks are small lines that are drawn along the edges of the bounding box surfaces. Tick marks can be drawn on a variety of axes by selecting a new option from the Show tick marks menu. Tick marks can also be drawn on the inside, outside, or both sides of the plot bounding box by selecting a new option from the Tick mark locations menu.

Tick mark spacing is usually changed to best suite the plots in the visualization window but you can explicitly set the tick mark spacing by first unchecking the Auto set ticks check box and then typing new tick spacing values into the Major minimum, Major maximum, Major spacing, and Minor spacing text fields in the X-Axis, Y-Axis and Z-Axis tabs.
Fig. 1.228: 3D plot with annotations
Fig. 1.229: The general 3D properties
Setting the X-Axis, Y-Axis and Z-Axis properties

There are tabs for separately controlling the properties of the X, Y and Z axes. The tab for setting the X-Axis properties is shown in Figure 1.230.

![Fig. 1.230: The 3D axes properties](image)

The axis titles are the names that are drawn along each axis, indicating the meaning of the values shown along the axis. Normally, the names used for the axis titles come from the database being plotted so the axis titles are relevant for the displayed plots. Many of VisIt’s database readers plugins read file formats that have no support for storing axis titles so VisIt uses default values such as: “X-Axis”, “Y-Axis” and “Z-Axis”. VisIt provides options that allow you to override the defaults or the axis titles that come from the file. You can control the display of the axis titles by enabling and disabling the Title check box. If you want to override the axis titles that VisIt uses for 3D visualizations, turn on the Custom title check box and type the new axis title into the adjacent text field.

In addition to overriding the names of the axis titles, you can also override the units that are displayed next to the axis.
titles. Units are displayed only when they are available in the file format and like axis titles, they are not always stored in the file being plotted. If you want to specify units for the axes, turn on the Custom Units check box and type new units into the adjacent text field.

The axis labels are the labels that appear along the edges of the bounding box. By default, the axis labels are enabled and set to appear. You can turn the labels off by unchecking the Labels check box. You can change the label scale factor by changing the Scaling \((x10^?)\) text field.

Tick mark spacing is usually changed to best suite the plots in the visualization window but you can explicitly set the tick mark spacing by first unchecking the Auto set ticks check box on the General 3D tab then typing new tick spacing values into the Major minimum, Major maximum, Major spacing, and Minor spacing text fields.

The 3D grid lines are a set of lines that make a grid over the bounding box. The grid lines are disabled by default but you can enable them by checking the Show grid check box. The grid lines correspond to the major tick marks.

**Annotation Colors**

Colors are very important in a visualization since they help to determine how easy it is to read annotations. VisIt provides a tab in the Annotation Window, shown in Figure 1.231, specifically devoted to choosing annotation colors. The Colors tab contains controls to set the background and foreground for the visualization window which, in turn, set the colors used for annotations. The Colors tab also provides controls for more advanced background colors called gradients which are colors that bleed into each other.  

The Background color and Foreground color buttons allow you to set the background and foreground colors. To set the color, click the color button and select a color from the Popup color menu (see Figure 1.232). Releasing the mouse outside of the Popup color menu cancels color selection and the color is not changed. Once you select a new color and click the Apply button, the colors for the active visualization window change. Note that each visualization window can have different background and foreground colors.

The Background style setting allows you to select from four background styles. The default background style is Solid where the entire background is a single color. The second style is a Gradient background. In a gradient background, two colors are blended into each other in various ways. The resulting background offers differing degrees of contrast and can enhance the look of many visualizations. The third style is an Image background, where an image is tiled across the background. The fourth style is an Image sphere, where an image is projected onto a sphere. This can be used to paint the stars onto the background of an astrophysics simulation. To change the background style, click the Background style radio buttons.

VisIt provides controls for setting the colors and style used for gradient backgrounds. There are two color buttons: Gradient color 1 and Gradient color 2 that are used to change colors. To change the gradient colors, click on the color buttons and select a color from the Popup color menu. The gradient style is used to determine how colors blend into each other. To change the gradient style, make a selection from the Gradient style menu. The available options are Bottom to Top, Top to Bottom, Left to Right, Right to Left, and Radial. The first four options blend gradient color 1 to gradient color 2 in the manner prescribed by the style name. For example, Bottom to Top will have gradient color 1 at the bottom and gradient color 2 at the top. The radial gradient style puts gradient color 1 in the middle of the visualization window and blends gradient color 2 radially outward from the center. Examples of the gradient styles are shown in Figure 1.233.

The Background image text field allows you to specify the name of the file to use for the background image. The Repetitions in X and Repetitions in Y settings allow you to specify how many times to replicate the image in each of the X and Y image directions.

**Annotation Objects**

So far, the annotations that have been described can only have a single instance. To provide more flexibility in the types and numbers of annotations, VisIt allows you to create annotation objects, which are objects that are added to the visualization window to convey information about the visualization. Currently, VisIt supports six types of annotation
Fig. 1.231: The annotation colors tab
Fig. 1.232: The popup color menu and the color selection dialog

Fig. 1.233: The various gradient styles
objects: 2D text objects, 3D text objects, time slider objects, 2D line objects, 3D line objects and image objects. All of those types of annotation objects will be described herein. The **Objects** tab, in the **Annotation Window** (Figure 1.234) is devoted to managing the list of annotation objects and setting their properties.

![Annotation Window](image)

**Fig. 1.234:** The annotation objects tab

The **Objects** tab in the **Annotation Window** is divided up into three main areas. The top of the window is split vertically into two areas that let you create new annotation objects and manage the list of annotation objects. The bottom half of the **Objects** tab displays the controls for setting the attributes of the selected annotation object. Each annotation object provides a separate user interface that is tailored for setting its particular attributes. When you select an annotation in the annotation object list, the appropriate annotation object interface is displayed.
Creating a new annotation object

The Create new area in the Annotation Window’s Objects tab contains one button for each type of annotation object that VisIt can create. Each button has the name of the type of annotation object VisIt creates when you push it. After pushing one of the buttons, VisIt creates a new instance of the specified annotation object type, adds a new entry to the Annotation objects list, and displays the appropriate annotation object interface in the bottom half of the Objects tab to display the attributes for the new annotation object.

Selecting an annotation object

The Objects tab displays the annotation object interface for the selected annotation object. To set attributes for a different annotation object, or to hide or delete a different annotation object, you must first select a different annotation object in the Annotation objects list. Click on a different entry in the Annotation objects list to highlight a different annotation object. Once you have highlighted a new annotation object, VisIt displays the object’s attributes in the lower half of the Objects tab.

Hiding an annotation object

To hide an annotation object, select it in the Annotation objects list and then click the Hide/Show button on the Objects tab. To show the hidden annotation object, click the Hide/Show button a second time. The interfaces for the currently provided annotation objects also have a Visible check box that can be used to hide or show the annotation object.

Deleting an annotation object

To delete an annotation object, select it in the Annotation objects list and then click the Delete button on the Objects tab. You can delete more than one object if you select multiple objects plots in the Annotation objects list before clicking the Delete button.

Text annotation objects

Text annotation objects, shown in Figure 1.235, are created by clicking the Text button in the Create new area on the Objects tab. Text annotation objects are simple 2D text objects that are drawn on top of plots in the visualization window and are useful for adding titles to a visualization.

The text annotation object properties, shown in Figure 1.236, can be used to set the position, size, text, colors, and font properties.

Text annotation objects are placed using 2D coordinates where the X, and Y values are in the range [0,1]. The point (0,0) corresponds to the lower left corner of the visualization window and the point (1,1) corresponds to the upper right of the visualization window. The 2D coordinate used to position the text annotation matches the text annotation’s lower left corner. To position a text annotation object, enter a new 2D coordinate into the Lower left text field. You can also click the down arrow next to the Lower left text field to interactively choose a new lower left coordinate for the text annotation using the screen positioning control, which represents the visualization window. The screen positioning control, shown in Figure 1.237, lets you move a set of cross-hairs to any point on a square area that represents the visualization window. Once you release the left mouse button, the location of the cross-hairs is used as the new coordinate for the text annotation object’s lower left corner.

The size of the text is set using the Height spin box. The height is the fraction of the visualization window height.

To set the text that a text annotation object displays, type a new string into the Text text field. You can make the text annotation object display any characters that you type in but you can also use the $time wildcard string to make
Fig. 1.235: Examples of text annotations
Fig. 1.236: The text annotation interface

Fig. 1.237: Screen positioning control
the text annotation object display the time for the current time state of the active database. A text string of the form: Time=$time will display Time=10 in the visualization window when the active database’s time is 10. Whatever text you enter for the text annotation object is used to identify the text annotation object in the Annotation objects list.

In addition to the usual text properties, text annotation objects can also include a shadow.

### 3D text annotation objects

3D text annotation objects, shown in Figure 1.238, are created by clicking the 3D Text button in the Create new area on the Objects tab. 3D text annotation objects are extruded text that are positioned in 3D and are part of the 3D scene, so they may become obscured by other objects in the scene and will move in space as the image is panned and zoomed.

![Fig. 1.238: Examples of 3d text annotations](image)

The 3D text annotation object properties, shown in Figure 1.239, can be used to set the text, position, size, orientation and color properties.

To set the text that a 3D text annotation object displays, type a new string into the Text text field.

3D text annotation objects are placed in 3D coordinates in the same coordinate system used by the simulation data. To position a 3D text annotation object, enter a new 3D coordinate into the Position text field.

The size of the text can be specified in two different ways. The first is using a relative height, where the height is a fraction of the size of the simulation data. The second is a fixed size, where the size is specified in the coordinate.
system of the simulation data. If you were to specify a relative height and apply the Transform operator to scale the
data in each direction by a factor of 10, the size of the text would not change. If you were to specify a fixed height,
scaling the data by a factor of 10 would result in the text being one tenth the size. To specify a relative height, select
the Relative radio button and set the size using the spin box next to it. To specify a fixed height, select the Fixed
radio button and enter the new height in the text box next to it.

The orientation of the text can also be specified in two different ways. The first is relative to the screen coordinate
system and the second is in the coordinate system of the simulation data. If the orientation is relative to the screen
coordinate system, then rotating the image will not change the orientation of the text. If the orientation is relative to
the coordinate system of the simulation data, then rotating the image will change the orientation of the text. To make
the orientation relative to the screen, select the Preserve orientation when view changes radio button. To make the
orientation relative to the simulation coordinate system, uncheck the Preserve orientation when view changes radio
button. To set the orientation, set the Rotate Y, Rotate X and Rotate Z spin boxes. The rotations are applied in the
left to right order of the spin boxes in the interface.

Time slider annotation objects

Time slider annotation objects, shown in Figure 1.240, are created by clicking the Time slider button in the Create
new area on the Objects tab. Time slider annotation objects consist of a graphic that shows the progress through an
animation using animation and text that shows the current database time. Time slider annotation objects can be placed
anywhere in the visualization window and you can set their size, text, colors, and appearance properties.

Time slider annotation objects are placed using 2D coordinates where the X, and Y values are in the range [0,1]. The
point (0,0) corresponds to the lower left corner of the visualization window and the point (1,1) corresponds to the upper
right of the visualization window. The 2D coordinate used to position the text annotation matches the text annotation’s
lower left corner. To position a text annotation object, enter a new 2D coordinate into the Lower left text field. You
can also click the down arrow next to the Lower left text field to interactively choose a new lower left coordinate for
the text annotation using the screen positioning control, which represents the visualization window.

The size of a time slider annotation object is controlled by settings its height and width as a percentage of the visu-
alization window height and width. Type new values into the Width and Height spin buttons to set a new width or
height for the time slider annotation object.
Fig. 1.240: An example of a time slider annotation object
You can set the text displayed by the time slider annotation object by typing a new text string into the Text label text field. Text is displayed below the time slider annotation object and it can contain any message that you want. The text can even include wildcards such as $time$, which evaluates to the current time for the active database. If you use $time$ to make VisIt incorporate the time for the active database, you can also specify the format string used to display the time. The format string is a standard C-language format string (e.g. “%4.6g”) and it determines the precision used to write out the numbers used in the time string. You will probably want to specify a format string that uses a fixed number of decimal places to ensure that the time string remains the same length during the animation, preventing distracting differences in the length of the string from taking the eye away from the visualization. Type a C-language format string into the Time format text field to change the time format string.

Time slider annotations have three color attributes: start color, end color, and text color. A time slider annotation object displays time like a progress bar in that the progress bar starts out small and then grows to the right until it takes up the whole length of the annotation. The color used to represent the progress can be set by clicking the Start color button and choosing a new color from the Popup color menu. As the time slider annotation object shows more progress, the color that is used to fill up the time that has not been reached yet (end color) is overtaken by the start color. To set the end color for the time slider annotation object, click the End color button and choose a new color from the Popup color menu. Normally, time slider annotation objects use the foreground color of the visualization window when drawing the annotation’s text. If you want to make the annotation use a special color, turn off the Use foreground color check box and click the Text color button and choose a new color from the Popup color menu.

Time slider objects have two more attributes that affect their appearance. The first of those attributes is set by clicking on the Rounded check box. When a time slider annotation object is rounded, the ends of the annotation are curved. The last attribute is set by clicking on the Shaded check box. When a time slider annotation object is shaded, simple lighting is applied to its geometry and the annotation will appear to be more 3-dimensional.

**2D line annotation objects**

2D line annotation objects, shown in Figure 1.243, are created by clicking the 2D Line button in the Create new area on the Objects tab. 2D line annotation objects are simple line objects that are drawn on top of plots in the visualization window and are useful for pointing to features of interest in a visualization. 2D line annotation objects can be placed anywhere in the visualization window and you can set their locations, arrow properties, and color.
Fig. 1.242: Examples of 2D line annotations
2D line annotations are described mainly by two coordinates that specify the start and end points for the line. The start and end coordinates are specified as pairs of floating point numbers in the range [0,1] where the point (0,0) corresponds to the lower left corner of the visualization window and the point (1,1) corresponds to the upper right corner of the visualization window. You can set the start or end points for the 2D line annotation by entering new start or end points into the **Start** or **End** text fields in the 2D line object interface. You can also click the down arrow to the right of the **Start** or **End** text fields to interactively choose new coordinates using the screen positioning control.

![2D Line Object Interface](image)

**Fig. 1.243: The 2D line object interface**

Once the 2D line annotation has been positioned there are other attributes that can be set to improve its appearance. First of all, if the 2D line annotation is being used to point at important features in a visualization, you might want to increase the 2D line annotation’s width to make it stand out more. To change the width, select the new pixel width from the **Width** menu. It is also possible to set the line style. To change the style of the line, select the new line style from the **Style** menu. After changing the width and style, the color of the 2D line annotation should be chosen to stand out against the plots in the visualization. The color that you use should be chosen such that the line contrasts sharply with the plots over which it is drawn. To choose a new color for the line, click on the **Line color** button and choose a new color from the **Popup color** menu. You can also adjust the opacity of the line by using the opacity slider next to the **Line color** button.

The last properties that are commonly set for 2D line annotations determine whether the end points of the line have arrow heads. The 2D line annotation supports two different styles of arrow heads: filled and lines. To make your line have arrow heads at the start or the end, make new selections from the **Begin arrow** and **End arrow** menus.

### 3D line annotation objects

3D line annotation objects, shown in Figure 1.238, are created by clicking the **3D Line** button in the **Create new** area on the **Objects** tab. 3D line annotation objects are lines that are positioned in 3D and are part of the 3D scene, so they may become obscured by other objects in the scene and will move in space as the image is panned and zoomed.

The 3D line annotation object properties, shown in Figure 1.244, can be used to set the position, style and color properties.

3D text annotation objects are placed in 3D coordinates in the same coordinate system used by the simulation data. To position a 3D line annotation object, specify the start and end location of the line by entering the start location in the **Start** text field and the end location in the **End** text field.
There are two types of lines supported, one is a normal line and the other is a tube. The line type is selected through the Line type menu. When using a normal line, you can specify the normal line width and line style properties using the Line Width and Line Style menus. When using a tube you can specify the tube quality and radius. The tube is created from a series of flat surfaces around the center of the line to approximate a tube. The number of surfaces used is controlled by the tube quality. The tube radius is the radius of the tube in the coordinate system of the simulation data. These properties can be changed through the Tube Quality and Tube Radius menus.

It is also possible to add arrows to the beginning and end of the line. These can be enabled with the Begin Arrow and End Arrow toggle buttons. For each arrow, the user can also control the resolution and radius of the arrows. The arrows consist of cones placed at the ends of the line and are constructed out of triangles that approximate a cone. The number of triangles used is controlled by the resolution. The radius is the radius of the cone in the same coordinate system as the simulation data. The resolution can be changed using the Resolution spin box and the radius is changed by typing a new value into the Radius text field.

**Image annotation objects**

Image annotation objects, shown in Figure 1.245, are created by clicking the Image button in the Create new area on the Objects tab. Image annotation objects display images from image files on disk in a visualization window. Images are drawn on top of plots in the visualization window and are useful for adding logos, pictures of experimental data, or other views of the same visualization. Image annotation objects can be placed anywhere in the visualization window and you can set their size, and optional transparency color.

The first step in incorporating an image annotation into a visualization is to choose the file that contains the image that will serve as the annotation. To choose an image file for the image annotation, type in the full path and filename to the file that you want to use into the Image source text field. You can also use the file browser to locate the image file if you click on the “…” button to the right of the Image source text field in the Image annotation interface, shown in Figure 1.246. Note that since image annotations are incorporated into a visualization inside of VisIt's viewer component, the image file must be located on the same computer that runs the viewer.

After selecting an image file, you can position its lower left coordinate in the visualization window. The lower left corner of the visualization window is the origin (0,0) and the upper right corner of the visualization window is (1,1).

Once you position the image where you want it, you can optionally scale it relative to its original size. Unlike some
Fig. 1.245: An Example of a visualization with two overlaid image annotations

![Visualization with overlayed image annotations](image.png)

Fig. 1.246: The image object interface

![Image object interface](image.png)
other annotation objects, the image annotation does not scale automatically when the visualization window changes size. The image annotation will remain the same size - something to take into account when setting up movies that use the image annotation. To scale the image relative to its original size, enter new percentages into the **Width** and **Height** spin boxes. If you want to scale one dimension of the image and let the other dimension remain unchanged, turn off the **Lock aspect** check box.

Finally, if you are overlaying an image annotation whose image contains a constant background color or other area that you want to remove, you can pick a color that VisIt will make transparent. For example, Figure 1.245 shows an image of some Curve plots overlaid on top of the plots in the visualization window and the original background color in the annotation object was removed to make it transparent. If you want to make a color in an image transparent before VisIt displays it as an image annotation object, click on the **Transparent color** check box and then select a new color by clicking on the **Transparent color** button and picking a new color from the **Popup color menu**.

### 1.9.2 Color Tables

A color table is a set of colors that is used by certain plots to color variables. Color tables can be immensely important for understanding visualizations since changes in color can highlight interesting features. VisIt has several built-in color tables that can be used in visualizations. VisIt also provides a **Color table window** for designing custom color tables.

Color tables come in two types: continuous and discrete. A continuous color table is defined as a relatively few color control points defined at certain intervals in the color table and the gaps in between the color control points are filled by smoothly interpolating the colors. This makes continuous color tables look smooth since there are several colors that are blended to form the color table. Continuous color tables are used by several plots including the Pseudocolor, Tensor, and Vector plots. A plot that uses a continuous color table attempts to use all of the colors in the color table. Some plots that opt to only use a handful of colors from a continuous color table pick colors that are evenly distributed through the color table so that the plots end up with colors that still somewhat resemble the original colors from the continuous color table.

A discrete color table is a set of N colors that can be set individually. There are no other colors in a discrete color table other than the colors that you provide. Discrete color tables are usually used by plots like the Boundary, Contour, FilledBoundary, or Subset plots, which need only a small set of colors. Typically, these plots use a color from a discrete color table to color some object and then use the next color to color another object, and so on. When they reach the end of the color table and still need more colors, they start again at the beginning with the first color from the discrete color table.

#### Color Table Window

You can open VisIt’s **Color table window**, shown in Figure 1.247, by selecting **Color table** from the **Main Window's Controls** menu. The **Color table window** is vertically separated into three areas. The top area allows you to set the active color tables. The middle area, or manager portion of the window, allows you to create or delete new color tables, as well as export color tables. The bottom area, or editor portion of the window, allows you to edit color tables by adding, removing, moving, or changing the color of color control points. A color control point is a point with a color that influences how the color table will look.

#### Setting the active color table

VisIt has the concept of active color tables, which are the color tables used to color plots that do not specify a color table. There is both an active continuous color table (for plots that prefer to use continuous color tables) and an active discrete color table (for plots that prefer to use discrete color tables). The active color table can be different for each visualization window. To set the active continuous color table, select a new color table name from the **Continuous** menu in the **Active color table** area. To select a new active discrete color table, select a new color table name from the **Discrete** menu in the **Active color table** area.
Fig. 1.247: The color table window
Creating a new color table

Creating a new color table is a simple process where you first type a new color table name into the Name text field and then click the New button. This creates a copy of the currently highlighted color table, which is the color table that is selected in the Manager area, and inserts it into the color table list with the specified name. After creating the new color table, you can modify the color control points to fashion a new color table.

Deleting a color table

To delete a color table, click on a color table name in the color table list and then click the Delete button. You can delete all color tables except for the last color table. VisIt makes no distinction between built-in color tables and user-defined color tables so any color table can be deleted. When you delete a color table, the active color table is set to the color table that comes first in the list. If a color table is in use when it is deleted, plots that used the deleted color table will use the default color table from that point on.

Exporting a color table

If you design a color table that you want to share with colleagues, click the Export button in the Manager area to save an XML file containing the color table definition for the highlighted color table to your .visit directory. The name of a color table file will usually be composed of the name of the color table with a “.ct” extension. Copying a color table file to a user’s .visit directory will allow VisIt to find the color table the next time VisIt runs. Look for the color table file in the directory in which VisIt was installed if you use the Windows version of VisIt.

Editing a continuous color table

![Image of Color Table Editor]

Fig. 1.248: The continuous color table editor

There are a handful of controls in the editor portion of the Color table window, shown in Figure 1.248, that are used to change the definition of a color table. To change a color table definition, you must alter its color control points. This means adding and removing color control points as well as changing their colors and locations.
You can change the number of color control points in a color table using the **Number of colors** spin box. When a new color control point is added, it appears to the right of the selected color control point and to the left of the next color control point. Color control points are represented as a pointy box just above the color spectrum. The color control point that has a small triangular mark is the selected color control point. When a color control point is removed, the color control point that was created before the deleted color control point becomes the new selected color control point. Clicking the **Align** button makes all color control points have equal spacing.

Clicking on a color control point makes it active. You can also use the Space bar if the color spectrum has keyboard focus. Clicking and dragging on a color control point changes its position. Clicking the arrow keys on the keyboard also moves a color control point. To change a color control point’s color, right click on it and choose a new color from the **Popup color** menu that appears under the mouse cursor. You can also change the color control point’s color by making the color control point active and then using the **Red**, **Green** and **Blue** sliders.

The **Color table window** also has a couple of settings that can be set to influence a color table’s appearance without having permanent effects on the color table. The **Smoothing** menu can be used to select between no smoothing, linear smoothing and cubic spline smoothing. The **Equal** check box can temporarily tell the color table to ignore the positions of its color control points and use equal spacing instead. The **Equal** check box is often used with no smoothing.

**Editing a discrete color table**

![Discrete color table editor](image)

**Fig. 1.249: The discrete color table editor**

The **Color table window’s Editor** area looks different when you edit a discrete color table. Instead of showing a spectrum of colors, the window shows a grid of colors that correspond to the colors in the discrete color table. The order of the color control points if left to right, top to bottom. To edit a discrete color table, first left click on the color that you want to edit and then use the **Red**, **Green**, and **Blue** sliders to change the color. You can also right click on a color to select it and open the **Popup color** menu to choose a new color.
Converting color table types

It is possible to convert a continuous color table to a discrete color table and vice-versa using the Continuous and Discrete radio buttons in the editor portion of the Color table window. Changing the color table type from discrete to continuous does not change the color table’s color control points; it only changes how they are used. If you select the levels color table and click the Continuous radio button, the color table will be changed into a continuous color table and the Editor area will change to continuous mode and show the color table in a spectrum but no color control points will have changed. You can even turn the color table back into a discrete color table and the Editor area will show the color table in discrete mode, but the color control points will not have changed.

1.9.3 Lighting

Lighting is an important element when producing 3D visualizations because all areas of interest in the visualization should be lit so they can be easily seen. To this end, it is often necessary to have multiple light sources so all of the visualization’s important areas are bright enough. VisIt can have up to 8 light colored light sources in order to improve the look of 3D visualizations. Each light source can be positioned and colored using VisIt’s Lighting Window. It is also possible to have specular highlights in addition to multiple colored lights. For more information on specular highlights, which can make visualizations appear much more realistic, read about specular lighting in the Preferences chapter.

Lighting Window

You can open the Lighting Window (see Figure 1.250) by selecting the Lighting option from the Main Window’s Controls menu. The Lighting Window has two modes of operation: edit and preview. When the window is in preview mode, light sources cannot be modified, but they are all visible and illuminate the Lighting Window’s test sphere so the cumulative effect of the lights can be observed. When the window is in edit mode, light sources can be modified one at a time. You set light properties using the controls in the Properties panel and you can position lights interactively by moving them around in the lighting panel to the left of the Properties panel.

Switching between edit mode and preview mode

Changing the Mode between Edit and Preview switches the Lighting Window into the desired mode. When the Lighting Window is in edit mode, one light source at a time is shown in the lighting panel and the lights properties can be set by moving the light interactively or by settings its properties by using the controls in the Properties panel. When the Lighting Window is in preview mode, all lights are shown in the lighting panel and none of them can be modified.

Choosing the active light

The active light is the light whose properties are shown in the Lighting Window. Only the active light can be modified so you must switch active lights each time you want to make changes to a light. To change the active light, select a new light from the Active light menu (Figure 1.251). The Active light menu contains a list of eight possible lights of which only light 1 is active by default. When a light is active, it has a small light bulb icon next to it. Inactive lights have no light bulb icon. Once a new light has been selected from the Active light menu, its properties are displayed in the Lighting Window’s Properties panel.
Fig. 1.250: The lighting Window

Fig. 1.251: The active light menu
Turning a light on

You can turn lights on and off using the Enabled check box that appears at the bottom of the Lighting Window’s Properties panel. You can only modify lights when the Lighting Window is in edit mode.

Light type

![Image of different kinds of lights]

Fig. 1.252: The different kinds of lights

VisIt supports three types of lights. The first type is called an ambient light. An ambient light is a light that has no direction and contributes brightness to the entire visualization. When an ambient light is present, the lighting panel displays a small light bulb. The second type of light and the default light in VisIt is a camera light. A camera light stays fixed in space and always points the same direction regardless of how the objects in the visualization are positioned. Camera lights are represented in the lighting panel as small blue arrows. The third type of light in VisIt is the object light. An object light has a direction that is relative to the orientation of the object in the visualization. When the objects in the visualization are rotated, an object light keeps shining on the same area of the object. Object lights are represented in the lighting panel as small yellow cones. To change the light type for the active light, select a new light type from the Light type menu in the Properties panel.

Positioning a light

There are two ways to position a light. The first, and most intuitive, way is to interactively position the light by dragging it to the desired location in the lighting panel. Lights move in a sphere around the test sphere. Experiment with the motion until you are comfortable moving the light. The second way to move the light is to type a direction vector into the Direction text field. The coordinate system for specifying a direction vector is right-handed. Suppose you want to create a light that looks directly into the visualization. Since the Z-axis points directly out of the screen, the negative Z-axis points into the screen. This can be captured by entering a direction vector of: 0 0 -1. Note that ambient lights have no direction.
Light color and brightness

VisIt allows lights to have color as well as brightness. Colored lighting can produce interesting effects that may be desirable for presentations. To change the light color, click on the light Color button and select a new color from the Color menu. Once a color is picked, you can also set the brightness for the light. The brightness is essentially a knob that allows you to dim the light. If the brightness is set completely to the right then the light will have exactly the color that was picked for it. If the brightness is not set to full intensity then the light will be dimmer. You can set the brightness by adjusting the Brightness slider in the Lighting Window.

1.9.4 Rendering Options

VisIt provides support for setting various global rendering options that improve quality and realism of the plots in the visualization. Specifically, VisIt provides controls that let you smooth the appearance of lines, add specular highlights, add shadows, and apply depth cueing to plots in your visualizations. The controls for setting these options are located in the Rendering options Window (see Figure 1.253) and they will be covered here while the other controls in that window will be covered in the Preferences chapter. To open the Rendering options Window, click on Rendering in the Main Window’s Preferences menu.

Making Lines Look Smoother

Computer monitors contain an array of millions of tiny rectangular pixels that light up to form patterns which your eyes perceive as images. Lines tend to look blocky on computer monitors because they are drawn using a relatively small set of pixels. Lines can be made to look better by blending the edges of the line with the color of the background image. This is a form of antialiasing that VisIt can use to make plots which use lines, such as the Mesh plot, look better (see Figure 1.254). If you want to enable antialiasing, which is off by default, you check the Antialiasing check box located at the top of the Basic tab (see Figure 1.253). When antialiasing is enabled, all lines drawn in a visualization window are blended with the background image so that they look smoother.

Specular Lighting

VisIt supports specular lighting, which results in bright highlights on surfaces that reflect a lot of incident light from VisIt’s light sources. Specular lighting is not handled in the Lighting Window because specular lighting is best described as a property of the material reflecting the light. The controls for specular lighting don’t control any lights but instead control the amount of specular highlighting caused by the plots. Specular lighting is not enabled by default. To enable specular lighting, click the Specular lighting check box near the bottom of the Basic tab (see Figure 1.253).

Once specular lighting is enabled, you can change the strength and sharpness properties of the material reflecting the light. The strength, which you can set using the Strength slider, influences how glossy the plots are and how much light is reflected off of the plots. The sharpness, which is set using the Sharpness slider, controls the locality of the reflections. Higher sharpness values result in smaller specular highlights. Specular highlights are a crucial component of lighting models and including specular lighting in your visualizations enhances their appearance by making them more realistic. Compare and contrast the plots in Figure 1.255. The plot on the left side has no specular highlights and the plot on the right side has specular highlights.

Shadows

VisIt supports shadows when scalable rendering is being used. Shadows can be useful for increasing the realism of your visualization. The controls to turn on shadows can be found near the bottom of the Advanced tab (see Figure 1.256). To turn on shadows, you must turn on scalable rendering by clicking on the Always radio button under the Use scalable rendering label. Once scalable rendering has been turned on, the shadows controls become enabled. The default shadow strength is 50%. If you desire a stronger or weaker shadow, adjust the Strength slider until you
Fig. 1.253: The basic rendering options

Fig. 1.254: An example of antialiasing
are satisfied with the amount of shadow that appears in the visualization. The same plot is shown with and without shadows in Figure 1.257.

**Depth Cueing**

VisIt supports depth cueing when scalable rendering is being used. Depth cueing can be useful for increasing the realism of your visualization. Depth cueing causes objects to be blended with the background with increasing distance from the camera. The controls to turn on depth cueing can be found near the bottom of the Advanced tab (see Figure 1.256). To turn on depth cueing, you must turn on scalable rendering by clicking on the **Always** radio button under the **Use scalable rendering** label. Once scalable rendering has been turned on, the depth cueing controls become enabled. By default, depth cueing is performed along the camera direction. The depth cueing can be done along a different direction by unchecking the **Cue automatically along camera depth** check box and then entering the coordinates defining the direction to perform the depth cueing in the **Manual start point** and **Manual end point** text fields. The coordinates are defined in the coordinate system of the simulation data. The same plot is shown with and without depth cueing in Figure 1.258.

**1.9.5 View**

The view is one of the most critical properties of a visualization since it determines what parts of the dataset are seen. The view is also one of the most difficult properties to set. It is not that the act of setting the view is difficult. In fact, it is quite the opposite. The problem with setting the view is finding a flattering view for a database that will continue to be a good view for the entire life of the visualization. Many plots will deform or expand over the course of an animation and you have to decide how to pick a good view. You can pick a view that is zoomed way out and then let your plots expand and deform until they make good use of the visualization window. You can also decide to keep changing the view throughout the animation. A common technique is to interpolate views or do some sort of fly-by animation when the plots in the animation are expanding or not behaving in a static manner. The fly-by animation is used to distract the audience from the fact that you need to change to a more suitable view.

The view in VisIt can be set in two different ways. The first and best way to set the view is to navigate to it interactively in the visualization window. This is the fastest and most direct way of setting the view. The problem with setting the
Fig. 1.256: The advanced rendering options
Fig. 1.257: The effects of shadows on plots

Fig. 1.258: The effects of depth cueing on plots
view in this manner is that it is not very reproducible. It is often the case that users want to look at the same feature in their database using the same view. VisIt provides a View Window that they can use to set the view information exactly the same every time.

**View Window**

You can open the View Window by selecting View from the Main Window’s Controls menu. The View Window is divided into five tabbed sections. The first tab sets the curve view, the second tab sets the 2D view, the third tab sets the 3D view, the fourth tab sets the axis array view, and the last tab sets advanced view options. The View Window also contains a Command text field at the bottom for entering view commands.

**Setting the curve view**

Visualization windows that contain Curve plots use a special type of view known as a curve view. A curve view consists of: viewport, domain, and range. The viewport is the area of the visualization window that will be occupied by the plots and is specified using X and Y values in the range [0,1]. The point (0,0) corresponds to the lower-left corner of the visualization window while the point (1,1) corresponds to the visualization window’s upper-right corner. To change the viewport, type new numbers into the Viewport text field on the Curve view tab of the View Window (Figure 1.259). The minimum and maximum X values should come first, followed by the minimum and maximum Y values.

The domain and range refer to the limits on the X and Y axes. You can set the domain, which is the range of X values that will be displayed in the viewport, by typing new minimum and maximum values into the Domain text field. You should use domain values that use the same dimensions as the Curve plot that will be plotted in the visualization window. You can set the range, which is the range of Y values that will be displayed in the viewport, by typing new values into the Range text field. The domain and range values may also be log scaled and may be controlled independently. To log scale the domain, check the Log radio box to the right of the Domain Scale label. To log scale the range, check the Log radio box to the right of the Range Scale label.

**Setting the 2D view**

Setting the 2D view is conceptually simple. There are only two pieces of information that you need to supply. The first piece of information that you must enter is the viewport, which is an area of the visualization window in which you want the 2D plots to appear. Imagine that the lower left corner of the visualization window is the origin of a coordinate system and that the upper left and lower right corners both have values of 1. Every point in the visualization window can be characterized as a Cartesian coordinate where both values in the coordinate are in the range [0,1]. The viewport is specified by entering four numbers in the form x0 x1 y0 y1 where x0 is the leftmost X value, x1 is the rightmost X value, y0 is the lower Y value, and y1 is the upper Y value that will be used in the viewport. The window is an area in the space occupied by the 2D plots. You can start with a window that is the same size as the plot’s spatial extents and then zoom in from there by making the window values smaller and smaller. The window values are also of the form x0 x1 y0 y1. To change the 2D view, type new values into the Viewport and Window text fields on the View Window’s 2D view tab (Figure 1.260).

Some databases yield plots that are so long and skinny that they leave most of the visualization window blank when VisIt displays them. A common example is equation of state data, which often has at least 1 exponential dimension. VisIt provides Fullframe mode to stretch long, skinny plots so they fill more of the visualization window so it is easier to see them. It is worth noting that Fullframe mode does not preserve a 1:1 aspect ratio for the displayed plots because they are stretched in each dimension so they fit better in the visualization window. To activate full frame mode, click on the Auto or On radio buttons to the left of the Full Frame label. When full frame mode is set to Auto, VisIt determines the aspect ratio of the X and Y dimensions for the plots being visualized and automatically scales the plots to fit the window when extents for one of the dimensions are much larger than the extents of the other dimension.
Fig. 1.259: The curve view options
Fig. 1.260: The 2D view options
Fig. 1.261: The effect of full frame mode on an extremely skinny plot

Just like the with the curve view, the x and y values may be log scaled independently. To log scale the x values, check the Log radio box to the right of the X Scale label. To log scale the y values, check the Log radio box to the right of the Y Scale label.

Setting the 3D view

Setting the 3D view using controls in the View Window’s 3D view tab (see Figure 1.262) demands an understanding of 3D views. A 3D view is essentially a location in space (view normal) looking at another location in space (focus) with a cone of vision (view angle). There are also clipping planes that lie along the view normal that clip the near and far objects from the view. Figure 1.263 depicts the various components of a 3D view.

To set the 3D view, first decide on where you want to look from. Type a vector value into the View normal text field. Next, type the vector valued location of what you want to look at into the Focus text field. The Up axis vector is simply a vector that determines which way is up. A good default value for the up axis is 0 1 0. VisIt will often calculate a better value to use for the up axis so it is not too important to figure out the right value. The View Angle determines how wide the field of view is. The view angle is specified in degrees and a value around 30 is usually sufficient. Near clipping and Far clipping are values along the view normal that determine where the near and far clipping planes are to be placed. It is not easy to know that good values for these are so you will have to experiment. Parallel scale acts as a zoom factor and larger values zoom the camera towards the focus. The Perspective check box applies to 3D visualizations and it causes a more realistic view to be used where objects that are farther away are drawn smaller than closer objects of the same size. VisIt uses a perspective view for 3D visualizations by default.

VisIt supports stereo rendering, during which VisIt draws the image in the visualization window twice with the camera eye positioned in slightly different locations to mimic the differences in images seen by your left eye and your right eye. With the right stereo goggles, the image that you see appears to hover in 3D space within your monitor since the effect of the stereo image adds much more depth to the visualization. You can set the angle that VisIt uses to separate the cameras used to draw the images by typing a new angle into the Eye angle text field or by using the Eye angle slider.

The Align to axis menu provides a convenient way to get side, top, and bottom views of your 3D data. It provides six options corresponding to the six axis aligned directions and sets both the View normal and the Up vector.
Fig. 1.262: The 3D view options
Fig. 1.263: The 3D perspective view volume

**Setting the axis array view**

Visualization windows that contain Parallel Coordinate plots use a special type of view known as an axis array view. An axis array view consists of: viewport, domain, and range. The viewport is the area of the visualization window that will be occupied by the plots and is specified using X and Y values in the range [0,1]. The point (0,0) corresponds to the lower-left corner of the visualization window while the point (1,1) corresponds to the visualization window’s upper-right corner. To change the viewport, type new numbers into the **Viewport** text field on the **Curve view** tab of the **View Window** (Figure 1.264). The minimum and maximum X values should come first, followed by the minimum and maximum Y values.

The **Domain** and **Range** settings are not very intuitive and we will give a short description followed by some examples. The domain controls the position and spacing of the parallel axes. The larger the value the more tightly they are spaced or the more axes that will fit in the view. For example, a domain of 0. to 2. would have room for exactly three coordinate axes, with the first one at the extreme left edge of the viewport and the third one at the extreme right edge of the viewport. Changing the domain to 1. to 3. would shift the second axis to the extreme left edge of the viewport and move the third axis to the center of the viewport. If there were only three axes, then the right half of the viewport would be empty. The range controls the height of the coordinate axes. The larger the value, the shorter the axes. For example, the default range of 0. to 1. results in the axes filling the height of the viewport. A range of 0. to 2. results in the axes filling the bottom half of the viewport. You can play with the controls to get a better understanding of the domain and range settings.

**Advanced view features**

The **View Window’s Advanced** tab, shown in **Figure 1.265**, contains advanced features that are not needed by all users.
Fig. 1.264: The axis array view options
Fig. 1.265: The advanced view options
The **View based on** menu is used to specify if the view is set based on the original spatial extents of the plot or the actual current extents which are the plot’s current extents after it has been subsetted in some way. By default, VisIt bases the view on the plot’s original extents which leaves the remaining bits of a plot, after being subsetted, in the same space as the original plot. This makes it easy to see where the remaining pieces of the plot were situated relative to the whole plot but it does not always make best use of the visualization window. To fill up more of the visualization window, you might want to base the view on the actual current extents by selecting **Actual current extents** from the **View based on** menu.

When using more than one visualization window, such as when comparing plots using two different databases side by side, it is often useful for the plots being compared to have the same view. VisIt allows you to lock the views together for the multiple visualization windows so that when you change the view of any window whose view is locked, all other windows with locked views get the new view. To lock the view for a visualization window, click the **Locked view** check box or click on the Toolbar button to lock views.

Normally, VisIt will adjust the view to match the extents of the data. For example, if you are looking at data from a simulation whose extents expand over time, VisIt will automatically adjust the view so that the data fills roughly the same amount of space as the extents expand. Another example is when the extents move from left to right, VisIt will adjust the view so that the extents are always centered in the same portion of the screen. This behavior is not always desired in certain situations. To turn off this behavior and fix the view, no matter how the extents of the data change, click on the **Maintain view limits** check box.

The **Reset view**, **Recenter view**, and **Undo view** can be used to reset the view, recenter the view, and undo the last view change. Resetting the view resets all aspects of the view based on the data extents. Recentering the view resets all aspects of the view except the view orientation based on the data extents. Undoing the view returns the view to the last view setting. The last 10 views are stored so you can undo the view up to 10 times.

The **Locked view** check box, the **Maintain view limits** check box, the **Reset view** button, the **Recenter view** button, and **Undo view** buttons behave differently than the rest of the controls in the view window in that they effects take effect immediately, without having to press the **Apply** button.

The **Copy view from camera** check box and the **Make camera keyframe from view** button are deprecated and will be removed in the next release.

The center of rotation is the point about which plots are rotated when you set the view. You can type a new center of rotation into the **Center** text field and click the **User defined center of rotation** check box if you want to specify your own center of rotation. The center of rotation is, by default, the center of your plots’ bounding box. When you zoom in to look at smaller plot features and then rotate the plot, the far away center of rotation causes the changes to the view to be large. Large view changes when you are zoomed in often make the parts of the plot that you were inspecting go out of the view frustum. If you are zoomed in, you should pick a center of rotation that is close to the surface of the plot that you are inspecting. You can also pick a center of rotation using the **Choose center** from the visualization window’s **Popup** menu.

### Using view commands

The **Commands** text field at the bottom of the **View Window** allows you to enter one or more semi-colon delimited legacy MeshTV commands to change the view. The following list has a description of the supported view commands:

- **pan x y** Pans the 3D view to the left/right or up/down. The x, y arguments, which are floating point fractions of the screen in the range [0,1], determine how much the view is panned in the X and Y dimensions.

- **pan3 x y** Same as pan.

- **pany y** Pans the 3D view left or right. The x argument is a floating point fraction of the screen in the range [0,1].

- **pany y** Pans the 3D view up or down. The y-argument is a floating point fraction of the screen in the range [0,1].

- **ytrans y** Same as pany.

- **rotx x** Rotates the 3D view about the X-axis x degrees.
rx x  Same as rotx.

roty y  Rotates the 3D view about the Y-axis y degrees.

roty z  Rotates the 3D view about the Z-axis z degrees.

rz z  Same as rotz.

zoom val  Scales the 3D zoom factor. If you provide a value of 2.0 for the val argument, the object being viewed will appear twice as large. A value of 0.5 for the val argument will make the object appear only half as large.

zf  Same as zoom.

zoom3  Same as zoom.

vp x0 x1 y0 y1  Sets the window, which is how much space relative to the plot will be visible inside of the viewport, for the 2D view. All arguments are floating point numbers that are in the same range as the plot extents. The x0 and x1 arguments are the minimum and maximum values for the edges of the window in the X dimension. The y0 and y1 arguments are the minimum and maximum values for the edges of the window in the Y dimension.

wp x0 x1 y0 y1  Sets the window, which is how much space relative to the plot will be visible inside of the viewport, for the 2D view. All arguments are floating point numbers that are in the same range as the plot extents. The x0 and x1 arguments are the minimum and maximum values for the edges of the window in the X dimension. The y0 and y1 arguments are the minimum and maximum values for the edges of the window in the Y dimension.

reset  Resets the 2D and 3D views.

recenter  Recenters the 3D view.

undo  Changes back to the previous view.

1.10 Animation

This chapter discusses how to use VisIt to create animations. There are three ways of creating animations using VisIt: flipbooks, keyframing, and scripting. For complex animations with perhaps hundreds or thousands of database time steps, it is often best to use scripting. VisIt provides Python and Java language interfaces that allow you to program animation and save image files that get converted into a movie. The flipbook approach is strictly for static animations in which only the database time step changes. This method allows database behavior over time to be quickly inspected without the added complexity of scripting or keyframing. Keyframed animation can exhibit complex behavior of the view, plot attributes, and database time states over time. This chapter emphasizes the flipbook and keyframe approaches and explains how to create animations both ways.

1.10.1 Animation basics

Animation is used mainly for looking at how scientific databases evolve over time. Databases usually consist of many discrete time steps that contain the state of a simulation at a specific instant in time. Creating visualizations using just one time step from the database does not reveal time-varying behavior. To be most effective, visualizations must be created for all time steps in the database.

The .visit file

Since scientific databases usually consist of dozens to thousands of time states. Those time states can reside in any number of actual files. Some database file formats support multiple time states in a single file while other formats require each time state to be located in its own file. When all time states are in their own file, it is important for VisIt to know which files comprise the database. VisIt attempts to use automatic file grouping to determine which files are in a database but sometimes it is better if you provide the actual list of files in a database when you want to generate an
animation using VisIt. You can create a .visit file that contains a list of the files in the database. By having a list of files that make up the database, VisIt does not have to guess database membership based on file naming conventions. While this may appear to be inconvenient, it removes the possibility that VisIt will include a file that is not in the database. It also frees VisIt from having to know about dozens of ad hoc file naming conventions. Having a .visit file also allows VisIt to make certain optimizations when generating a visualization.

VisIt provides a **File grouping** combo box in the **File open** window (see Figure 1.266) to assist in grouping related time-varying files into a virtual database. A virtual database accomplishes the same function as a .visit file except that no extra file needs to be created. Selecting **On** or **Smart** will group files into a virtual database. The **On** setting applies file matching rules to group files with similar prefixes into a virtual database. VisIt will attempt to generate a pattern from a filename so sequences of numbers can be abstracted out. Multiple files that match the same pattern are added to the same virtual database. The **Smart** setting applies the same logic as well as some extra rules that permit additional file grouping. For instance, certain file extensions that include numbers such as .hdf5 are excluded from the pattern generation so the number in the file extension does not prevent useful file groupings.

![File open window](image)

**Fig. 1.266:** File open window
Fig. 1.267: Animation controls
Flipbook animation

All that is needed to create a flipbook animation is a time-varying database. To view a flipbook animation, open a time-varying database, create plots as usual, and click the Play button in the GUI shown in Figure 1.267 highlighted in red or in the visualization window’s Animation Toolbar. A flipbook animation repeatedly cycles through all of the time states in the database displaying the plots for the current time state in the visualization window. The result is an animation that allows you to see the database evolve over time. The VCR buttons, shown in Figure 1.267, allow you to control how a flipbook animation plays. The animation controls are also used for controlling keyframe animations. Clicking the Play button causes VisIt to advance the database timestep until the Stop button is clicked. As the plots are generated for each database time state, the animation proceeds only as fast as the compute engine can generate plots. As described in the Animation Window section, you have the option of caching the geometry for each time state so animations will play smoothly according to the animation playback speed once the plots for each database time state have been generated.

Setting the time state

There are several ways that you can set the time state for an animation. You can use the VCR controls to play animations or step through them one state at a time. You can also use the Time slider to access a specific animation time state. To set the animation time state using the Time slider, click on the time slider and drag horizontally to a new time state. The time state to which you drag it will be displayed in the Cycle/Time text field as you drag the time slider so you will know when to let go of the Time slider. Once you release the mouse button at a new time state, VisIt will calculate the visualized plots using the data at the specified time state.

If you prefer more precise control over the time state, you can type a cycle or time into the Cycle/Time text field to make VisIt jump to the closest cycle or time for the active database. You can also highlight a new time state for the active database in the Selected files list and then click the Replace button to make VisIt change the time state for the visualization.

Animation Window

You can open the Animation Window, shown in Figure 1.268, by clicking on the Animation... option from the Controls menu. The Animation Window contains controls that allow you to turn off pipeline caching and adjust the animation playback mode and speed.

Animation playback speed

The animation playback speed is used when playing flipbook or keyframe animations. The playback speed determines how fast VisIt cycles through the database states that make up the animation. Rather than using states per second as a measurement for the playback speed, VisIt uses a simple scale of slower to faster. To set the animation playback speed, use the Animation speed slider. Moving the slider to the left and slower setting slows down animations so they change time states once every few seconds. Moving the slider to the right and faster setting will make VisIt play the animation as fast as the host graphics hardware allows.

Pipeline caching

When pipeline caching is enabled, VisIt tries to retain all of the geometric primitives that are used to draw a plot. This greatly speeds up animations once the geometry for all time states is cached. The downside to pipeline caching is that it can consume large amounts of memory. Pipeline caching is enabled by default, but sometimes it makes sense to turn it off. The deciding factors are the size of the database, the number of animation frames, and the number of plots in each animation frame. Try leaving pipeline caching enabled until you notice performance degradation. To turn off pipeline caching, uncheck the Pipeline caching check box in the Animation Window.
Fig. 1.268: Animation window
Animation playback mode

The animation playback mode determines how VisIt gets to the next time state after playing until the end of the animation. There are three animation playback modes: looping, play once, and swing. VisIt loops animations by default so once the end of the animation is reached, it starts playing from the beginning. When the animation mode is set to play once, VisIt plays the animation through until the end and then stops playing the animation. When VisIt reaches the end of the animation in swing mode, the animation starts playing in reverse until it gets to the start, at which point, it starts playing forward again. To set the animation mode, click on one of the Looping, Play once, and Swing radio buttons in the Animation Window.

1.10.2 Keyframing

Keyframing is an advanced form of animation that allows you create animations where certain animation attributes such as view or plot attributes can change as the animation progresses. You can design an entire complex animation upfront by specifying a number of animation frames to be created and then you can tell VisIt which plots exist over the animation frames and how their time states map to the frames. You can also specify the plot attributes so they remain fixed over time or you can make individual plot and operator attributes evolve over time. With keyframing, you can make a plot fade out as the animation progresses, you can make a slice plane move, you can make the view slowly change, etc. Keyframe animations allow for quite complex animation behavior.

Keyframing Window

Keyframe animations are designed using VisIt’s Keyframing Window (see Figure 1.269), which you can open by selecting the Keyframing option from the Controls menu. The window is dominated by the Keyframe area, which consists of many vertical lines that correspond to each frame in the animation and horizontal lines, or Keyframe lines, that correspond to the state attributes that are being keyframed. The horizontal lines are the most important because they allow you to move and delete keyframes and set the plot range, which is the set of animation frames over which the plot is defined.

Keyframing mode

To create a keyframe animation, you must first open the Keyframing Window and check the Keyframing enabled check box. When VisIt is in keyframing mode, a keyframe is created for the active animation state each time you set plot or operator attributes and time is set using the Animation time slider. The Animation time slider is a special time slider that is made active when you enter keyframing mode and the animation frame can only be set using it. Changing time using any other time slider results in a new database state keyframe instead of changing the animation frame.

If you have created plots before entering keyframing mode, VisIt converts them into plots that can be keyframed when you enter keyframing mode. When you leave keyframing mode, extra keyframing attributes associated with plots are deleted, the animation containing the plots reverts to a flipbook animation, and the Animation time slider is no longer accessible.

Setting the number of frames

When you go into keyframing mode for the first time, having never set a number of keyframes, VisIt will use the number of states in the active database for the number of frames in the new keyframe animation. The number of frames in the keyframe animation will vary with the length of the database with the most time states unless you manually specify a number of animation frames, which you can do by entering a new number of frames into the Keyframing Window’s Number of frames text field. Once you enter a number of frames, the number of frames will not change unless you change it.
Fig. 1.269: Keyframing Window
Adding a keyframe

To add a keyframe, you must first have created some plots and put VisIt into keyframing mode by clicking the **Keyframing enabled** check box in the **Keyframing Window**. After you have plots and VisIt is in keyframing mode, you can add a keyframe by opening a plot's attribute window, changing settings, and clicking its **Apply** button. To set a keyframe for a later frame in the animation, move the **Keyframe time** slider, which is located under the **Keyframe area** (see Figure 1.270), to a later time and change the plot attributes again. Each time you add a keyframe to the animation, a small black diamond, called a **Keyframe indicator**, will appear along the **Keyframe line** for the plot. When you play through the animation using any of VisIt's animation controls, the plot attributes are calculated for each animation frame and they are used to influence how the plots look when they appear in the **Viewer** window.

Adding a database state keyframe

Each plot that exists at a particular animation frame must use a specific database state so the correct data will be plotted. When VisIt is in keyframing mode, the database state can also be keyframed so you can control the database state used for a plot at any given animation frame. The ability to set an arbitrary database state keyframe for a plot allows you to control the flow of time in novel ways. You can, for example, slow down time, stop time, or even make time flow backwards for a little while.

There are two ways to set database state keyframes in VisIt. The first way is to move the **Keyframe time** slider to the desired animation frame, enter a new number into the text field next to the **Keyframe Window's Add state keyframe** button, and the click the **Add state keyframe** button. As an alternative, you can use the **Main Window's Time slider** to create a database state keyframe, provided the active time slider is not the Animation time slider. To set a database state keyframe using the **Time slider**, select a new database time slider from the **Active time slider combo box** and then change time states using the **Time slider**. Instead of changing the active state for the plots that use the specified database, VisIt uses the information to create a new database state keyframe for the active animation frame.

Adding a view keyframe

In addition to being able to add keyframes for plot attributes, operator attributes, and database states, you can also set view keyframes so you can create sophisticated flybys of your data. To create a view keyframe, you must interactively change the view in the **Viewer** window using the mouse or specify an exact view in the **View Window**. Once the view is where you want it for the active animation frame, open the **View Window** and click the **Make camera keyframe from view** button on the **Advanced** tab in order to make a view keyframe. Once the view keyframe has been added, a keyframe indicator will be drawn in the **Keyframing Window**.

VisIt will not use view keyframes by default when you are in keyframing mode because it can be disruptive for VisIt to set the view while you are still adding view keyframes. Once you are satisfied with your view keyframes, click the
Copy view from camera button on the Advanced tab in the View Window in order to allow VisIt to set the view using the view keyframes when you change animation frames.

Deleting a keyframe

To delete a keyframe, move the mouse over a Keyframe indicator and right click on it with the mouse once the indicator becomes highlighted.

Moving a keyframe

To move a keyframe, move the mouse over a Keyframe indicator, click the left mouse button and drag the Keyframe indicator left or right to a different animation frame. If at any point you drag the Keyframe indicator outside of the green area, which is the plot time range, and release the mouse button, moving the keyframe is cancelled and the Keyframe indicator returns to its former animation frame.

Changing the plot time range

The plot time range determines when a plot appears or disappears in a keyframed animation. Since VisIt allows plots to exist over a subset of the animation frames, you can set a plot’s plot range in the Keyframe area to make a plot appear later in an animation or be removed before the animation reaches the last frame. You may find it useful to set the plot range if you have increased the number of animation frames but found that the plot range did not expand to fill the new frames. To change the plot time range, you left-click on the beginning or ending edges of the Plot time range (the green band on the Keyframe line) in the Keyframe area and drag it to a new animation frame.

1.10.3 Scripting

Scripting is an alternate method of producing animations that can have the simplicity of flipbook animations with the flexibility of keyframing. Scripting animations is more difficult than other methods because you have to script each event by writing a Python or Java program to control VisIt’s viewer. One clear strength of this method is that it is very reproducible and can be used to generate animation frames in a batch computing environment. For in-depth information about writing Python scripts for Visit, consult the Visit Python Interface manual.

Command Window

It is possible for VisIt’s GUI and Python Interface to share the same viewer component at runtime. When you invoke visit at the command line, VisIt’s GUI is launched. When you invoke visit -cli at the command line, VisIt’s CLI (Python interface) is launched. If you want to use both components simultaneously then you can use VisIt’s Command Window. The Command Window can be opened by clicking on the Command menu option from the Controls menu. The Command Window consists of a set of eight tabs in which you can type Python scripts. When you type a Python script into one of the tabs, you can then click the tab’s Execute button to make VisIt try and interpret your Python code. If VisIt detects that it has no Python interpreting service available, it will launch the CLI (connected to the same viewer component) and then tell the CLI to execute your Python code. Note that the Command Window is just for editing Python scripts. Any output that results from the Python code’s execution will be displayed in the CLI program window (see Figure 1.271).

Saving the Command Window’s Python scripts

The Command Window is meant to be a sandbox for experimenting with small Python scripts that help you visualize your data. You will often hit upon small scripts that can be used over and over. The scripts in each of the eight tabs in
Fig. 1.271: Command Window
the **Command Window** can be saved for future VisIt sessions if you save your settings. Once you save your settings, any Python scripts that are present in the **Command Window** are preserved for future use.

**Clearing a Python script from a tab**

If a Python script in one of the **Command Window**’s tabs is no longer useful then you can click that tab’s **Clear** button to clear out the contents of the tab so you can begin creating a new script in that tab. If you want VisIt to permanently delete the script from the tab then you must save your settings after clicking the **Clear** button.

**Using the GUI and CLI to design a script**

Writing a Python script that performs visualization from scratch can be difficult. The process of setting up a complex visualization can be simplified by using both the GUI and the CLI at the same time. For example, you can use VisIt’s GUI to set up the plots that you initially want to visualize and then you can save out a session file that captures that setup. Next, you can open a text editor and create a new Python script. The first line of your Python script can use VisIt’s **RestoreSession** command to restore the session file that you set up with the GUI from within the Python scripting environment. For more information on functions and objects available in VisIt’s Python interface, see the **VisIt Python Interface** manual. After using the **RestoreSession** function to set VisIt situated with all of the right plots, you can proceed with more advanced Python scripting to alter the view or move slice planes, etc. Once you have completed your Python script in a text editor, you can pasted it into the **Command Window** to test it or you can pass it along to VisIt’s command line movie tools to make a movie.

### 1.10.4 Movie tools

VisIt provides a command line utility based on VisIt’s Command Line Interface that is called **visit -movie**. The **visit -movie** movie generation utility is installed with all versions of VisIt and can be used to generate movies using session files or Python scripts as input. If you want to design movies based on visualizations that you have created while using VisIt’s GUI then you might also want to read about the **Save movie wizard**. If the **visit** command is in your path then typing **visit -movie** at the command prompt, regardless of the platform that you are using, will launch the **visit -movie** utility. The following list provides **visit -movie** command line arguments:

- **-format fmt** The format option allows you to set the output format for your movie. The supported values for fmt are:
  - mpeg : MPEG 2 movie.
  - qt : QuickTime movie.
  - sm : Streaming movie format.
  - png : Save raw movie frames as individual PNG files.
  - ppm : Save raw movie frames as individual PPM files.
  - tiff : Save raw movie frames as individual TIFF files.
  - jpeg : Save raw movie frames as individual JPEG files.
  - bmp : Save raw movie frames as individual BMP (Windows Bitmap) files.
  - rgb : Save raw movie frames as individual RGB (SGI format) files.

- **-geometry size** The geometry option allows you to set the movie resolution. The size argument is of the form WxH where W is the width of the image and H is the height of the image. For example, if you want an image that is 1024 pixels wide and 768 pixels tall, you would provide: -geometry 1024x768.
-**sessionfile name**  The sessionfile option lets you pick the name of the VisIt session to use as input for your movie. The VisIt session is a file that describes the movie that you want to make and it is created when you save your session from within VisIt's GUI after you set up your plots how you want them.

-**scriptfile name**  The scriptfile option lets you pick the name of a VisIt Python script to use as input for your movie.

-**framestep name**  The number of frames to advance when going to the next frame.

-**start frame**  The frame at which to start.

-**end frame**  The frame at which to end.

-**fps number**  Sets the frames per second at which the movie should be played.

-**output**  The output option lets you set the name of your movie.

The `visit -movie` utility always supports creation of series of image files but it does not always support creation of movie formats such as QuickTime, or Streaming movie. Support for movie formats varies based on the platform. QuickTime and Streaming movie formats are currently limited to computers running IRIX and the appropriate movie conversion tools (`makemovie`, `img2sm`) must be in your path or VisIt will create a series of image files instead of a single movie file. You can always use `visit -movie` to generate the individual movie frames and then use your favorite movie generation software to convert the frames into a single movie file.

If you browse the Windows file system and come across a VisIt session file, which ends with a `.session` extension, you can right click on the file and choose from several movie generation options. The movie generation options make one-click movie generation possible so you don’t have to master the arguments for `visit -movie` like you do on other platforms. After selecting a movie generation option for a VisIt session file, Windows runs `visit -movie` implicitly with the right arguments and saves out the movie frames to the same directory that contains the session file, and will have the same name as the session file. The movie generation options in a session file’s context menu are shown in Figure 1.272.

**Fig. 1.272: Movie generation options for session files on Windows platform**

If you browse the Windows file system and come across a VisIt session file, which ends with a `.session` extension, you can right click on the file and choose from several movie generation options. The movie generation options make one-click movie generation possible so you don’t have to master the arguments for `visit -movie` like you do on other platforms. After selecting a movie generation option for a VisIt session file, Windows runs `visit -movie` implicitly with the right arguments and saves out the movie frames to the same directory that contains the session file, and will have the same name as the session file. The movie generation options in a session file’s context menu are shown in Figure 1.272.

### 1.11 Interactive Tools

An interactive tool is an object that can be added to a visualization window to set attributes for certain plots and operators such as the Parallel Coordinates plot or Slice operator. You can turn interactive tools on and off by clicking...
on the tool icons in a visualization window’s **Toolbar** or **Popup menu** (see Figure 1.273). Note that some tools prefer to operate in visualization windows that contain plots of a certain dimension so some tools are not always available.

![Tools menu](image)

**Fig. 1.273: Tools menu**

Once you enable a tool, it appears in the visualization window. Tools have one or more small red rectangles called *hot points* that cause the tool to perform an action when you click or drag the hot point with the mouse. When you use the mouse to manipulate a tool’s hot point, all mouse events are delivered to the tool so it can respond to the mouse interaction. When the mouse is outside of a hot point, the mouse responds as it would if there were no tools activated so you can still rotate and zoom-in on plots while still having tools enabled.

### 1.11.1 Box Tool

The box tool, which is shown in Figure 1.274, allows you to move an axis-aligned box around in 3D space. You can use the box tool with the Box and Clip operators to interactively restrict plots to a certain volume. The box tool is drawn as a box with five hotpoints that allow you to move the box in 3D space or resize it in any or all dimensions.

You can move the box tool around the **Viewer** window by clicking on the origin hotpoint, which has the word “Origin” next to it, and dragging it around the **Viewer** window. When you move the box tool, it moves in a plane that is parallel to the screen. You can move the box tool backward and forward along an axis by holding down the keyboard’s *Shift* key before you click and drag the origin hotpoint. When the box tool moves, red, green, and blue boxes appear to give a point of reference for the box with respect to the X, Y, and Z dimensions (see Figure 1.275).

You can extend one of the box’s faces at a time by clicking on the appropriate hotpoint and moving the mouse up to extend the box or by moving the mouse down to shrink the box in the given dimension. Hotpoints for the box’s back faces are drawn smaller than their front-facing counterparts. When the box is resized in a single dimension, reference planes are drawn in the dimension that is changing so you can see where the edges of the box are in relation to the bounding box for the visible plots. You can also resize all of the dimensions at the same time by clicking on the “Resize XYZ” hotpoint and dragging the mouse in an upward motion to scale the box to a larger size in X,Y, and Z or by dragging the mouse down to shrink the box. When all box dimensions are resized at the same time, the shape of the box remains the same but the scale of the box changes.
Fig. 1.274: Box tool with a plot restricted to the box

Fig. 1.275: Box tool while it is resized or moved
### 1.11.2 Line Tool

It is common to create Curve plots when analyzing a simulation database. Curve plots are created using VisIt’s lineout mechanism where reference lines are drawn in a visualization window and Curve plots are created in another visualization window using the path described by the reference lines. VisIt’s line tool allows reference lines to be moved after they are initially drawn. The line tool allows the user to see a representation of a line in a visualization window and position the line relative to plots that exist in the window.

![Line Tool with a 2D plot](image)

**Fig. 1.276**: Line tool with a 2D plot

The line tool is drawn as a thick line with three hot points positioned along the length of the line. Both of the line tool’s endpoints, as well as its center, have a hotspot. Since the line tool can be used for both 2D and 3D databases, the line tool’s behavior is slightly different for 2D than it is for 3D. Clicking and dragging on either endpoint will move the selected endpoint causing the line to change shape. Another way of moving an endpoint is to hold down the Ctrl key and then click on the point and move the mouse up and down to extend or shorten the line. Clicking and dragging the middle hot point moves the entire line tool.

In 2D, the line endpoints can only be moved in the X-Y plane (Figure 1.276). In 3D, the line endpoints can be moved in any dimension. Since it is more difficult to see how the line is oriented relative to plots in 3D, when the line tool is moved, 3D crosshairs appear. The crosshairs intersect the bounding box and show the position of the line endpoint relative to the plots. Clicking and dragging endpoints will move them in a plane that is perpendicular to the screen. Moving the endpoints, while first pressing and holding down the Shift key, causes the selected endpoint to move back and forth in the dimension that most faces the screen. This allows endpoints to be moved in one dimension at a time.

An example of the line tool in 3D is shown in Figure 1.277.

The line tool can be used to set the attributes for certain VisIt operators such as VisIt’s Lineout operator. If a plot has a Lineout operator applied to it, the line tool is initialized with that operator’s endpoints when it is first enabled. As the line tool is repositioned and reoriented, the line tool’s line endpoints are given to the Lineout operator and and Curve plots that are fed by the Lineout operator are recalculated.

### 1.11.3 Plane Tool

The plane tool allows the user to see a representation of a slice plane in a visualization window and position the plane relative to plots that may exist in the window. The plane tool, shown in Figure 1.278, is represented as a set of 3D axes,
a bounding rectangle, and text which gives the plane equation in origin-normal form. The plane tool provides several hot points positioned along the 3D axes that are used to position and orient the tool. The hot point nearest the origin allows the user to move the plane tool in a plane parallel to the computer screen. The hot point that lies in the middle of the plane’s Z-axis translates the plane tool along its normal vector when the hotpoint is dragged up and down. The hot point on the end of the Z-axis causes the plane tool to rotate freely when the hot point is moved. When the plane tool is facing into the screen, the Z-axis vector turns red to indicate which direction the plane tool is pointing. The other hot points also rotate the plane tool but they restrict the rotation to a single axis of rotation.

You can use the plane tool to set the attributes for certain VisIt plots and operators. The Slice operator, for example, can update its plane equation from the plane tool’s plane equation. If a plot has a Slice operator applied to it, the plane tool is initialized with that operator’s slice plane when it is first enabled. As the plane tool is repositioned and reoriented, the plane tool’s plane equation is given to the operator and the sliced plot is recalculated.

1.11.4 Point Tool

The point tool allows you to position a single point relative to plots that exist in the visualization window. The point tool provides one hot point at the tool’s origin. Clicking on the hot point and moving the mouse moves the point tool’s origin in a plane perpendicular to the screen. Holding down the Shift key before clicking on the hot point moves the point tool’s origin along the plot axis that most faces the user. Holding down the Ctrl key moves the point tool along the plot axis that points up. Figure 1.279 shows the point tool being used to set the origin for the ThreeSlice operator.

1.11.5 Sphere Tool

The sphere tool allows you to position a sphere relative to plots that exist in the visualization window. The sphere tool, shown in Figure 1.280, provides several hot points that are used to position and scale the sphere. The hot point nearest the center of the sphere is the origin hot point and it is used to translate the sphere in a plane parallel to the screen. The other hot points are all used to scale the sphere. To scale the sphere, click on one of the scaling hot points and move the mouse towards the origin hot point to shrink the sphere or move the hot point away from the origin to enlarge the sphere.
Fig. 1.278: Plane tool with sliced plot
Fig. 1.279: Point tool
Fig. 1.280: Sphere tool
You can use the sphere tool to set the attributes for certain VisIt plots and operators. The sphere tool is commonly used to set the attributes for the SphereSlice operator. After applying a SphereSlice operator to a plot, enable the Sphere tool to interactively position the sphere that slices the plot.

### 1.11.6 Axis Restriction Tool

The AxisRestriction tool is used in conjunction with the Parallel Coordinates plot allowing you to modify the axis restrictions used by the plot. The Axis Restriction tool, shown in Figure 1.281, provides triangular hot points that are originally positioned at the tops and bottoms of each axis in the plot. As the hot points are moved up or down the axis, the plot is changed to reflect the new min or max.

![Axis Restriction Tool](image)

Fig. 1.281: Axis Restriction tool
1.12 Multiple Databases and Windows

In this chapter, we discuss how to use VisIt to visualize multiple databases using either a single window or multiple visualization windows that have been locked together. After a general discussion of databases, we move to database correlation, which is used to relate multiple time-varying databases together in some fashion. The use of database correlations will be explained in detail followed by a description of database comparisons, then common useful operations involving multiple visualization windows.

1.12.1 Databases

One main use of a visualization tool such as VisIt is to compare multiple related simulation databases. Simulations are often run over and over with a variety of different settings or physics models and this results in several versions of a simulation database that all describe essentially the same object or phenomenon. Simulations are also often run using different simulation codes and it is important for a visualization tool to compare the results from both simulations for validation purposes. You can use VisIt to open any number of databases at the same time so you can create plots from different simulation databases in the same window or in separate visualization windows that have been locked together.

Active database

VisIt can have any number of databases open simultaneously but there is still an active database that is used to create new plots. VisIt calls this the Active source. Each time you open a database, the newly opened database becomes the active source for any new plots that you decide to create. If you want to create a plot using a database that is open but is not your active source, you must make that database the active source. When a database becomes the active source, its variables are added to the menus for the various plot types. To changing the active source, select a database from the Active source combo box in the Main Window as shown in Figure 1.282.

Multiple time sliders

When your open databases all have only a single time state, the Time slider in the Main Window is disabled. When you have one database that has multiple time states, the Time slider is enabled and can be used exclusively to change time states for the database that has multiple time states; the database does not even have to be the active database. Things get a little more complicated when you have opened more than one time-varying database - especially if you have plots from more than one of them.

When you open a database in VisIt, it becomes the active database. If the database that you open has multiple time states, VisIt creates a new logical time slider for it so you can end up having a separate time slider for every open database with multiple time states. When VisIt has to create a time slider for a newly opened database, it also makes
the new database’s (the active source) be the active time slider. There is only one Time slider control in the Main Window so when there are multiple logical time sliders, VisIt displays an Active time slider combo box (see Figure 1.283) that lets you choose which logical time slider to affect when you change time using the Time slider.

Fig. 1.283: Time slider and related controls

Since VisIt allows each time-varying database to have its own logical time slider, you can create plots from more than one time-varying database in a single visualization window and change time independently for each database. Another benefit of having multiple logical time sliders is that the databases plotted in the visualization windows are free to have different numbers of time states. Suppose you have opened time-varying databases A and B and created plots from both databases in the same visualization window. Assuming you opened database A and then database B, database B will be the active database. If you want to change time states for database A but not for database B, you can select database A from the Active time slider combo box and then change the time state using the Time slider. If you then wanted to change time states for database B, you could select it in the Active time slider combo box and then change the time state using the Time slider. If you wanted to change time states for both A and B at the same time, you have to use database correlations, which are covered next.

1.12.2 Database correlations

A database correlation is a map that relates one or more different time-varying databases so that when accessed with a common time state, the database correlation can tell VisIt which time state to use for any of the databases in the database correlation. VisIt supports multiple logical time sliders, so time states can be changed independently for different time-varying databases in the same window. No time slider for any database can have any effect on another database. Sometimes when comparing two different, but related, time-varying databases, it is useful to make plots of both databases and see how they behave over time. Since changing time for each database independently would be tedious, VisIt provides database correlations to simplify visualizing multiple time-varying databases.

Database correlations and time sliders

When you open a database for the first time, VisIt creates a trivial database correlation for that single database and creates a new logical time slider for it. Each database correlation has its own logical time slider. Figure 1.285 shows a database correlation as the active time slider.
Fig. 1.284: Active time slider and time slider controls

Fig. 1.285: Database correlation as the active time slider

1.12. Multiple Databases and Windows
Suppose you have plots from time-varying database A and database B in the same visualization window. You can use the logical time slider for database A to change database A’s time state and you can use the logical time slider for database B to change database B’s time state. If you want to change the time state for both databases at the same time using a single logical time slider, you can create a database correlation involving database A and database B and then change time states using the database correlation’s logical time slider. When you change time states using a database correlation’s time slider, the time state used in each plot is calculated by using the database correlation’s time slider’s time state to look up the plot’s expected time state in the database correlation. Thus changing time states using a database correlation also updates the logical time slider for each database involved in the database correlation.

Types of database correlations

A database correlation is a map that relates one or more databases. When there is more than one database involved in a database correlation, the time states from each database are related using a correlation method. Database correlations currently have 4 supported correlation methods: padded index, stretched index, time, and cycle. This section describes each of the correlation methods and when you might want to use each method.

For illustration purposes, the examples describing each correlation method use two databases, though database correlations can have any number of databases. The examples refer to the databases as: database A and database B. Both databases consist of a rectilinear grid with a material variable. The material variable is used to identify the database using a large letter A or B and also to visually indicate progress through the databases’ numbers of time states by sweeping out a red material like a clock in reverse. At the first time state, there is no red material but as time progresses, the red material increases and finally totally replaces the material that was blue. Database A has 10 time states and database B has 20 time states. The tables below list the cycles and times for each time state in each database so the time and cycle behavior of database A and database B will make more sense later when time database correlations and cycle database correlations are covered.

<table>
<thead>
<tr>
<th>Table 1.1: Database A</th>
</tr>
</thead>
<tbody>
<tr>
<td>Time state</td>
</tr>
<tr>
<td>Times</td>
</tr>
<tr>
<td>Cycles</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Table 1.2: Database B (part 1)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Time state</td>
</tr>
<tr>
<td>Times</td>
</tr>
<tr>
<td>Cycles</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Table 1.3: Database B (part 2)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Time state</td>
</tr>
<tr>
<td>Times</td>
</tr>
<tr>
<td>Cycles</td>
</tr>
</tbody>
</table>

Padded index database correlation

A padded index database correlation, like any other database correlation, involves multiple input databases where each database potentially has a different number of time states. A padded index database correlation has as many time states as the input database with the largest number of time states. All other input databases that have fewer time states than the longest database have their last time state repeated until they have the same number of time states as the input database with the largest number of time states. Using the example databases A and B, since B has 20 time states and A only has 10 time states, database A will have its last time state repeated 10 times to make up the difference in time states between A and B. Note how database A’s last time state is repeated in Figure 1.286.
A stretched index database correlation, like any other database correlation, involves multiple input databases where each database potentially has a different number of time states. Like a padded index database correlation, a stretched index database correlation also has as many time states as the input database with the largest number of time states. The difference between the two correlation methods is in how the input databases are mapped to a larger number of time states. The padded index database correlation method simply repeated the last frame of the input databases that needed more time states to be made even with the length of the database correlation. Stretched index database correlations on the other hand do not repeat only the last frame; they repeat frames throughout the middle time states until shorter input databases have the same number of time states as the database correlation. The effect of repeating time states throughout the middle is to evenly spread out the time states over a larger number of time states.

Stretched index database correlations are useful for comparing related simulation databases where one simulation wrote out data at 2x, 3x, 4x, ... the frequency of another simulation. Stretched index database correlations repeat the data for smaller databases, which makes it easier to compare the databases. Figure 1.287 shows example databases A and B related using a stretched index database correlation. Note how the plots for both databases, even though the databases contain a different number of time states, remain roughly in sync.

A time index database correlation, like any other database correlation, involves multiple input databases where each database potentially has a different number of time states. The number of time states in a time database correlation is not directly related to the number of time states in each input database. The number of time states in the database correlation are instead determined by counting the number of unique time values for every time state in every input database. The times from each input database are arranged on a number line and each unique time value is counted as one time state. Once the time values have been arranged on the number line and counted, VisIt calculates a list of time state indices for each database that identify the right time state to use for each database with respect to the time database correlation’s time state. The first time state for each database is always the first time state index stored for a database. The first time state is used until the time exceeds the first time on the number line, and so on.

Time database correlations are useful in many of the same situations as stretched index database correlations since
they are both used to align different databases in time. Unlike a stretched index database correlation, the time database correlation does a better job of aligning unrelated databases in actual simulation time rather than just spreading out the time states until each input database has an equal number. Use a time database correlation when you are correlating two or more databases that were generated with different dump frequencies or databases that were generated by totally different simulation codes. Figure 1.288 shows the behavior of databases A and B when using a time database correlation.

Cycle database correlation

Cycle database correlations operate in exactly the same way as time database correlations except that they correlate using the cycles from each input database instead of using times. Figure 1.288 shows the behavior of databases A and B when using a cycle database correlation.

Managing database correlations

If you want to create a new database correlation or edit properties related to database correlations, you can use the Database Correlation Window. You can open the Database Correlation Window, shown in Figure 1.290, by clicking on the Database correlations option in the Main Window’s Controls menu. The Database Correlation Window contains the list of database correlations, along with controls that allow you to create new database correlations, edit existing database correlations, delete database correlations, or set global settings that tell VisIt when to automatically create database correlations.

Creating a new database correlation

If you want to create a new database correlation to relate time-varying databases that you have opened, you can do so by opening the Database Correlation Window. The Database Correlation Window contains a list of trivial database correlations for the time-varying databases that you have opened. You can create a new, database correlation by clicking on the New button to the left of the list of database correlations. Clicking the New button opens a Database Correlation Properties Window (Figure 1.291) that you can use to edit properties for the database correlation.

New database correlations are automatically named when you first create them but you can change the name of the database correlation to something more memorable by entering a new name into the Name text field. Once you have entered a name, you should set the correlation method that the database correlation will use to relate the time states from all of the input databases. The available choices, shown in Figure 1.292, are: padded index, stretched index, time, and cycle.
1.12. Multiple Databases and Windows

Fig. 1.290: Database Correlation Window

Fig. 1.291: Database Correlation Properties Window
Once you have chosen a correlation method, it is time to choose the input databases for the correlation. The input databases, or sources as they are sometimes called in VisIt, are listed in the **Sources** list (see Figure 1.293). The **Sources** list only contains the databases that you have opened so far. If you do not see a database that you would like to have in the database correlation, you can either click the **Cancel** button to cancel creating the new database correlation or you can continue creating the database correlation and then add the other database to the correlation later after you have opened it. To add databases to the new database correlation, click on the them in the **Sources** list to highlight then and then click on the **Right arrow** button to move the highlighted databases into the database correlation’s **Correlated sources** list. If you want to remove a database from the **Correlated sources** list, highlight the database in the **Correlated sources** list and then click the **Left arrow** button to move it back to the **Sources** list. Once you are satisfied with the new database correlation, click the **Create database correlation** button to create a new database correlation.

When you create a new database correlation, VisIt also creates a new time slider for the new database correlation. The database correlation’s active time state is initially set to the first time state, which might not match the time state of
individual plots in the vis window. Once you change time states using the Time slider, the plots in the vis window will be updated using the correct time state with respect to the correlation’s active time state. As always, if you want to update the time state for only one database, you can select a different time slider using the Active time slider combo box and then change time states using the Time slider. Any time state changes made to an individual database that is also an input database for a database correlation has no effect on the database correlations that involve the changed database. Time state changes for a database correlation can only happen if you have selected the database correlation as your active time slider.

**Altering an existing database correlation**

![Fig. 1.294: Altering a database correlation](image)

Once a database correlation has been created, you can alter it at any time by highlighting it in the Correlation list in the Database Correlation Window and clicking the Edit button to the left of the Correlation list. Clicking the Edit button opens the Database Correlation Properties Window and allows you to change the correlation method and the input databases. Once the desired changes are made, clicking the Alter database correlation button will make the specified database correlation use the new options and all plots in all vis windows that are subject to the changed database correlation will update to the new time states prescribed by the altered database correlation.

Using the Database Correlation Properties Window explicitly alters a database correlation. Reopening a file or refreshing the file list can implicitly alter a database correlation if after reopening the affected databases, there are different numbers of time states in the databases. When reopened databases that are input databases to database correlations have a new number of time states, VisIt recalculates the indices used to access the input databases via the time slider and updates any plots that were affected. In addition to the time state indices changing, the number of time states in the database correlation and its time slider can also change.

**Deleting a database correlation**

Database correlations are automatically deleted when you close a database that you are not using anymore provided that the closed database is not an input database to any database correlation except for that database’s trivial database correlation. You can delete non-trivial database correlations that you have created by highlighting a database correlation in the Correlation list in the Database Correlation Window and clicking the Delete button to the left of the Correlation list. When you delete a database correlation, the new active time slider will be set to the active database’s
time slider if the active database has more than one time state. Otherwise, the new active time slider, if any, will be set to the time slider for the first source that has more than one time state.

**Automatic database correlation**

VisIt can automatically create database correlations when they are needed if you enable certain global settings to control the creation of database correlations. By default, VisIt will prompt you when it wants to create a database correlation. VisIt can automatically create a database correlation when you add a plot of a multiple time-varying database to a vis window that already contains a plot from a different time-varying database. VisIt first looks for the most suitable existing database correlation and if the one it picks must be modified to accommodate a new input database or if an entirely new database correlation must be created, VisIt will prompt you using a **Correlation question** dialog (Figure 1.295). If you prevent VisIt from creating a database correlation or altering the most suitable correlation, you will no longer be prompted to create a database correlation for the list of databases listed in the **Correlation question** dialog.

![Correlation question dialog](image)

Fig. 1.295: Correlation question dialog

By default, VisIt will only attempt to create a database correlation for you if the new plot’s database has the same number of time states as the existing plot. You can change when VisIt creates a database correlation for you by selecting a different option from the **When to create correlation** combo box in the **Database Correlation Window**. The available options are: **Always**, **Never**, and **Same number of states**. You can change the default correlation method by selecting a new option from the **Default correlation method** combo box. Finally, you can prevent VisIt from prompting you when it needs to create a database correlation if you turn off the **Prompt before creating new correlation** check box.

### 1.12.3 Database comparison

Comparing the results of multiple related simulation databases is one of VisIt’s main uses. Users can plot multiple databases in the same window or adjacent windows, allowing comparison of plots visually. In addition to these visual modes of comparison, VisIt also supports more direct **numerical** comparison through the expression system. Database comparison allows users to plot direct differences between two databases or between different time states in the same database including even in the definition of time derivatives.

Numerical database comparisons use special expressions called **Cross-Mesh Field Evaluation (CMFE)** expressions, `pos_cmfe()` and `conn_cmfe()`, which are capable of mapping a field from one mesh, the *donor*, onto another mesh, the *target*. The name `conn_cmfe()` stands for connectivity-based cross mesh field evaluation (CMFE). It is a specialization of position-based cmfe, `pos_cmfe()`, for cases in which donor and target meshes be topologically congruent (e.g. size, connectivity, decomposition, etc. are identical). More information on CMFE expressions are found in the **Cross-Mesh Field Evaluation (CMFE)** section of the **Expressions** chapter. There is also a helpful wizard, the **Data Level Comparison Wizard**, that simplifies the process of defining comparison expressions. Here, we walk through a few basic examples of using **CMFE** expressions and demonstrate how to use them in comparisons.
Plotting the difference between two databases

The typical case is where two slightly different databases time series have been generated from the same simulation code and the user wishes to work with the difference between the two databases and to have this difference update as the time slider is changed.

\[ \text{<mesh/ireg> - conn_cmfe(</usr/local/visit/data/dbB00.pdb[0]id:mesh/ireg>, <mesh>)} \]

In the above expression, the first argument to \textit{conn_cmfe()} serves as the donor field and the second argument is the target mesh. This expression is a simple difference operation of database A minus database B. Note the special \([0]id\) time specification syntax before the colon but after the file system path in the first argument \textit{conn_cmfe()}. The \(i\) means to interpret the number in brackets, \([0]\) as a time state index. The \(d\) means to further interpret that number as an index difference from the current time slider index. This syntax is described in greater detail in the section describing \textit{pos_cmfe()}.

The assumption made by this expression is that database A is the active database and the user wishes to map database B onto it to subtract it from database A's \textit{mesh/ireg} variable. In this example, database B's \textit{mesh/ireg} field is being mapped onto database A's mesh and their difference is then being taken. Figure 1.296 illustrates the database differencing operation.

![Database B subtracted from database A](image)

Fig. 1.296: Database B subtracted from database A

Plotting values relative to the first time state

Plotting a variable relative to its initial values can be important for understanding how the variable has changed over time. The \textit{conn_cmfe} expression is also used to plot values from one time state relative to the values at the current time state. Consider the following expression:

\[ \text{<mesh/ireg> - conn_cmfe(</usr/local/visit/data/dbA00.pdb[0]i.mesh/ireg>, mesh)} \]

The above expression subtracts the value of \textit{mesh/ireg} at time state zero (in the \([0]i\) without the \(d\) means to always map absolute time index zero from the donor) from the value of \textit{mesh/ireg} at the current time. As the time slider is changed, the values for the active database will change but the part of the expression using \textit{conn_cmfe}, which in this case uses the first database time state, will not change. This allows users to create expressions that compare the current time state to a fixed time state.

Plotting time derivatives

Plotting time derivatives is much like plotting the difference between the current time state and a fixed time state except that instead of being fixed, the second time state being compared is free to move relative to the current time state. To
plot a simple time derivative such as the current time state minus the last time state, create an expression similar to the following expression:

\[
<\text{mesh/ireg}> - \text{conn\_cmfe}(<\text{/usr/local/visit/data/dbA00.pdb[-1]}\text{id:/mesh/ireg}>, \text{mesh})
\]

The important piece of the above expression is its use of \([-1]id\) to specify a time state delta of -1, which means add -1 to the current time state to get the time state whose data will be used in the conn\_cmfe calculation. You could provide different values for the time state in the \[\] operator. Substituting a value of 3, for example, would make the conn\_cmfe expression consider the data for 3 time states beyond the current time state. If you use a time state delta, which always uses the \(d\) suffix, the time state being considered is always relative to the current time state. This means that as you change time states for the active database using the time slider, the plots that use the conn\_cmfe expression will update properly. Figure 1.298 shows an example plot of a time derivative.

1.12.4 Multiple window operations

This section focuses on some of the common techniques for exploring multiple databases when you have multiple visualization windows.
Reflection and Translation

When you visualize multiple related databases, they often occupy the same space in the visualization window since they may have been generated using the same computational mesh but with different physics. When this is the case, you can modify the location of the plots from one of the databases in two immediately obvious ways. First of all, if you simulated the same object and it does not make use of any symmetry then you could use the Transform operator to translate the coordinate system of one of the plots out of the way of the other plot so you can look at the two plots from the different databases side by side in the same visualization window. If your databases make use of symmetry (maybe you only simulated half of the problem) then you can apply the Reflect operator to one of the plots to show them side by side but reflected to show the entire problem. Each method has its merits.

Fig. 1.299: Plots side by side using the Reflect or Transform operator

Copying Windows

If you visualize multiple databases and you want to create identical plots for each database but have them placed in different visualization windows then you can either have VisIt copy windows on first reference or you can clone an existing window and then replace the database used in the new window’s plots with a different database.

If you have already created multiple visualization windows, perhaps as the result of a change to VisIt’s layout, then you can make VisIt copy the attributes of the active window to another visualization window when you switch active windows by enabling Clone window on first reference in the Preferences Window. To open the Preferences Window, choose the Preferences option from the Main Window’s Options menu. This form of window cloning copies the plots, lights, colors, etc from the active window to a pre-existing visualization window when you access it for the first time. If you have already accessed a visualization window but you would still like to copy plots, lights, colors, etc from another visualization window, you can make the destination visualization window be the active window and then copy everything from the source visualization window using the Copy everything menu option in the Main Window’s Windows menu.

If you have no empty visualization window to contain plots for the another database, you can click the Clone option in the Main Window’s Windows menu to create a new visualization window with the same plots and settings as the active window. Once the new window has been created, you could visualize a new database by choosing a new database in the Active source combo box and clicking the Replace button.
**Locking Windows**

When you visualize databases using multiple visualization windows, it is often convenient to keep the time state and view in sync between windows so you can concentrate on comparing plots instead of dealing with the intricacies of setting the view or time state for each visualization window. VisIt’s visualization windows can be locked with respect to time, view, or interactive tools. To lock visualization windows, use the **Popup menu**, **Toolbar**, or the **Lock** options from the **Main Window’s Windows** menu as shown in Figure 1.300.

![Lock menu](image)

![Lock popup menu](image)

**Fig. 1.300:** Mechanisms for locking windows

**Locking views**

If you have created plots from related databases in multiple visualization windows, you can lock the views for the visualization windows together so that as you change the view in one of the visualization windows with a locked view, the other visualization windows with locked views also update to have the same view. There are four types of views in VisIt: curve, 2D, 3D, and AxisArray. If you have 2D plots in a visualization window, the visualization window is considered to be 2D. Locking that 2D visualization window’s view will only update other visualization windows that are also 2D and vice-versa. The same is true for curve, 3D and AxisArray views.

**Locking time**

If you have created plots from related databases in multiple visualization windows, you can lock the visualization windows together in time so that as you change time in one visualization window, it updates in all other visualization windows that are locked in time.

Locking visualization windows together in time may cause VisIt to prompt you to create a new database correlation that involves all of the databases in the visualization windows that are locked in time. VisIt creates a database correlation because the visualization windows must use a common time slider to really be locked in time. If the visualization windows did not use a common time slider then changing time in one visualization window would not cause other visualization windows to update. Once VisIt creates a suitable database correlation for all windows, the active time slider is set to that database correlation in all visualization windows that are locked in time. If you alter a database correlation at this point, it will cause the time state in each locked visualization window to change. Since the same
database correlation is used in all locked visualization windows, changing the time state for the database correlation changes the time state in all of the locked windows. This frees you to examine time-varying database behavior without having to set the time state independently in each visualization window. See *Database correlations* for more information.

### Locking tools

In addition to locking visualization windows together with respect to the view and time, you can also lock their tools. This capability can be useful when exploring data that often requires the use of an operator whose attributes can be set interactively using a tool since the same tool can be used to set the operator attributes for operators in more than one visualization window. See *Interactive Tools* for information on the different tools and how they are used.

Consider the following scenario: you have two related 3D databases and you want to examine the same slice plane for each database and you want each database to be plotted in a separate visualization window. You can set up separate visualization windows and slice the plots from each database independently but locking tools is easier and requires much less setup.

Start off by opening the first 3D database and create the desired plots from it. If you want to maintain a 3D view of the plots, you can clone the visualization window to get a new window with the same plots or you can apply a *Slice operator* to the plots. Apply a Slice operator but make sure the slice is not projected to 2D and also be sure that its *Interactive* check box is turned on. Turn on VisIt’s plane tool and make sure that tools are locked. Clone the visualization window twice and for each of the new visualization windows, make sure that their Slice operator projects to 2D. There should now be four visualization windows if you opted to keep a 3D view of the data. In the last visualization window, replace the database with another related database that you want to compare to the first database.

Now that all of the setup steps are complete, you can save a session file so you can get back to this state when you run VisIt next time. Now, in the window that still has a slice in 3D, use the plane tool to reposition the slice. Both of the 2D visualization windows should also update so they use the new slice plane attributes calculated by the plane tool. The four visualization windows, arranged in a 2x2 window layout are shown in Figure 1.301.

### 1.13 Client Server

Scientific simulations are almost always run on a powerful supercomputer and accessed using desktop workstations. This means that the databases usually reside on remote computers. In the past, the practice was to copy the databases to a visualization server, a powerful computer with very fast computer graphics hardware. With ever increasing database sizes, it no longer makes sense to copy databases from the computer on which they were generated. Instead, it makes more sense to examine the data on the powerful supercomputer and use local graphics hardware to draw the visualization. VisIt can run in a client-server mode that allows this exact use case. The GUI and viewer run locally (client) while the database server and parallel compute engine run on the remote supercomputer (server). Running VisIt in client-server mode is almost as easy as running all components locally. This chapter explains the differences between running locally and remotely and describes how to run VisIt in client-server mode.

#### 1.13.1 Client-Server Mode

When you run VisIt locally, you usually select files and create plots using the open database. Fortunately, the procedure for running VisIt in client-server mode is no different than it is for running in single-computer mode. You begin by launching the *File Open Window* and typing the name of the computer where the files are stored into the *Host* text field.

Once you have told VisIt which host to use when accessing files, VisIt launches the VisIt Component Launcher (VCL) on the remote computer. The VCL is a VisIt component that runs on remote computers and is responsible for launching other VisIt components such as the metadata server (mdserver) and compute engine. (Figure 1.302). Once you are
Fig. 1.301: Multiple visualization windows with locked tools
connected to the remote computer and VCL is running, you won’t have to enter a password again for the remote computer because VCL stays active for the life of your VisIt session and it takes care of launching VisIt components on the remote computer.

![VisIt's Architecture](image)

Fig. 1.302: VisIt’s Architecture

If VCL was able to launch on the remote computer and if it was able to successfully launch the metadata server, the files for the remote computer will be listed in the Files pane of the File Open Window, just as if you were running locally. You then select the file or virtual database and click OK. Now that you have files from the remote computer at your disposal, you can create plots as usual.

**Passwords**

Sometimes when you try to access files on a remote computer, VisIt prompts you for a password by opening a Password Window (Figure 1.303). If you are prompted for a password, type your password into the window and click the Ok button. If the password window appears and you decide to abort the launch of the remote component, you can click the Password Window’s Cancel button to stop the remote component from being launched.

![Password Window](image)

Fig. 1.303: Password Window
If your username for the remote machine is not listed correctly, you can click on the Change username button and a new window will pop up allowing you to enter the proper username for the remote system. (Figure 1.304). Enter the correct username in the text field provided and click Confirm username. Proceed with entering the password in the Password Window.

Fig. 1.304: Change Username Window

VisIt uses ssh for authentication and you can set up ssh so that passwords are not required. This is called passwordless ssh and once it is set up for a computer, VisIt will no longer need to prompt for a password.

Setting Up Password-less SSH

The following instructions describe how to set up ssh to allow password-less authentication among a collection of machines.

On the Local Machine

If you do not already have a ~/.ssh/id_rsa.pub file, generate the key:

```
cd
ssh-keygen -t rsa
```

Accept default values by pressing <Enter>. This will generate two files, ~/.ssh/id_rsa and ~/.ssh/id_rsa.pub. The ~/.ssh/id_rsa.pub file contains your public key in one very long line of text. This information needs to be concatenated to the authorized_keys file on the remote machine, so copy it to a temp file on the remote machine:

```
scp ~/.ssh/id_rsa.pub <your-user-name>@<the.remote.machine>:tmp
```

On the Remote Machine

If you do not already have a ~/.ssh directory, create one with r-w-x permission for the owner only:

```
cd
mkdir .ssh
chmod 700 .ssh
```

If you do not already have a ~/.ssh/authorized_keys file, create an empty one with permission for the owner only:
```
cd ~/.ssh
touch authorized_keys
chmod 600 authorized_keys
```

Concatenate the temporary file you copied into authorized_keys:
```
cd ~/.ssh
cat authorized_keys ~/tmp > authorized_keys
rm ~/tmp
```

### Completing the Process

If you have more remote machines you want to access from the same local machine using *passwordless* ssh, repeat the process starting with copying the `~/.ssh/id_rsa.pub` file from the local machine to the remote, and continuing from there.

You can also repeat the above sections, reversing the local and remote machines, in order to allow *passwordless* ssh to the local machine from the remote machine.

### Environment

It is important to have VisIt in your default search path instead of specifying the absolute path to VisIt when starting it. This is not as important when you run VisIt locally, but VisIt may not run properly in client-server mode if it is not in your default search path on remote machines. If you regularly run VisIt using the network configurations provided for LLNL computers then VisIt will have host profiles, which are sets of information that tell VisIt how to launch its components on a remote computer. The provided host profiles have special options that tell the remote computer where it can expect to find the installed version of VisIt so it is not required to be in your path. If you did not opt to install the provided network configurations or if you are at a site that requires other network configurations then you will probably not have host profiles by default and it will be necessary for you to add VisIt to your path on the remote computer. You can add VisIt to your default search path on Linux systems by editing the initialization file for your command line shell.

### Launch Progress Window

When VisIt launches a compute engine or metadata server, it opens the **Launch Progress Window** when the component cannot be launched in under four seconds. An exception to this rule is that VisIt will always show the **Launch Progress Window** when launching a parallel compute engine or any compute engine on OSX. VisIt’s components frequently launch fast enough that it is not necessary to show the **Launch Progress Window** but you will often see it if you launch compute engines using a batch system.

The **Launch Progress Window** indicates VisIt is waiting to hear back from the component being launched on the remote computer and gives you some indication that VisIt is still alive by animating a set of moving dots representing the connection from the local computer to the remote computer. The icon used for the remote computer will vary depending on whether a serial or parallel VisIt component is being launched. The **Launch Progress Window** for a parallel compute engine is shown in Figure 1.305. The window is visible until the remote compute engine connects back to the viewer or the connection is cancelled. If you get tired of waiting for a remote component to launch, you can cancel it by clicking the **Cancel** button. Once you cancel the launch of a remote component, you can return to your VisIt session. Note that if the remote compute is a parallel compute engine launched via a batch system, the engine
will still run when it is finally scheduled but it will immediately die since VisIt has stopped listening for it. On heavily saturated batch systems, it might be prudent for you to manually remove your compute engine job from the queue.

1.13.2 Host Profiles

When VisIt launches a component on a remote computer, it looks for something called a host profile. A host profile contains information that VisIt uses to launch components on a remote computer. Host profiles allow you to specify information like the remote username, the number of processors, the parallel launch method, etc. You can have multiple launch profiles for any given host, most often a serial profile and one or more parallel profiles.

Host profiles window

VisIt provides a Host profiles window, shown in Figure 1.306, that you can use to manage your host profiles. You can open the Host profiles window by choosing Host profiles from the Options dropdown menu. The Host profiles window is divided into two main areas. The left area contains a list of host profiles currently installed, as well as controls to create, delete, copy and export profiles. The right area contains two vertical tabs: Remote Profiles, used for installing profiles retrieved from a remote location; and Machines, which displays all attributes for the selected host profile. The Remote Profiles tab is useful for obtaining profiles that were not installed with VisIt. Machines has two sections contained in tabs displayed horizontally across the top: Host Settings and Launch Profiles. The Host Settings tab displays information for the selected machine, including the nickname, the full host name, aliases, the username, and connection information. The Launch Profiles tab displays a list of available profiles in the top section, and information for the selected launch profile in tabs on the bottom.

If the Hosts section in the left pane of the Host profiles window has no hosts listed, you have two options for installing already generated profiles. The first is to install one or more of the pre-defined host profiles shipped with VisIt while the second is to install one or more of the pre-defined host profiles from the VisIt repository. See Installing pre-defined host profiles shipped with VisIt and Installing pre-defined host profiles from the VisIt repository.

Click Apply when you are finished making changes in this window, and remember to save your settings (How to Save Settings) before exiting VisIt in order for your changes to be available in future sessions of VisIt.

Creating a new host profile

You click the New Host button to create a new host profile. The host profile will have a default name corresponding to the machine on which you are running VisIt. When you change the Host nickname the new name will be reflected in the Hosts list. See Setting general options, Managing launch profiles and Setting parallel options for more information on the available settings.
Fig. 1.306: Host profiles window
Deleting a host profile

If a host profile is no longer useful, you can click on it in the hosts list to select it and then click the Delete Host button to delete it.

Copying a host profile

To copy a host profile, select the desired source host from the Hosts list, then click the Copy Host button at the bottom of the Hosts list. A new host profile called Copy of XXX (where XXX is the name of the host you chose to copy) will be added to the Hosts list. Select this new host from the list and modify its Host Settings and Launch Profiles appropriately. Once you change the Host nickname the new name will be reflected in the Hosts list.

Exporting a host profile

The Export Host button is useful for saving a host profile installed on your machine to share with someone else. Select the host profile you wish to export, and click the Export Host button. The exported host will be saved to your user VisIt directory (~/.visit/hosts on Linux). The name of the host profile file will start with hosts_, followed by the Host nickname, where letters are all converted to lower case and blanks are converted to underscores, followed by ".xml".

To share the host profile with someone else have them copy the host profile to their VisIt directory. It is recommended that you don’t change the name of the file, but if you do, be aware that VisIt will only recognize it as a host profile if it starts with hosts_ or HOSTS_ and ends with .xml or .XML.

Setting general options

The Host Settings tab allows you to set general attributes for all launch profiles on the host.

Host nickname

Change the Host nickname to the name as you would like it to appear in the Hosts list in the left pane.

Remote host name

The Remote host name should be the fully qualified host name (hostname.domain.net).

Host name aliases

Some clustered systems have one overall host name but also have names for the individual compute nodes that comprise the system. The compute nodes are often named by appending the node number to the host name. For example, if the clustered system is called cluster, you might be logged into node cluster023. When you launch a remote component, VisIt will not find any host profiles if the host name in the host profiles is: cluster.

To ensure that VisIt correctly matches a computer’s node name to one of VisIt’s host profiles, you should include host name aliases in the host profile for a clustered system. Host name aliases typically consist of the host name with different wildcard characters appended to it. Three wildcards are supported. The ? wildcard character lets any one character replace it while the * wildcard character lets any character or group of characters replace it and the # wildcard character lets any numeric digit replace it. Appropriate host aliases for the previous example would be:
Fig. 1.307: Host Settings tab
cluster#.cluster##.cluster###, etc. If you need to enter host name aliases for the host profile, type them into the **Host name aliases** text field.

**Maximum nodes/processors**

If the host has a maximum number of nodes and/or processors that can be allocated, these can be specified by checking the **Maximum nodes** or **Maximum processors** checkboxes and entering a number in the corresponding text fields.

**Path to VisIt installation**

Most of the host profiles that are installed with VisIt specify the expected installation directory for VisIt so VisIt does not have to be in your path on remote computers. Enter the path to VisIt on the host in the **Path to VisIt installation** text field. It should be the full path up-to but not including the **bin** directory.

**Account**

The remote user name is the name of the account that you want to use when you access the remote computer. The remote user name does not have to match your local user name and it is often the case that your desktop user name will not match your remote user name. To change the remote user name, type a new user name into the **Username** text field.

**Sharing a compute job**

Some computers place restrictions on the number of interactive sessions that a single user can have on the computer. To allow VisIt to run on computer systems that enforce these kinds of restrictions, VisIt can optionally force the metadata server and parallel compute engine to share the same job in the batch system. If you want to make the database server and parallel compute engine share the same batch job, you can click the **Share batch job with Metadata Server** check box.

**Determining the host name**

There are many different network naming schemes and each major operating system type seems to have its own variant. While being largely compatible, the network naming schemes sometimes present problems when you attempt to use a computer that has one idea of what its name is with another computer that may use a somewhat different network naming scheme. Since VisIt users are encouraged to use client-server mode because it provides fast local graphics hardware without sacrificing computing power, VisIt must provide a way to reconcile the network naming schemes when 2 different computer types are used.

Workstations often have a host name that was arbitrarily set when the computer was installed and that host name has nothing to do with the computer’s network name, which ultimately resolves to an IP address. This condition is common on computers running MS Windows though other operating systems can also exhibit this behavior. When VisIt launches a component on a remote computer, it passes information that includes the host name of the local computer so the remote component will know how to connect back to the local computer. If the local computer did not supply a valid network name then the remote component will not be able to connect back to the local computer and VisIt will wait for the connection until you click the **Cancel** button in the **Launch progress window**.

By default, VisIt tunnels data connections through SSH. If you don’t want to tunnel, or SSH tunneling is not working you can turn it off by unchecking **Tunnel data connections through SSH** in the **Connection** section. If you want VisIt to rely on the the name obtained from the local computer, click on **Use local machine name**. If you choose the **Parse from SSH_CLIENT environment variable** option then VisIt will not pass a host name for the local computer but
will instead tell the remote computer to inspect the `SSH_CLIENT` environment variable to determine the IP address of the local computer that initiated the connection. This option usually works if you have a local computer that does not accurately report its host name. If you don’t trust the output of any implicit scheme for getting the local computer’s name, you can provide the name of the local computer by typing its name or IP address into the text field next to the **Specify manually** radio button.

**SSH command**

*VisIt* uses ssh for its connections to remote computers. On Windows, *VisIt* packages its own putty-based ssh program: `qtssh.exe`. Regardless of the system, you can override *VisIt*’s SSH by clicking the **SSH command** checkbox and entering the full path to the ssh command you want to use in the text box.

**SSH port**

*VisIt* uses secure shell (ssh) to launch its components on remote computers. Secure shell often uses port 22 but if you are attempting to communicate with a computer that does not use port 22 for ssh then you can specify a port for ssh by clicking the **SSH port** check box and then typing a new port number into the adjacent text field.

In addition to relying on remote computers’ ssh port, *VisIt* listens on its own ports (5600-5605) while launching components. If your desktop computer is running a firewall that blocks ports 5600-5605 then any remote components that you launch will be unable to connect back to the viewer running on your local computer. If you are not able to successfully launch *VisIt* components on remote computers, be sure that you make sure your firewall does not block *VisIt*’s ports. Windows’ default software firewall configurations block *VisIt*’s ports so if you run those software firewall programs, you will have to unblock *VisIt*’s ports if you want to run *VisIt* in client-server mode.

**Gateway**

If access to the compute nodes on your remote cluster is controlled by a gateway computer, then check the **Use gateway** checkbox, and enter the fully qualified name of the gateway computer in the text field. In order for *VisIt* to tunnel SSH connections through the gateway computer, passwordless-ssh needs to be set up from the gateway computer to the hose where you ultimately want to run *VisIt*. See *Setting Up Password-less SSH* for instructions on how to do this.

**Managing launch profiles**

The **Launch Profiles** tab (Figure 1.308) displays the launch profiles available for the selected host, generally a serial profile and one or more parallel profiles. There are controls for creating, deleting and copying launch profiles as well as tabs for setting the launch profile attributes.

**Creating a new launch profile**

Click the **New Profile** button. Give the profile an appropriate name by filling in the **Profile name** text box. The new name will be reflected in the profiles list as soon as it is entered. After filling out all the necessary attributes, click **Apply** in the lower left corner of the window in order to use the new profile immediately. The new profile to be available in future sessions of *VisIt*.

**Deleting a launch profile**

Select the profile to be deleted by clicking on its name in the list, then click the **Delete Profile** button. If you have made a mistake in deleting the profile, you must exit *VisIt* and restart. Saving your settings will make the change
Fig. 1.308: Launch Profiles tab
permanent for future sessions.

**Activating a launch profile**

Only one launch profile can be active for any given host. When VisIt launches a remote component, it looks for the active launch profile for the host where the component is to be launched. The currently active launch profile is the one with the box to the left of the name checked in the list of launch profile names. To activate a different launch profile, select it from the list and click the **Apply** button. The VCL and the metadata server use the active launch profile but VisIt will prompt you for a launch profile to use before launching a compute engine if you have more than one launch profile or your only launch profile has parallel options set for the compute engine.

**Setting the timeout**

The compute engine and metadata server have a timeout mechanism that causes them to exit if no requests have been made of them for a certain period of time so they do not run indefinitely if their connection to VisIt’s viewer is severed. You can set this period of time, or timeout, by typing in a new number of minutes into the **Timeout** text field. You can also increase or decrease the timeout by clicking on the up and down arrows next to the **Timeout** text field.

**Setting the number of threads**

If VisIt is running in threading mode, the number of threads per task can be set by typing in the desired number of threads in the **Number of threads per task** text field, or by utilizing the up and down arrows next to the text field.

**Providing additional command line options**

The **Launch Profiles** tab allows you to provide additional command line options to the compute engine and metadata server through the **Additional arguments** text field. When you provide additional command line options, you should type them, separated by spaces, into the **Additional arguments** text field. Command line options influence how the compute engine and metadata server are executed. For more information on VisIt’s command line options, see VisIt’s command line options, see **Startup Options**.

**Setting parallel options**

The chief purpose of host profiles is to make launching compute engines easier. This is even more the case when host profiles are used to launch parallel compute engines on large computers that often have complex scheduling programs that determine when parallel jobs can be executed. It is easy to forget how to use the scheduling programs on a large computer because each scheduling program requires different arguments. In order to make launching compute engines easy, VisIt hides the details of the scheduling program used to launch parallel compute engines. VisIt instead allows you to set some common parallel options and then figures out how to launch the parallel compute engine on the specified computer using the parallel options specified in the host profile. Furthermore, once you create a host profile that works for a computer, you rarely need to modify it.

To enable parallel options open the **Parallel** tab of the **Launch Profiles** tab, and click the **Launch parallel engine** checkbox.

**Setting the parallel launch method**

The parallel launch method option allows you to specify which launch program should be used to execute the parallel compute engine. This setting depends on the computer where you plan to run the compute engine and how the
Fig. 1.309: Parallel options
computer is configured. Some computers have multiple launch programs depending on which part of the parallel machine you want to use. Figure 1.310 shows some common parallel-launch options that VisIt currently supports.

![Parallel launch options]

**Setting the partition/pool/queue**

Some parallel computers are divided into partitions so that batch processes might be executed on one part of the computer while interactive processes are executed on another part of the computer. You can use launch profiles to tell VisIt which partition to use when launching the compute engine on systems that have multiple partitions. To set the partition, check the **Partition/Pool/Queue** check box and type a partition name into the text field.

**Setting the number of processors**

You can set the number of processors by typing a new number of processors into the **Number of processors** text field in the **Defaults** section. When the number of processors is greater than 1, VisIt will attempt to run the parallel version of the compute engine. You can also click on the up and down arrows next to the text field to increase or decrease the number of processors. If VisIt finds a parallel launch profile, you will have the option of changing the number of processors before the compute engine is actually launched.

**Setting the number of nodes**

The number of nodes refers to the number of compute nodes that you want to reserve for your parallel job. Each compute node typically contains more than one processor (often 2, 4, 16) and the number of nodes required is usually the ceiling of the number of processors divided by the number of processors per node. It is only necessary to set the number of nodes if you want to use fewer processors than the number of processors that exist on a compute node. This option is not available on some computers as it is meant primarily for compute clusters. To set the number of nodes, check the **Number of nodes** check box and type a new number into the text field.
Setting the default bank

Some computers, if they are large enough, have scheduling systems that break up the number of processors into banks, which are usually reserved for particular projects. Users who contribute to a project take processors from their default bank of processors. By default, VisIt uses environment variables to get your default bank when submitting a parallel job to the batch system. If you want to override those settings, you can click the **Bank/Account** check box to turn it on and then type your desired bank into the text field next to the check box.

Setting the parallel time limit

The parallel time limit is the amount of time given to the scheduling program to tell it the maximum amount of time, usually in minutes, that your program will be allowed to run. The parallel time limit is one of the factors that determines when your compute engine will be run and smaller time limits often have a greater likelihood of running before jobs with large time limits. To specify a parallel time limit, click the **Time Limit** check box and enter a number of minutes or hours into the text field. If you want to specify minutes, be sure to append `m` to the number or append an `h` for hours. If you want to specify a timeout of 30 minutes, you would type: `30m`.

Specifying a machine file

When using VisIt with some versions of MPI on some clustered computers, it may be necessary to specify a *machine file*, which is a file containing a list of the compute nodes where the VisIt compute engine should be executed. If you want to specify a machine file when you execute VisIt in parallel on a cluster that requires a machine file, click on the **Machine File** check box and type the name of the machine file that you want to associate with your host profile into the text field.

Specifying constraints

Some machines constrain the processor-to-node ratio. In order to prevent accidentally requesting nodes/processors outside those constraints, they can be entered in table form by clicking the **Constraints** checkbox to enable the controls. Click **Add row** to add a new row to the table, and **Delete row** to remove a row from the table. For each row, enter number of nodes and appropriate associated number of processors in appropriate columns. When the launch engine dialog pops up, users won’t be able to specify node-processor combinations outside of the constraints.

Advanced host profile options

The **Advanced** tab (see Figure 1.312) in the **Launch Profiles tab** lets you specify advanced networking options to ensure that the VisIt components running on the remote computer use resources correctly and can connect back to the viewer running on your local workstation.

Load balancing

Load balancing refers to how well tasks are distributed among computer processors. The goal is to make each computer processor have roughly the same amount of work so they all finish at the same time. VisIt’s compute engine supports two forms of load balancing. The first form is static load balancing where the entire problem is distributed among processors and that distribution of work never changes. The second form of load balancing is dynamic load balancing. In dynamic load balancing, the work is redistributed as needed each time work is done. Idle processors independently ask for work until the entire task is complete. VisIt allows you to specify the form of load balancing that you want to use. You can choose to use static or dynamic load balancing by clicking the **Static** or **Dynamic** radio buttons. There is also a default setting that uses the most appropriate form of load balancing.
Fig. 1.311: Parallel launch constraints

Fig. 1.312: Advanced options tab
Setting up the parallel environment

VisIt is usually executed by a script called: visit, which sets up the environment variables required for VisIt to execute. When the visit script is told to launch a parallel compute engine, it sets up the environment variables as it usually does and then invokes an appropriate parallel launch program that takes care of either spawning the VisIt parallel compute engine processes or scheduling them to run in a batch system. When VisIt is used with some versions of MPI on some clusters, the parallel launch program does not replicate the environment variables that the visit script set up, preventing the VisIt parallel compute engine from running. On clusters where the parallel launch program does not replicate the VisIt environment variables, VisIt provides an option to start each process of the VisIt compute engine under the visit script. This ensures that the environment variables that VisIt requires in order to run are indeed set up before the parallel compute engine processes are started. To enable this feature, click on the Use VisIt script to set up parallel environment check box.

Setting launcher arguments

In addition to choosing a launch program, you can also elect to give it additional command line options to influence how it launches your compute engine. To give additional command line options to the launch program, click the Launcher arguments check box and type command line options into the text field to the right of that check box.

Setting sublauncher options

To give additional command line options to the sublauncher program, click the Sublauncher arguments, Sublauncher pre-mpi command or Sublauncher post-mpi command check box and type options into the text field to the right of that check box.

Installing pre-defined host profiles shipped with VisIt

The Setup Host Profiles And Configuration window is used to install pre-defined host profiles that are shipped with VisIt. It can be accessed from the Options dropdown. It will list all the pre-defined host profiles shipped with the installation, listed according to location. From the list, you can choose one or more locations and all the host profiles for the selected locations will be installed. However, you will need to exit and restart VisIt for them to become available for use. With this window, you can also specify a default configuration for VisIt to use. Don’t forget to click Install before dismissing the window. (Figure 1.313)

Installing pre-defined host profiles from the VisIt repository

The Remote Profiles tab can be used to install pre-defined host profiles from the VisIt repository. The advantage to using the VisIt repository is that it may have additional host profiles defined after a particular release of VisIt was released. To do so, click on the Remote Profiles vertical tab in the middle of the Host Profiles window. The top section of the tab allows you to choose the remote location (currently, only VisIt’s repository is available). (Figure 1.314)

If you click the Update button, the list of host profiles available from the remote location will be displayed. (Figure 1.315)

Scroll through the list, clicking on the arrow next to a location to view the profiles available for that location, then highlight a profile and click the Import button. (Figure 1.316) The selected host profile will now show up in the hosts list in the left pane.

It is important to save your settings before exiting VisIt in order to save the newly imported host profiles for future sessions.
To finish the VisIt install on this computer select any computing centers whose resources you are using to configure host profiles automatically for their machines.

Select computing centers used

- Louisiana State University network
- National Center for Atmospheric Research (NCAR) network
- National Center for Supercomputing Applications (NCSA) network
- National Energy Scientific Research Computing Center (NERSC) network
- National Institute for Computational Sciences (NICS) network
- Oak Ridge National Laboratory (ORNL) network
- Ohio Supercomputer Center (OSC) network
- Princeton University network
- Sandia National Laboratory (SNL) closed network
- Sandia National Laboratory (SNL) open network
- Stanford Linear Accelerator Center (SLAC) network

Select default configuration

- None (use VisIt’s standard defaults)
- Chombo Users
- Lawrence Livermore National Laboratory
- Princeton University

Fig. 1.313: The Host Profile Configuration Window
Fig. 1.314: Remote Profiles tab
Fig. 1.315: Remote Profiles tab with updated content
Fig. 1.316: Remote Profiles tab with host selected for import
Engine launch options window

The engine launch options window, shown in (Figure 1.317), is used to pick a launch profile to use when there are multiple launch profiles for a host or if there are any parallel launch profiles. When there is a single serial host profile or no host profiles, the window is not activated and VisIt launches a serial compute engine. The window’s primary purpose is to select a launch profile and set some parallel options such as the number of processors. This window is provided as a convenience so host profiles do not have to be modified each time you want to launch a parallel engine to run with a different number of processors.

The engine launch options window has a list of launch profiles from which to choose. The active profile for the host is selected by default though another profile can be used instead. Once a launch profile is selected, the parallel options such as the number of processors/nodes, processor count, can be changed to fine-tune how the compute engine is launched. After making any changes, click the window’s OK button to launch the compute engine. Clicking the Cancel button prevents the compute engine from being launched.

![Engine launch options window](image)

Fig. 1.317: Engine launch options window

Setting the number of processors

The number of processors determines how many processors are used by VisIt’s compute engine. Generally, a higher number of processors yields higher performance but it depends on the host platform and the database being visualized. The Num procs text field initially contains the number of processors used in the active host profile but you can change it by typing a new number of processors. The number of processors can also be incremented or decremented by clicking the up/down buttons next to the text field.
Setting batch queue options

Many compute environments schedule parallel jobs in batch queues. The engine launch options window provides a few controls that are useful for batch queue systems. The first option is the number of nodes which determines the number of smaller portions of the computer that are allocated to a particular task. Typically the number of processors is evenly divisible by the number of nodes but the window allows you to specify the number of nodes such that not all processors within a node need be active. You can set the number of nodes, by typing a new number into the Num nodes text field or you can increment or decrement the number by clicking on the arrow buttons to the right of the text field. The second option is the bank which is a large collection of nodes from which nodes can be allocated. To change the bank, you can type a new bank name into the Bank text field. The final option that the window allows to be changed is the time limit. The time limit is an important piece of information to set because it can help to determine when the compute engine is scheduled to run. A smaller time limit can increase the likelihood that a task will be scheduled to run sooner than a longer running task. You can change the time limit by typing a new number of minutes into the Time limit text field.

Setting the machine file

Some compute environments use machine files, text files that contain the names of the nodes to use for executing a parallel job, when running a parallel job. If you are running VisIt in such an environment, the engine launch options window provides a text field called Machine file. The Machine file text field allows you to enter the name of a new machine file if you want to override which machine file is used for the selected host profile.

1.14 Compute Engines

VisIt can have many compute engines running at the same time. Much of the time the compute engines are those that are installed with VisIt but on occasion, simulation codes may be instrumented to act as VisIt compute engines capable of performing visualization operations on simulation data as it is created. When a simulation is used as a VisIt compute engine, VisIt can access data directly from the simulation without the need to translate data into another format or write it out to disk. When simulations are instrumented to become VisIt compute engines, they have all of the capabilities of a standard VisIt compute engine and more. Specifically, simulations can accept additional simulation-defined control commands that direct them to perform actions such as writing a restart file. Since simulations offer extra capabilities over a normal VisIt compute engine, VisIt provides different windows in order to manage them. To manage compute engines and check on progress, VisIt provides a Compute Engine Window. VisIt provides the Simulation Window to manage simulations, display their progress, and provide extra controls for the simulations.

1.14.1 Compute Engines Window

You can open the Compute Engines Window, shown in Figure 1.318, by selecting the Compute engines option from the Main Window’s File menu. The main purpose of the Compute Engines Window is to display the progress of a compute engine as it completes a task. The window has two status bars. The top status bar indicates the progress of the overall task. The bottom status bar indicates that compute engine’s progress through the current processing stage. The window also provides buttons for interrupting and closing compute engines, as well as an Engine Information Area that indicates how many processors the engine uses and its style of load balancing.

Picking a compute engine

The Compute Engines Window has the concept of an active compute engine. Only the active compute engine’s progress is displayed in the status bars. The active compute engine is also the engine that is interrupted or closed. To pick a new active compute engine, choose a compute engine name from the Engine menu. The Engine menu contains the names of all compute engines that VisIt is running.
Fig. 1.318: Compute Engines Window
Interrupting a compute engine

Some operations in VisIt may take a long time to complete so most computations are broken down into stages. In the event that you do not want to wait for an operation to complete, or if you realize that you made a mistake, you can interrupt a compute engine. When you click the **Interrupt engine** button a signal is sent to the compute engine that tells it to stop its work. The compute engine handles the interrupt requests after it completes the current stage so there can be a small delay before the compute engine is interrupted. Any plots being generated when a compute engine is interrupted are sent into the error state and are listed in red in the **Plot list** until they are regenerated.

Closing a compute engine

![Close compute engine confirmation dialog](image)

You can close a compute engine when you no longer need it by clicking the **Close engine** button. The compute engine is closed only after you click **Yes** in a confirmation dialog window.

Clearing a compute engine’s cache

As the compute engine processes data, it caches calculation results in case they are needed again. This includes meshes and variables that have been read from databases as well as the results from more complicated calculations involving expressions and operators. VisIt’s compute engine periodically clears the cache of items that it no longer needs but if you want to explicitly clear the cache to free up more memory, you can click the **Clear cache** button in the **Compute Engine Window**.

1.14.2 Simulation Window

You can open the **Simulation Window**, shown in Figure 1.320, by selecting the **Simulations** option from the **Main Window’s File** menu. The main purpose of the **Simulation Window** is to display the progress of a simulation that is acting as a VisIt compute engine as it completes its visualization tasks. The **Simulation Window** also provides
![Simulation Window](image)

**Fig. 1.320: Simulation Window**

### Table 1: Simulation Attributes

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Host</strong></td>
<td>naples</td>
</tr>
<tr>
<td><strong>Name</strong></td>
<td>mandelbrot</td>
</tr>
<tr>
<td><strong>Date</strong></td>
<td>Thu Oct 26 16:14:46 2017</td>
</tr>
<tr>
<td><strong>Num Pro...</strong></td>
<td>1</td>
</tr>
<tr>
<td><strong>path</strong></td>
<td>/path/to/where/sim/was/started</td>
</tr>
<tr>
<td><strong>comment</strong></td>
<td>Demonstrates creating the Mandelbrot set on an AMR m...</td>
</tr>
<tr>
<td><strong>uiFile</strong></td>
<td>mandelbrot.ui</td>
</tr>
</tbody>
</table>

### Simulation Status

- **Simulation status**: Stopped

### Visit Status

- Interrupt
- Clear cache
- Disconnect

### Controls

- halt
- step
- run
- update
- toggleupdates

- Enable time cycle ranging

---

**1.14. Compute Engines**
buttons that direct the simulation to perform simulation-defined commands such as saving out a restart dump. The list of commands depends on the functionality that the simulation exposes to VisIt when instrumented.

The Simulation Window is divided up into two main areas. The top of the window, called the Simulation attributes area, displays various attributes of the simulation such as its name, when it was started, the name of the computer where it is running, the number of processors, etc. Below the Simulation attributes area, you will find controls that are also present in the Compute Engines Window such as the Interrupt button and Clear cache button. The Disconnect button is specific to the Simulation Window and when you click it, VisIt will detach from the running simulation, allowing it to continue its calculation. You can reconnect to the simulation later to check on its progress or create more visualizations.

Below the Simulation attributes area, you can access Commands, Messages, and Strip Charts. The Commands tab displays buttons for simulation-defined commands. When a simulation is instrumented to act as a VisIt compute engine, it publishes a list of commands that it will accept when connected to VisIt. This allows the simulation to provide hooks that allow the user to tell the simulation to execute certain commands like writing a restart file. Depending on the complexity of the commands exposed, VisIt could ultimately be used to steer the simulation as well as visualize its results. The Messages tab displays messages from the simulation. The Strip Charts tab shows traces of specific quantities published from the simulation to VisIt.

1.15 Preferences

In this chapter, we will discuss how to set and save user preferences. User preferences affect the default values for plots and operators as well as window properties like the background color. This chapter reveals where those settings are saved and how to modify them.

1.15.1 How VisIt Uses Preferences

VisIt’s preferences are saved into two levels of XML files that are stored in the user’s home directory and in the global VisIt installation directory. The global preferences are read first and they allow the system administrator to set global preferences for all users. After VisIt reads the global preferences, it reads the preferences file for the current user. These settings include things like the color of the GUI and the initial directory from which to read files. Most of the attributes that are settable in VisIt can be saved to the preferences files for future VisIt sessions.

1.15.2 Setting Default Values

Some windows have a button called Make default that sets the default attributes for the window. This is typically the case for plot and operator attribute windows. Other windows that have a Make default button include the Annotation, Lighting, Material Reconstruction Options, Mesh Management Options, Pick, QueryOverTime and Interactors windows. Setting the attributes with the Apply button sets the attributes for the active plots or operators. Setting the default attributes sets the attributes for future plots and operators. When saving the settings using Save Settings from the Options menu, the default attributes are saved. An example of a Make default button is shown in Figure 1.321.
1.15.3 How to Save Settings

To save preferences in VisIt, select **Save settings** from the **Main** window’s **Options** menu. When VisIt saves the current settings to the users preferences file they are used to set the initial state the next time the user runs VisIt. VisIt does not automatically save settings when changes are made to the default attributes for plots, operators, or various control windows. For windows that only have current attributes (windows without a **Make default** button), the current attributes are saved. For windows that have current and default attributes (windows with a **Make default** button), the default attributes are saved.

To save the entire state of VisIt, which includes things such as the plots in the window and the operators applied to the plots for each visualization window, select either **Save session** or **Save session as** from the **Main** window’s **File** menu. When using **Save session**, if a session has already been restored or saved, VisIt will overwrite the existing session file. If a session has not already been restored or saved, VisIt will bring up a dialog window that will allow the user to specify the location and name of the session file. When using **Save session as** VisIt will always bring up a dialog window that will allow the user to specify the location and name of the session file and prompt the user to confirm before overwriting an existing session file.

VisIt saves two preference files, the first of which stores preferences for VisIt’s GUI while the second file stores preferences for VisIt’s state. When running VisIt on UNIX and MacOS X systems, the preference files are called: guiconfig and config and they are saved in the .visit directory in the users home directory. The Windows version of the .visit directory is %USERPROFILE%\Documents\VisIt, which may be something like: C:\Users\<your-user-name>\Documents\VisIt.

To run VisIt without reading the saved settings, add **–noconfig** to the command line when running VisIt. The **–noconfig** argument is often useful when running an updated version of VisIt that is incompatible with the saved settings. VisIt settings are usually compatible between different versions but this is not always the case and some users have had trouble on occasion when transitioning to a newer version. If VisIt has stability problems when it starts up after upgrading to a newer version, add the **–noconfig** option to the command line and save the settings to write over any older preference files.

1.15.4 Appearance Window

The **Appearance** window is responsible for setting preferences for the appearance of the GUI windows. The **Appearance** window shown in Figure 1.322 is brought up by selecting **Appearance** from the main window’s **Options** menu. It can be used to set the GUI colors as well as other attributes such as the style and orientation. In order to change any of the appearance attributes, the user must first uncheck the **Use default system appearance** check box.
Changing GUI colors

To change the GUI colors using the Appearance window, click on the color button next to the color to be changed. To change the background color (the color of the GUI’s windows), click on the GUI background color button and select a new color from the Popup color menu. To change the foreground color (the color used to draw text), click the GUI foreground color button and select a new color from the Popup color menu.

VisIt will issue an error message if the colors chosen for both the background and foreground colors are close enough that they cannot be distinguished so that the user does not accidentally get into a situation where the controls in VisIt’s GUI become too difficult to read. Some application styles, such as Aqua, do not use the background color so setting the background has no effect unless an application style like Windows is chosen, which does use the background color.

Changing GUI Style

VisIt’s GUI adapts its look and feel, or application style, to the platform on which it is running. It is also possible to make the GUI use other application styles, although for the most part they look fairly similar.

To change the style select a new style from the GUI style menu. It is frequently necessary to change the GUI font by either changing the font description in the GUI font text box or selecting a new font from the font selection window, which is brought up by clicking on the ... button to the right of the GUI font text field.

Changing GUI Orientation

By default, VisIt’s Main window appears as a vertical window to the left of the visualization windows. The default configuration often makes the best use of the display with wide aspect ratio screens. It has become very rare to encounter screens where the horizontal orientation makes better use of the display, so it is not recommended and will most likely be deprecated in future versions of VisIt.

1.15.5 Plugin Manager Window

The Plugin Manager window, shown in Figure 1.323, allows the user to see which plugins are available for plots, operators, and databases. Not all plugins have to be loaded, in fact, many operator plugins are not loaded by default. The Plugin Manager window allows the user to specify which plugins are loaded when VisIt is started. The Plugin Manager window is brought up by selecting Plugin Manager from the Main window’s Options menu.

Enabling and Disabling Plugins

All of VisIt’s plots, operators, and database readers are implemented as plugins that are loaded when VisIt first starts up. Some plugins are not likely to be used by most people so they should not be loaded. The Plugin Manager window provides a mechanism to turn plugins on and off. The window has three tabs: Plots, Operators, and Databases. Each tab displays a list of plugins that can be loaded by VisIt. If a plugin is enabled, it has a check by its name.

Plugins can be turned on and off by checking or unchecking the check box next to a plugin’s name. Plugins are loaded at startup, so enabling or disabling plugins will not take effect unless the preferences are saved and VisIt is restarted. If plots or operators are disabled, they will not appear in the Add, Operator, PlotAtts and OpAtts menus. Similarly, disabled databases will not show up in the list of Open file type as menu in the File open window.

1.15.6 Rendering Options Window

The Rendering options window (shown in Figure 1.324) contains controls that set global options that affect how the plots in the active visualization window are drawn, as well as, look at information related to the performance of the
Fig. 1.323: The plugin manager window
graphics hardware VisIt is running on. The Rendering options window can be brought up by selecting Rendering from the Main window’s Preferences menu. The Rendering options window contains three tabs. The Basic tab contains basic rendering options, the Advanced tab contains advanced rendering options, and the Information tab contains information about the rendering performance of the graphics hardware VisIt is running on.

### Basic Rendering Options

The Antialiasing, and Specular lighting options are covered in the Making It Pretty chapter.

![Rendering options window](image)

**Fig. 1.324: The basic rendering options**

### Changing surface representations

Sometimes when visualizing large or complex databases, drawing plots with all of their shaded surfaces can take too long to be interactive, even for fast graphics hardware. To combat this problem, VisIt provides an option to view all of
the plots in the visualization window as wireframe outlines or point clouds instead of as shaded surfaces (see Figure 1.325). While being less visually informative, plots drawn as wireframe outlines or as clouds of points can still be useful for visualizations since it is possible to do the setup work like setting the view before switching back to a surface representation that is more costly to draw. To change the surface representation used to draw plots click on either the Surfaces, Wireframe or Points radio buttons below the Draw objects as label.

Fig. 1.325: The different surface representations

Using display lists

VisIt benefits from the use of hardware accelerated graphics and one of the concepts central to hardware accelerated graphics is the display list. A display list is a sequence of simple graphics commands that are stored in a computer’s graphics hardware so the hardware can draw the object described by the display list several times more quickly than it could if the graphics commands were issued directly. VisIt tries to make maximum use of display lists when necessary so it can draw plots as fast as possible.

By default, VisIt decides when to and when not to use display lists. Typically, when running VisIt on a local workstation with plots that result in fewer than a couple million graphics primitives, VisIt does not use display lists because the cost of creating them is more expensive than just drawing the graphics primitives without display lists. When running on a Unix version of VisIt on a remote computer and displaying the results back to a workstation using an X-server, it is almost always advantageous to create display lists for plot geometry. Without display lists, VisIt must transmit the plot geometry over the network to the X-server every time it renders an image. VisIt can be set to either use or not use display lists all the time. To change the way VisIt uses display lists click on either the Auto, Always or Never radio buttons below the Use display lists label.

Stereo images

Stereo images, which are composites of left and right eye images, can convey additional depth information that cannot be expressed by images that are generated using a single eye point. VisIt provides four forms of stereo images: red/blue, red/green, interlace, and crystal eyes. A red/blue stereo image (see Figure 1.326) is similar to frames from early 3D movies in that it appears stereo only when using red/blue stereo glasses. Unfortunately, red/blue stereo images are not very useful for visualization because colors are lost since most of the color ends up in the magenta range when the red and blue color channels are merged. Red/green stereo suffers a similar color loss. Interlaced images alternate lines in the image with left and right eye views so that squinting makes the image look somewhat 3D. VisIt’s crystal eyes option requires the use of special virtual reality goggles for images to appear to be 3D but this option is by far the best since it allows interactive frame rates with images that really appear to stand out from the computer monitor. VisIt does not use stereo imaging by default since it makes images draw slower because an image must be drawn for both the left eye and the right eye. To enable stereo images, check the Stereo check box. To change the type of stereo
images generated, click on either the **Red/Blue**, **Interlace**, **Crystal Eyes** or **Red/Green** radio boxes under the **Stereo** check box.

![Image of stereo image types](image.png)

**Fig. 1.326: Some various stereo image types**

### Advanced Rendering Options

The **Shadows**, and **Depth Cueing** options are covered in the *Making It Pretty* chapter.

### Scalable rendering

**VisIt** typically uses graphics hardware on the local computer to very quickly draw plots once they have been generated by the compute engine. This becomes impractical for very large databases because the amount of memory needed to store the graphics commands that draw the plots quickly exceeds the amount of memory in the graphics hardware. Large sets of graphics commands can also degrade performance when they must be shipped over slow networks from the compute engine to the **VisIt**’s viewer. **VisIt** provides a scalable rendering option that can improve both of these situations by creating the actual plot images, in parallel, on the compute engine, compressing them, and then transmitting only an image to the viewer where the image can be displayed.

Scalable rendering can be orders of magnitude faster for large databases than **VisIt**’s conventional image drawing strategy because large databases are typically processed using a parallel compute engine. When using scalable rendering with a parallel compute engine, **VisIt** is able to draw small pieces of the plot on each processor in parallel and then glue the image together before sending it to the viewer to be displayed. Not only has the image likely been created faster, but the size of the image is usually on the order of a megabyte instead of the tens or hundreds of megabytes needed to transmit graphics commands, which results in faster transmission of the image to the viewer. The drawback of scalable rendering is that it is usually not as interactive as graphics hardware because each time the view is changed or some other change is made to the plots, the image must be resent to the viewer over the network.

**VisIt** can automatically determine when to stop sending geometry to the viewer in favor of sending scalably rendered images. The scalable rendering threshold determines when **VisIt** switches between sending geometry and doing scalable rendering. The threshold is based on the number of polygons to be rendered. The scalable rendering threshold can be changed by entering a new number of polygons into the **When polygon count exceeds** spin box. The number is specified in thousands of polygons.

It is also possible to have **VisIt** always or never use scalable rendering. To change the scalable rendering mode, click on either the **Auto**, **Always** or **Never** radio boxes under the **Use scalable rendering** label.
Fig. 1.327: The advanced rendering options
Fig. 1.328: The rendering information
Scalable rendering

The scalable rendering indicates if the compute engine used scalable rendering to render the image displayed in the viewer. The use of scalable rendering is indicated next to the Use Scalable Rendering: label.

Frames per second

The frames per second refers to the number of times that VisIt can draw the plots in the visualization window in the course of a second. VisIt displays the minimum, average, and maximum frame rates achieved during the last draw operation, like rotating the image with the mouse. They are displayed next to the Frames per second: label. Some actions that force a redraw do not cause the information to update. An example of this is resizing the visualization window. To make VisIt update the frame rate information after each time it draws the plots in the visualization window, check the Query after each render check box.

Polygon count

The polygon count refers to the number of polygons used to represent the plots in the visualization window. VisIt displays the polygon count next to the Approximate polygon count: label.

Plot extents

The plot extents are the minimum and maximum locations of the plot in each spatial dimension. The plot extents are the smallest bounding box that can contain the plots in the visualization window. VisIt displays the plot extents for each dimension next to the X Extents:, Y Extents: and Z Extents: labels.

1.15.7 Preferences Window

The Preferences window, shown in Figure 1.329, contains controls that allow setting global options that influence VisIt’s behavior. The General tab contains a collection of miscellaneous options. This is followed by options that are grouped by functionality. The groups are contained in the Database, Session file and File panel tabs.

Copying Plots On First Reference

The Clone windows on first reference option clones all attributes of the active window to a new window when a window is made active for the first time. To control this behavior check or uncheck the Clone window on first reference check box.

Posting Windows By Default

When a postable window, such as a plot attributes window is brought up, the window manager is free to show the window wherever it likes. When using VisIt on a large display where the windows might pop up very far away from VisIt’s Main window, it is sometimes convenient to make sure that windows that can be posted to the Notepad area are initially posted to the Notepad area instead of popping up wherever the window manager puts them. To make postable windows post to the Notepad area by default when they are shown, check the Post windows when shown check box.
Fig. 1.329: The preferences window
Reading Accurate Cycles and Times From Databases

Many of the file formats that VisIt reads contain a single time state, making accurate cycles and times unavailable in VisIt’s metadata for all but the open time state. To get accurate times and cycles for these types of files, VisIt would have to open each file in the database, which can be a costly operation. VisIt does not go to this extra effort unless the **Try harder to get accurate cycles/times** option is enabled. This option allows VisIt to create meaningful cycle or time-based database correlations for groups of single time state databases. Note that databases that are already open will need to be reopened in order for VisIt to retrieve updated cycles and times.

File Panel Properties

The File panel is a deprecated feature that will be removed in a future release of VisIt. The File panel is enabled by checking the **Show selected files** check box. It is not recommended for use.

1.16 Help

In this chapter, we will discuss how to use VisIt’s online help. VisIt’s online help consists of release notes, copyright information, Frequently Asked Questions (FAQ), and the contents of this manual. The release notes help page lists the complete set of bug fixes and enhancements for the current version of VisIt with links to the release notes for older versions. The copyright information help page lists VisIt’s copyright agreement. The FAQ help page lists commonly asked questions and the answers to those questions. Beginning VisIt users should read through the FAQ help page to find the answers to commonly asked questions. Finally, the contents of this manual are available as online help.

1.16.1 About VisIt

VisIt provides a Splash screen (Figure 1.330) that appears when the tool is launched. The Splash screen has three purposes: entertainment, displaying startup progress, and telling the user about VisIt. As VisIt launches, the Splash screen cycles through a handful of images that show some of VisIt’s capabilities and it also tells the user what happens while VisIt is launching. Once VisIt is launched, you can look at some information about VisIt by selecting the **About** option from the Main Window’s Help menu. Choosing that menu option displays the Splash screen which can be hidden by clicking its **Dismiss** button.

1.16.2 Help Window

VisIt’s Help Window, shown in Figure 1.331, displays all of VisIt’s online help content. You can open the Help Window by choosing the Help option from the Main Window’s Help menu. The Help Window has a toolbar along the top of the window while the rest of the window is divided vertically into two main areas. The left side of the window is used to select online help pages and it is further divided with tabs for help contents, help index, and bookmarks. The right side of the window displays the content for the online help pages.

Help Window Toolbar

The Help Window’s toolbar exposes buttons for navigation, changing font size, and adding bookmarks. You can hide the toolbar by double-clicking on the handle located at the far left of the toolbar. The toolbar can also be moved to other parts of the Help Window by clicking on its handle and dragging it to the top, sides, or the bottom of the Help Window.
Fig. 1.330: Splash screen
Fig. 1.331: Help window
Navigation

The toolbar contains buttons that you can use to cycle forward and backward in the list of visited help pages. The Back button has an arrow icon that points to the left and the button changes the active help page to the last visited help page. The Forward button has an arrow icon that points to the right and it switches the help page to the page that was active before the Back button was clicked. If have not visited any help pages, both of these buttons are disabled. The toolbar also contains a Home button which has a house icon. The Home button displays the VisIt home page, which describes VisIt’s features.

Changing font size

The toolbar contains two buttons that allow you to change the font size used to display online help. The Larger font button is distinguished by a large capital ‘A’ and a small triangle which points up. When the Larger font button is clicked, the font size is increased and the active help page is redrawn with the larger font. The Smaller font button looks similar to the Larger font button except that its icon’s triangle points down and its ‘A’ is smaller. The Smaller font button decreases the font size and immediately redraws the active help page using the new smaller font.

Adding a bookmark

VisIt’s Help Window provides the ability to create and save personal bookmarks. This allows you to easily come back to frequently-used sections of the online help. The toolbar contains an Add bookmark button that adds the current help page to the list of bookmarks. The Bookmarks tab in the left part of the Help Window also has this feature.

Selecting a help page

The Help Window has three tabs, shown in Figure 1.332, that allow a help page to be located in different ways. The first tab is the Contents tab and it lists all of the online help pages and allows them to be grouped into related topics. The Index tab lists all of the online help pages in an alphabetized list that can be searched for a particular help topic. The Bookmarks tab shows all bookmarked help pages which can be recalled by clicking on a bookmark.
1.16. Help
Fig. 1.332: Help tabs
Contents tab

The Contents tab lists all of the online help pages and groups them into related topics which are sometimes organized in tree format. When items are organized into a tree, an entry in the list of help pages often has a book icon next to it indicating that the topic contains other help topics. When an item has a book icon, it can be opened by double-clicking on its title or by clicking the check box to the left of the title. Items that have an icon that looks like a stack of papers contain the actual help content and clicking on them displays the help page in the right half of the Help Window.

Index tab

The Index tab lists all of the help topics alphabetically in a single searchable list. Help topics can be selected by clicking on an item in the list or by typing a help topic into the text field above the list. As words are typed into the text field, the closest match is found in the list of help topics and the topic is displayed in the right half of the Help Window.

Bookmarks tab

The Bookmarks tab lists all of the help topics that have been bookmarked for further use. To view a page that has been previously bookmarked, simply click on its title in the bookmark list. To add a bookmark for the current help page, click the Add button in the Bookmarks tab or in the Help Window's toolbar. To remove a bookmark, click on its title in the bookmark list and then click the Remove button.

1.17 Startup Options

You can get help on starting VisIt with the commands

```
visit -help
visit -fullhelp
```

For convenience, the output from visit -fullhelp is shown below.

USAGE: visit [options]:

<table>
<thead>
<tr>
<th>Interface options</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>-gui</td>
<td>Run with the Graphical User Interface (default).</td>
</tr>
<tr>
<td>-cli</td>
<td>Run with the Command Line Interface.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Movie making options</th>
<th></th>
</tr>
</thead>
</table>
| -movie                                   | Run the CLI in a movie making mode. Must be combined with -sessionfile. Will produce a simple movie by drawing all the plots in the specified session for every timestep of the database.

<table>
<thead>
<tr>
<th>Startup options</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>-o &lt;filename&gt;</td>
<td>Open the specified data file at startup.</td>
</tr>
<tr>
<td>-s &lt;filename&gt;</td>
<td>Run the specified VisIt script. Note: This argument only takes effect with -cli or -movie.</td>
</tr>
<tr>
<td>-sessionfile &lt;filename&gt;</td>
<td>Open the specified session file at startup Note that this argument only takes effect with</td>
</tr>
</tbody>
</table>

(continues on next page)
-gui or -movie.

-config <filename> Initialize the viewer at startup using the named config file. If an absolute path is not given, the file is assumed to be in the .visit directory.

-noconfig Don't process configuration files at startup.

-launchengine <host> Launch an engine at startup. The <host> parameter is optional. If it is not specified, the engine will be launched on the local host. If you wish to launch an engine on a remote host, specify the host's name as the <host> parameter.

-nosplash Do not display the splash screen at startup.

**Window options**

- small Use a smaller desktop area/window size.
- geometry <spec> What portion of the screen to use. This is a standard X Windows geometry specification. This option can be used to set the size of images generated from scripts and movies.

- viewer_geometry <spec> What portion of the screen the viewer windows will use. This is a standard X Windows geometry specification. This option overrides the -geometry option that the GUI passes to the viewer.

- window_anchor <x,y> The x,y position on the screen where VisIt's GUI will show its windows (Main window excluded).
- style <style> One of: windows,cde,motif,sgi.
- locale <locale> The locale that you want VisIt to use when displaying translated menus and controls. VisIt will use the default locale if the -locale option is not provided.

- background <color> Background color for GUI.
- foreground <color> Foreground color for GUI.
- nowin Run with viewer windows off-screen (i.e. OSMesa).
- stereo Enable active stereo, also known as the page-flipping, or 'CrystalEyes' mode.
- nowindowmetrics Prevents X11 from grabbing and moving a test widget used in calculating window borders. This option can be useful if VisIt hangs when displaying to an Apple X-server.

**Version options**

- version Do NOT run VisIt. Just print the current version.
- git_version Do NOT run VisIt. Just print the Git version it was built from.
- beta Run the current beta version.
- v <version> Run a specified version. Specifying 2 digits, such as X.Y, will run the latest patch release for that version. Specifying 3 digits, such as X.Y.Z, will run that specific version.
run-time: While running VisIt, look under the "Help" menu.
on-line: https://visit.llnl.gov
email: visit-users@ornl.gov

***************************************************************************
ADDITIONAL OPTIONS
***************************************************************************

Parallel launch options
---------------------------------------------------------------------------
Notes: All of these options are ordinarily obtained from host profiles. However, the command line options override anything in the profiles.
When parallel arguments are added but the engine is not the component being launched, -launchengine is implied. Explicitly add -launchengine to launch a remote parallel engine.
---------------------------------------------------------------------------
-settupenv Use the VisIt script to set up the environment for the engine on the compute nodes.
-par Run the parallel version. This option is implied by any of the other parallel options listed below.
-l <method> Launch in parallel using the given method.
-pl <method> Launch only the engine in parallel as specified.
-la <args> Additional arguments for the parallel launcher.
-sla <args> Additional arguments for the parallel sub-launcher.
-np <# procs> The number of processors to use.
-nn <# nodes> The number of nodes to allocate.
-p <part> Partition to run in.
-n <name> The parallel job name.
-b <bank> Bank from which to draw resources.
-t <time> Maximum job run time.
-machinefile <file> Machine file.
-expedite Makes DPCS give priority scheduling.
-icet In scalable rendering mode, use the IceT parallel image compositor (default).
-no-icet Do not use the IceT parallel compositor.

Hardware accelerated parallel (scalable) rendering options
---------------------------------------------------------------------------
Notes: These options should only be used with parallel clusters that have graphics cards. If you are using a serial version of VisIt, you are already getting hardware acceleration and these options are not needed. Furthermore, you must be in scalable rendering mode for VisIt to utilize a cluster's GPUs. By default, VisIt is configured to switch into scalable rendering mode when rendering complexity exceeds a predefined limit.
VisIt can manage the creation and tear down of X servers for you. It will do this automatically if you specify the -launch-x parameter, but you can customize the process with the -x-args and -display parameters, which respect %l and %n format specifiers.
See the VisIt wiki for more information:
http://visitusers.org/index.php?title=Parallel_Hardware_Acceleration

(continues on next page)
-hw-accel Tells VisIt that it should use graphics cards.
-n-gpus-per-node <int> Number of GPUs per node of the cluster (1).
-launch-x Tell VisIt to manage the X servers
-no-launch-x Let the cluster manager X servers [default]
-display Tells VisIt which display to use.
-x-args '<string>' Extra arguments to X server.

Load balance options

Note: Each time VisIt executes a pipeline the relevant domains for the execution are assigned to processors. This list of domains is sorted in increasing global domain number. The options below effect how domains in this list are assigned to processors. Assuming there are D domains and P processors...

-lb-block Assign the first D/P domains to processor 0, the next D/P domains to processor 1, etc.
-lb-stride Assign every Pth domain starting from the first to processor 0, every Pth domain starting from the second to processor 1, etc.
-lb-absolute Assign domains by absolute domain number % P. This guarantees a given domain is always processed by the same processor but can also lead to poor balance when only a subset of domains is selected.
-allowdynamic Dedicate one processor to spreading the work dynamically among the other processors. This mode has limitations in the types of queries it can perform. Under development.
-allowstream Similar to -lb-block, but have the domains travel down the pipeline one at a time, instead of all together. Under development.

Database differencing options

Use the '-diff <ldb> <rdb>' option to run VisIt in a database differencing mode. VisIt will generate expressions to facilitate visualization and analysis of the difference between the left-database, <ldb>, and right-database, <rdb>. VisIt will open windows to display both the left and right databases as well as their difference.

VisIt uses the Cross-Mesh Field Evaluation (CMFE) expression functions to help generate these differences. A CMFE function creates an instance of a variable from another (source) mesh on the specified (destination) mesh. VisIt can use two variants of CMFE expression functions depending on how similar the source and destination meshes are; connectivity-based (conn_cmfe) which assumes the underlying mesh(s) for the left and right databases have identical connectivity and position-based (pos_cmfe) which does not make this assumption. VisIt will attempt to automatically select which variant of CMFE expression to use based on some simple heuristics. For meshes with identical connectivity, conn_cmfe expressions are preferable because they are higher performance and do not require VisIt to perform any interpolation. In fact, the conn_cmfe operation is perfectly anti-symmetric. That is <ldb> - <rdb> = -(<rdb> - <ldb>). The same cannot be said for pos_cmfe expressions. However, pos_cmfe
expressions will attempt to generate useful results regardless of the similarity of the underlying meshes.

Note that the differences VisIt will compute in this mode are single precision. This is true regardless of whether the input data is itself double precision. VisIt will convert double precision to single precision before processing it. Although this is a result of earlier visualization-specific design requirements and constraints, the intention is that eventually double precision will be supported.

Finally, be sure to bring up Controls->Macros in the GUI to find a set of useful operations specifically tailored to database differencing. Also, typing 'help()' (including the ')' at the python prompt after starting 'visit -diff' will generate a more detailed help message.

-diff <ldb> <rdb> Indicate you wish to run VisIt in database differencing mode and specify the two databases to difference.

Note: All options occurring on the command-line *after* the '-diff' option are treated as options to the differencing script while all options occurring *before* the '-diff' option are treated as options to VisIt.

-diffsum <ldb> <rdb> Run only the difference summary method of the 'visit -diff' script, in nowin mode so its fast, print the results, and immediately exit.

-force_pos_cmfe Force use of position-based CMFE expressions.

Advanced options

-guesshost Try to guess the client host name from one of the SSH_CLIENT, SSH2_CLIENT, or SSH_CONNECTION environment variables.

-noloopback Disable use of the 127.0.0.1 loopback device.

-sshtunneling Tunnel all remote connections through ssh. NOTE: this overrides values set in the host profiles.

-noint Disable interruption capability.

-nopty Run without PTYs.

-verbose Prints status information during pipeline execution.

-dir <directory> Run a version of VisIt in the specified directory. The directory argument should specify the path to a VisIt installation directory. /bin is automatically appended to this path.

-forceversion <ver> Force the given version. Overrides all intelligent version selection logic.

-publicpluginsonly Disable all plugins but the default ones.

-compiler <cc> Require version built with the specified compiler.

-objectmode <mode> Require a specific object file mode.

-forceinteractivecli Force the CLI to behave interactively, even if run with no terminal; similar to python's '-i' flag.

-fullscreen Create the viewer window in full screen mode.

-may not be compatible with all window managers.

-viewerdisplay <dpy> Have the viewer use a different display than the
current value of DISPLAY. Can be useful for
power wall displays with a separate console.

-cycleregex <string>
A regex-style regular expression to be used
in extracting cycle numbers from file names. It
is best to bracket this string in single
quotes (' ') to avoid shell interpretation of
special characters such as star (*). The format
of the string begins with an opening '<' character,
followed by the regular expression itself followed
by a closing '>' character, optionally followed by
a space ' ' character and sub-expression reference
to indicate which part of the regular expression is
the cycle number. Default behavior is as if
-cycleregex '((0-9]+)[0-9]*\$> \0'
was specified meaning the last sequence of one
or more digits before the end of the string found
is used as the cycle number. Do a 'man 7 regex'
to get more information on regular expression
syntax.

-ui-bcast-thresholds <int1> <int2>
Two integers controlling behavior of parallel
engine waiting in a broadcast for the next RPC
from the viewer. VisIt used to rely solely upon
MPI_Bcast for this. However, many implementations
of MPI_Bcast use a polling loop that winds up
keeping all processors busy and can make them
unusable by other processes. This is particularly
bad for SMPs. So, VisIt implemented its own
broadcast using MPI's send/receive methods. <int1>
specifies the number of nanoseconds a processor
sleeps while polling for completion of the
broadcast. Specifying a value of zero (0) for <int1>
results in falling back to older behavior using
MPI's MPI_Bcast. <int1> effectively controls how
'busy' processors will be, polling for completion
of the broadcast. <int2> specifies the number of
seconds all processors should spin, polling as fast
as possible, checking for completion of the
broadcast BEFORE inserting sleeps into their
polling loops. <int2> effectively controls how
many seconds VisIt's server will be maximally
responsive (although also keeping all processors
occupied) before becoming more 'friendly' to
other processes on the same node. The defaults
are <int1> = 50000000 nanoseconds (1/20th of a sec)
and <int2> = 5 seconds meaning VisIt will spin
processors maximally for 5 seconds before inserting
sleeps such that polling happens at the rate of 20
times per second.

-idle-timeout <int>
An integer representing the number of minutes an
engine is allowed to idle (e.g. sit there doing no
execution whatsoever, waiting for commands from
the viewer). If this timeout is reached, the engine
will terminate itself. The default is 480 minutes
(8 hours).

-exec-timeout <int>
An integer representing the number of minutes an
executing engine is allowed to remain in the
execution of any single command from the viewer. If this timeout is reached, the engine will terminate itself. The default is 30 minutes. Beware that among other things, this timeout effects how long orphaned parallel processes will hang around, tying up parallel compute resources, following an exit-triggering error condition on any one process.

**Developer options (most for xml2... tools)**

- **public** xml2cmake: force install plugins publicly
- **private** xml2cmake: force install plugins privately
- **-noprimer** Permit xml2... tools to overwrite old files
- **-noprime** Silence debugging output from xml2... tools
- **-outputtoinputdir** Force xml2... tools to write output files to the directory containing the input XML file
- **-arch** Print supported architecture(s) and exit

**Debugging options**

Note: Debugging options may degrade performance

- **-debug <level>** Run with <level> levels of output logging. <level> must be between 1 and 5. This will generate debug logs (called 'vlogs' for ALL components. Note that debug logs are unbuffered. However, if you also specify 'b' immediately after the digit indicating the debug level (e.g. '-debug 3b'), the logs will be buffered. This can substantially improve performance when a lot of debug output is generated. However, also beware that when debug logs are buffered, there isn't necessarily any guarantee they will contain the most recent debug output just prior to a crash.

- **-debug_<compname> <level>** Run specified component with <level> of output logging. For example, '-debug_mdserver 4' will run the mdserver with level 4 debugging. Multiple '-debug_<compname> <level>' args are allowed.
- **-debug_engine_rank <r>** Restrict debug output to the specified rank.
- **-debug-processor-stride N** Have only every Nth processor output debug logs. Prevents overwhelming parallel file systems.
- **-clobber_vlogs** By default, VisIt maintains debug logs from the 5 most recent invocations or restarts of each VisIt component. They are named something like A.mdserver.5.vlog, A.engine_ser.5.vlog, etc with the leading letter (A-E) indicating most to least recent. The clobber_vlogs flag causes VisIt to remove all debug logs and begin creating them anew.
- **-vtk-debug** Turn on debugging of VTK objects used in pipelines.
- **-pid** Append process ids to the names of log files.
- **-timing** Save timing data to files.
- **-withhold-timing-output** Withhold timing output during execution. Prevents output of timing information from affecting
-never-output-timings

Never output timings files. This is used when you want the timer to be enabled (for usage by developers to measure inner loops), but you want to avoid blowing memory with the bookkeeping for each and every timing call.

-timing-processor-stride N

Have only every Nth processor output timing info. Prevents overwhelming parallel file systems.

-env

Print env. variables VisIt will use when run.

-dump (dump_dir)

Dump intermediate results from AVT filters, scalably rendered images, and html pages. Takes an optional argument that specifies the directory for -dump output files.

-info-dump (dump_dir)

Dump html pages only. Takes an optional argument that specifies the directory for -info-dump output files.

-gdb <args> <comp>

Run gdb with <args> on component <comp>. Default <args> is whitespace.

-break <funcname>

Add the specified breakpoint in gdb.

-xterm

With -gdb-something, run gdb in an xterm window.

-newconsole (dump_dir)

Run any VisIt component in a new console window.

-totalview <args> <comp>

Run totalview with <args> on component <comp>. Default <args> is whitespace.

-valgrind <args> <comp>

Run valgrind with <args> on component <comp>. Default <args> is --tool=memcheck --error-limit=no --num-callers=50.

-strace <args> <comp>

Run strace with <args> on component <comp>. Default <args> is -ttt -T.

In the above, all arguments between the tool name and the VisIt component name are treated as args to the tool.

-apitrace <args> <comp>

Run apitrace with <args> on component <comp>. Default <args> is trace --api gl.

In the above, all arguments between the tool name and the VisIt component name are treated as args to the tool.

-debug-malloc <args> <comp>

Run the component with the libMallocDebug library on MacOS X systems. The libMallocDebug library lets the MallocDebug application attach to the instrumented application and retrieve memory allocation statistics. The -debug-malloc flag also sets up the environment for the leaks and heap tools.

Printing heap allocations:


% visit -debug-malloc gui &
% Get the gui's <pid>
% heap <pid>

Printing memory leaks:
% visit -debug-malloc gui &
% Get the gui's <pid>
% leaks <pid>

Run with MallocDebug:
Perl does not seem to be happy with libMallocDebug
so you can run the GUI like this:
% visit -cli
>>> OpenGUI('-debug-malloc', 'MallocDebug', 'gui')
Connect to the gui with MallocDebug and do your
sampling.

- numrestarts <#> Number of attempts to restart a failed engine.
- quiet Don't print the Running message.
- protocol

Print the definitions of the state objects that
comprise the VisIt protocol so they can be compared
against the values on other computers.

1.18 Building

In this chapter, we will discuss how to build visit. The building of VisIt is automated with the build_visit script.
It will build VisIt and all of VisIt's third party libraries. It can be configured to build VisIt with a minimum of third
party libraries to building VisIt with all of it's third party libraries. This chapter describes how to build VisIt, starting
with the most simple case and moving then moving to more complex use cases.

1.18.1 Basic Usage

Doing a minimal build

When using build_visit without any arguments it will do a minimal build of VisIt downloading the VisIt source
code by making an anonymous git clone from GitHub and downloading the source code for the third party libraries
from NERSC. It will build a serial version of the code without any of the optional I/O libraries. This will result in only
the file readers that require no external dependencies to be built. Building VisIt in this fashion will give you the highest
probability of success.

./build_visit3.0.1

Building with multiple cores

When build_visit is run by default it will build the code using a single core. This may take a half a day or longer.
Modern computers have anywhere from 4 to 80 cores at the time of the writing of this chapter. You can speed up
the build process by specify that build_visit use more cores. If you are using a shared resource you probably
shouldn’t use all the cores in consideration of other users of the system. The following example specifies using 4
cores.
Specifying the third party library install location

When `build_visit` is run by default it will install the third party libraries in the directory `third_party` in the current directory. If you would like to install the libraries in another directory for the purposes of sharing them with other users of the system, you can have `build_visit` install them in a different directory. The following example specifies installing the third party libraries in a another location.

```
./build_visit3_0_1 --thirdparty-path /usr/gapps/visit/third_party
```

Building with the HDF5 and Silo libraries

Some of the more common I/O libraries that will result in building a larger number of file readers are HDF5 and Silo. The following example specifies building HDF5 and Silo.

```
./build_visit3_0_1 --hdf5 --silo
```

Building the stable optional libraries

If you are feeling lucky you can have `build_visit` build all of the optional I/O libraries that have a high probability of building. The following example specifies building the more reliable of the optional I/O libraries.

```
./build_visit3_0_1 --optional
```

Using a VisIt source code tar file

You can also have visit use the prepackaged source code for a specific version of VisIt instead of doing a git download of the source code. The tar file should be considerably smaller than a git clone. The following example uses the VisIt source code corresponding to the official 3.0.1 release of VisIt.

```
./build_visit3_0_1 --optional --tarball visit3.0.1.tar.gz
```

If `build_visit` is interrupted

If `build_visit` is interrupted while it is executing, it is suggested that you remove the directories associated with the last package it was in the process of building. `build_visit` always leaves directories intact when it runs to aid with troubleshooting failures. Likewise, `build_visit` doesn’t remove existing directories before starting to build a package. This can sometimes problems when `build_visit` is interrupted and you restart the build again.

Finishing up

Once you have successfully built VisIt, there are a couple of directions you can go. The first option is to use it in the location where it was built. The executable can run by executing the following command:

```
visit/build/bin/visit
```

if you built using a git clone.
if you built using a tar file.

The second option is to create a distribution file that you can install using visit-install. This can be done by executing the following command:

```
cd visit/build
make package
```

if you built using a git clone.

```
cd visit3.0.1/build
make package
```

if you built using a tar file.

### 1.18.2 Advanced Usage

build_visit comes with many options for features such as building a parallel version, overcoming issues with OpenGL, a rendering library used by VisIt to render images, and controlling precisely what libraries VisIt is built with.

#### Building a parallel version

One of powerful capabilities of VisIt is running in parallel on large parallel clusters. VisIt runs in parallel using a library called MPI, which stands for Message Passing Interface. There are a couple of ways in which you can build a parallel version of VisIt using MPI. If your system doesn’t already have MPI installed on it, which is typically the case with a desktop system or small cluster, then you can use MPICH, which is an open source implementation of MPI. The following example builds a parallel version using MPICH.

```
./build_visit3_0_1 --mpich
```

If your system already has MPI installed on it, which is typically the case with a large system at a computer center, you can set several environment variables that specify the location of the MPI libraries and header files. The following example uses a system installed MPI library.

```
env PAR_COMPILER=/usr/packages/mvapich2/bin/mpicc \　
PAR_COMPILER_CXX=/usr/packages/mvapich2/bin/mpicxx \　
PAR_INCLUDE=-I/usr/packages/mvapich2/include \　
PAR_LIBS=-lmp1 
./build_visit3_0_1 --parallel
```

When running in parallel, the user will typically use scalable rendering for rendering images in parallel. VisIt does this through the use of the Mesa 3D graphics library. Because of this you will want to include Mesa 3D when building a parallel version. In the following example we have included building with the Mesa 3D library.

```
./build_visit3_0_1 --mpich --osmesa
```

#### Building with Mesa as the OpenGL implementation

Mesa 3D is also an implementation of OpenGL and it can be used in place of the system OpenGL when building VisIt. There are a couple of reasons you would want to use Mesa 3D instead of the system OpenGL. The first is when you don’t have a system OpenGL, which typically occurs when building in a container or on a virtual machine. The second
is when your system implementation of OpenGL is too old to support VTK. In the following example we use Mesa 3D instead of the system OpenGL.

```bash
./build_visit3_0_1 --mesagl
```

**The difference between --mesagl and --osmesa**

When you specify `--mesagl` VTK will be built against Mesa 3D. When you specify `--osmesa` VTK is built against the system OpenGL and the Mesa 3D library is substituted at run time for OpenGL when running the parallel engine to enable scalable rendering. If you specify `--mesagl` then `--osmesa` is unnecessary and ignored if specified.

**Building on a system without internet access**

When you want to build visit on a system without internet access, you can use `build_visit` to download the third party libraries and source code to a system that has internet access and then move those files to your machine without access. The following example downloads the optional libraries, mpich and osmesa.

```bash
./build_visit3_0_1 --optional --mpich --osmesa --download-only
```

Unfortunately, due to the way the code that builds Python is implemented, some Python libraries will not be downloaded. Here is the list of commands to download those additional libraries.

```bash
wget http://portal.nersc.gov/project/visit/releases/3.0.1/third_party/Imaging-1.1.7.tar.gz
wget http://portal.nersc.gov/project/visit/releases/3.0.1/third_party/setuptools-28.0.0.tar.gz
wget http://portal.nersc.gov/project/visit/releases/3.0.1/third_party/Cython-0.25.2.tar.gz
wget http://portal.nersc.gov/project/visit/releases/3.0.1/third_party/numpy-1.14.1.zip
wget http://portal.nersc.gov/project/visit/releases/3.0.1/third_party/pyparsing-1.5.2.tar.gz
wget http://portal.nersc.gov/project/visit/releases/3.0.1/third_party/requests-2.5.1.tar.gz
wget http://portal.nersc.gov/project/visit/releases/3.0.1/third_party/seedme-python-client-v1.2.4.zip
```

It’s possible that the list could change and the above list becomes outdated. In this case you can run `build_visit` to just build Python and that will end up downloading all the files you need. The following command builds only Python.

```bash
./build_visit3_0_1 --no-thirdparty --no-visit --python
```

**Different versions of build_visit**

When you use a version of `build_visit` that has a version number in it, for example `build_visit3_0_1` then it builds that tagged version of *VisIt*. If the version of `build_visit` was from the develop branch of *VisIt*, then it will grab the latest version of *VisIt* from the develop branch. If the version of `build_visit` came from a release candidate branch, for example the v3.0 branch, then it will grab the latest version of *VisIt* from that branch.

**Troubleshooting build_visit failures**

When `build_visit` runs, it generates a log file with `_log` added to the name of the script. For example, if you are running `build_visit3_0_1` then the log file will be named `build_visit3_0_1_log`. The error that caused
the failure should be near the end of the log file. When build_visit finishes running, it will leave the directories that it used to build the packages intact. You can go into the directory of the package that failed and correct the issue and finish building and installing the package. You can then execute the build_visit command again to have it continue the build.

1.18.3 Common Build Scenarios

Building VisIt is an involved process and even with build_visit, just determining the correct selection of options can sometimes be daunting. To help, here are the steps used to build VisIt on a collection of different platforms that may serve as a starting point for your system.

In each of the scenarios below, the result is a distribution file that can be used with visit-install to install VisIt. Furthermore, in all these scenarios, build_visit was used to build the third party libraries and the initial config site file. VisIt was then manually built as outlined by doing an out of source build. The advantage to building VisIt manually is that you have more control over the build and its easier to troubleshoot failures. The advantage to an out of source build is that you can easily restart the build simply by deleting the build directory.

Kickit, a RedHat Enterprise Linux 7 system

build_visit was run to generate the third party libraries. In this case all the required and optional libraries build without problem, so --required --optional could be used. Also, in this case there wasn’t a system MPI installed so --mpich was specified to use MPICH. The --osmesa flag was also included so that VisIt could do off screen rendering.

```
./build_visit3_0_1 --required --optional --mpich --osmesa --no-visit --thirdparty-path /usr/gapps/visit/thirdparty_shared/3.0.1 --makeflags -j4
```

This built the third party libraries and generated a kickit.cmake config site file. The Setup VISITHOME & VISITARCH variables section was changed to

```
##
## Setup VISITHOME & VISITARCH variables.
##
SET(VISITHOME /usr/gapps/visit/thirdparty_shared/3.0.1)
SET(VISITARCH linux-x86_64_gcc-4.8)
VISIT_OPTION_DEFAULT(VISIT_SLIVR TRUE TYPE BOOL)
```

VisIt was then manually built with the following steps.

```
tar zxf visit3.0.1.tar.gz
cp kickit.cmake visit3.0.1/src/config-site
cd visit3.0.1
mkdir build
cd build
/usr/gapps/visit/thirdparty_shared/3.0.1/cmake/3.9.3/linux-x86_64_gcc-4.8/bin/cmake ..
-DCMAKE_BUILD_TYPE:STRING=Release
-DVISIT_INSTALL_THIRD_PARTY:BOOL=ON
-DVISIT_ENABLE_XDB:BOOL=ON
-DVISIT_PARADIS:BOOL=ON
make -j 4 package
```

Quartz, a Linux X86_64 TOSS3 cluster

build_visit was run to generate the third party libraries. In this case the system MPI was used, so information about the system MPI had to be provided with environment variables and the --parallel flag had to be specified.
In this case, all the required and optional third party libraries build without problem, so `--required` `--optional` could be used. Also, the system OpenGL implementation was outdated and `--mesagl` had to be included to provide an OpenGL implementation suitable for VisIt. Lastly, the Uintah library was built to enable building the Uintah reader.

```bash
env PAR_COMPILER=/usr/tce/packages/mvapich2/mvapich2-2.3-gcc-4.9.3/bin/mpicc \
   PAR_COMPILER_CXX=/usr/tce/packages/mvapich2/mvapich2-2.3-gcc-4.9.3/bin/mpicxx \
   PAR_INCLUDE=/usr/tce/packages/mvapich2/mvapich2-2.3-gcc-4.9.3/include \
   PAR_LIBS=-lmpl \
./build_visit3_0_1 --required --optional --mesagl --uintah --parallel \
--no-visit --thirdparty-path /usr/workspace/wsa/visit/visit/thirdparty_shared/3.0.1/ \
--toss3 \
--makeflags -j16
```

This built the third party libraries and generated a `quartz386.cmake` config site file. The `Setup VISITHOME` & `VISITARCH` variables. section was changed to

```bash"
##
## Setup VISITHOME & VISITARCH variables.
##
SET(VISITHOME /usr/gapps/visit/thirdparty_shared/3.0.1)
SET(VISITARCH linux-x86_64_gcc-4.8)
VISIT_OPTION_DEFAULT(VISIT_SLIVR TRUE TYPE BOOL)
```

The `Parallel build Setup.` section was changed to

```bash"
##
## Parallel build Setup.
##
VISIT_OPTION_DEFAULT(VISIT_PARALLEL ON TYPE BOOL)
VISIT_OPTION_DEFAULT(VISIT_MPI_CXX_FLAGS -I/usr/tce/packages/mvapich2/mvapich2-2.3- \
   -gcc-4.9.3/include TYPE STRING)
VISIT_OPTION_DEFAULT(VISIT_MPI_C_FLAGS -I/usr/tce/packages/mvapich2/mvapich2-2.3- \
   -gcc-4.9.3/include TYPE STRING)
VISIT_OPTION_DEFAULT(VISIT_MPI_LD_FLAGS "-L/usr/tce/packages/mvapich2/mvapich2-2.3- \
   -gcc-4.9.3/lib -Wl,-rpath=/usr/tce/packages/mvapich2/mvapich2-2.3-gcc-4.9.3/lib") \
   TYPE STRING)
VISIT_OPTION_DEFAULT(VISIT_MPI_LIBS mpich mpl)
VISIT_OPTION_DEFAULT(VISIT_PARALLEL_RPATH "/usr/tce/packages/mvapich2/mvapich2-2.3- \
   -gcc-4.9.3/lib")
```

VisIt was then manually built with the following steps.

```bash
tar zxf visit3.0.1.tar.gz
 cp kickit.cmake visit3.0.1/src/config-site
cd visit3.0.1
 mkdir build
 cd build
 /usr/workspace/wsa/visit/visit/thirdparty_shared/3.0.1/toss3/cmake/3.9.3/linux-x86_64_ \
   -gcc-4.9/bin/cmake \n../src -DCMAKE_BUILD_TYPE:STRING=Release \n-DVISIT_INSTALL_THIRD_PARTY:BOOL=ON -DVISIT_PARADIS:BOOL=ON
 make -j 16 package
```

**Lassen, a Linux Power9 BlueOS cluster**

`build_visit` was run to generate the third party libraries. In this case the system MPI was used, so information about the system MPI had to be provided with environment variables and the `--parallel` flag had to be specified.
In this case, a few of the optional third party libraries do not build on the system so all the desired optional third party libraries had to be explicitly listed. Also, the system OpenGL implementation was outdated and \texttt{--mesagl} had to be included to provide an OpenGL implementation suitable for VisIt. Lastly, the Uintah library was built to enable building the Uintah reader.

```bash
  /3/bin/mpicc \
  /3/bin/mpicxx \
PAR_INCLUDE=/usr/tce/packages/spectrum-mpi/ibm/spectrum-mpi-rolling-release/
  /include \
./build_visit3_0_1 \
--no-thirdparty --no-visit \ 
--cmake --python --vtk --qt --qwt \ 
--adios --adios2 --advio --boost --cfitsio --cgns --conduit \ 
--gdal --glu --h5part --hdf5 --icet --llvm --mfem \ 
--mili --moab --mxml --netcdf --openssl --p7zip \ 
--silo --szip --vtkm --vtkh --xdmf --zlib \ 
--mesagl --uintah --parallel \ 
--thirdparty-path /usr/workspace/wsa/visit/visit/thirdparty_shared/3.0.1/blueos \
--makeflags -j16
```

This built the third party libraries and generated a `lassen708.cmake` config site file. The `Setup VISITHOME & VISITARCH` variables section was changed to

```bash
## Setup VISITHOME & VISITARCH variables.
##
SET(VISITHOME /usr/workspace/wsa/visit/visit/thirdparty_shared/3.0.1/blueos)
SET(VISITARCH linux-ppc64le_gcc-4.9)
VISIT_OPTION_DEFAULT(VISIT_SLIVR TRUE TYPE BOOL)
```

The `Parallel build Setup.` section was changed to

```bash
## Parallel Build Setup.
##
VISIT_OPTION_DEFAULT(VISIT_PARALLEL TRUE TYPE BOOL)
VISIT_OPTION_DEFAULT(VISIT_MPI_CXX_FLAGS -I/usr/tce/packages/spectrum-mpi/ibm/\ 
  /include TYPE STRING)
VISIT_OPTION_DEFAULT(VISIT_MPI_C_FLAGS -I/usr/tce/packages/spectrum-mpi/ibm/\ 
  /include TYPE STRING)
VISIT_OPTION_DEFAULT(VISIT_MPI_LD_FLAGS "-L/usr/tce/packages/spectrum-mpi/ibm/\ 
  /lib -Wl,-rpath=/usr/tce/packages/spectrum-mpi/ibm/\ 
  /lib" TYPE STRING)
VISIT_OPTION_DEFAULT(VISIT_MPI_LIBS mpi_ibm)
VISIT_OPTION_DEFAULT(VISIT_PARALLEL_RPATH "/usr/tce/packages/spectrum-mpi/ibm/\ 
  /lib")
```

VisIt was then manually built with the following steps.

```bash
tar zxf visit3.0.1.tar.gz
cp lassen708.cmake visit3.0.1/src/config-site
cd visit3.0.1
mkdir build
cd build
/usr/workspace/wsa/visit/visit/thirdparty_shared/3.0.1/blueos/cmake/3.9.3/linux-
  -ppc64le_gcc-4.9/bin/cmake
```

(continues on next page)
Cori, a Cray KNL cluster

The system is set up to support the Intel compiler by default so we need to swap out the Intel environment for the GNU environment.

```
module swap PrgEnv-intel/6.0.4 PrgEnv-gnu/6.0.4
```

The Cray compiler wrappers are set up to do static linking, which causes a problem with building parallel hdf5. The linking can be changed to link dynamically by setting a couple of environment variables.

```
export XTPE_LINK_TYPE=dynamic
export CRAYPE_LINK_TYPE=dynamic
```

The linker has a bug that prevents VTK from building, which is fixed with the linker in binutils 2.32. Binutils was then manually built with the following steps.

```
wget https://mirrors.ocf.berkeley.edu/gnu/binutils/binutils-2.32.tar.gz
mkdir /project/projectdirs/visit/thirdparty_shared/3.0.1/binutils
tar zxf binutils-2.32.tar.gz
cd binutils-2.32
./configure --prefix=/project/projectdirs/visit/thirdparty_shared/3.0.1/binutils
make
make install
```

The following lines in `build_visit`

```bash
vopts=${vopts} -DCMAKE_C_FLAGS:STRING="\"${C_OPT_FLAGS}\""
vopts=${vopts} -DCMAKE_CXX_FLAGS:STRING="\"${CXX_OPT_FLAGS}\""
```

were changed to

```bash
vopts=${vopts} -DCMAKE_C_FLAGS:STRING="\"${C_OPT_FLAGS}\" -B/project/projectdirs/visit/thirdparty_shared/3.0.1/binutils/bin\"
vopts=${vopts} -DCMAKE_CXX_FLAGS:STRING="\"${CXX_OPT_FLAGS}\" -B/project/projectdirs/visit/thirdparty_shared/3.0.1/binutils/bin\"
```

to build VTK with the linker from binutils 2.32.

`build_visit` was run to generate the third party libraries. In this case the system MPI was used, so information about the system MPI had to be provided with environment variables and the `--parallel` flag had to be specified. In this case, all the required and optional third party libraries built without problem, so `--required --optional` could be used. Also, the system OpenGL implementation was outdated and `--mesagl` had to be included to provide an OpenGL implementation suitable for VisIt. Lastly, the Uintah library was built to enable building the Uintah reader.

```
env PAR_COMPILER=/opt/cray/pe/craype/2.5.15/bin/cc 
   PAR_COMPILER_CXX=/opt/cray/pe/craype/2.5.15/bin/CC 
   PAR_INCLUDE=-I/opt/cray/pe/mpt/7.7.3/gni/mpich-gnu/7.1/include 
   PAR_LIBS="-L/opt/cray/pe/mpt/7.7.3/gni/mpich-gnu/7.1/lib -Wl,-rpath=/opt/cray/pe/mpt/7.7.3/gni/mpich-gnu/7.1/lib -lmpich" 
./build_visit3_0_1 --required --optional --mesagl --uintah --parallel
```

(continues on next page)
This built the third party libraries and generated a cori08.cmake config site file. The Setup VISITHOME & VISITARCH variables. section was changed to

```bash
SET(VISITHOME /project/projectdirs/visit/thirdparty_shared/3.0.1)
SET(VISITARCH linux-x86_64_gcc-7.3)
VISIT_OPTION_DEFAULT(VISIT_SLIVR TRUE TYPE BOOL)
```

The VISIT_C_FLAGS and VISIT_CXX_FLAGS were changed to

```bash
VISIT_OPTION_DEFAULT(VISIT_C_FLAGS " -m64 -fPIC -fvisibility=hidden -B/project/‐
→projectdirs/visit/thirdparty_shared/3.0.1/binutils/bin" TYPE STRING)
VISIT_OPTION_DEFAULT(VISIT_CXX_FLAGS " -m64 -fPIC -fvisibility=hidden -B/project/‐
→projectdirs/visit/thirdparty_shared/3.0.1/binutils/bin" TYPE STRING)
```

The Parallel build Setup. section was changed to

```bash
VISIT_OPTION_DEFAULT(VISIT_PARALLEL ON TYPE BOOL)
VISIT_OPTION_DEFAULT(VISIT_MPI_CXX_FLAGS -I/opt/cray/pe/mpt/7.7.3/gni/mpich-gnu/7.1/‐
→include TYPE STRING)
VISIT_OPTION_DEFAULT(VISIT_MPI_C_FLAGS -I/opt/cray/pe/mpt/7.7.3/gni/mpich-gnu/7.1/‐
→include TYPE STRING)
VISIT_OPTION_DEFAULT(VISIT_MPI_LD_FLAGS "-L/opt/cray/pe/mpt/7.7.3/gni/mpich-gnu/7.1/‐
→lib -Wl,-rpath=/opt/cray/pe/mpt/7.7.3/gni/mpich-gnu/7.1/lib" TYPE STRING)
VISIT_OPTION_DEFAULT(VISIT_MPI_LIBS mpich)
VISIT_OPTION_DEFAULT(VISIT_PARALLEL_RPATH "/opt/cray/pe/mpt/7.7.3/gni/mpich-gnu/7.1/‐
→lib")
```

VisIt was then manually built with the following steps.

```bash
tar zxf visit3.0.1.tar.gz
cp cori08.cmake visit3.0.1/src/config-site
cd visit3.0.1
mkdir build
cd build
../project/projectdirs/visit/thirdparty_shared/3.0.1/cmake/3.9.3/linux-x86_64_gcc-7.3/‐
→bin/cmake \
./src -DCMAKE_BUILD_TYPE:STRING=Release \
-DVISIT_INSTALL_THIRD_PARTY:BOOL=ON -DVISIT_PARADIS:BOOL=ON
make -j 8 package
```

**Summit, a Linux Power9 BlueOS cluster**

The system is set up to support the IBM XL compiler by default so we need to swap out the XL compiler for the GNU compiler.
There was an error building CMake, so we used the system CMake after module loading CMake 3.9.2.

build_visit was run to generate the third party libraries. In this case the system MPI was used, so information about the system MPI had to be provided with environment variables and the `--parallel` flag had to be specified. In this case, a few of the optional third party libraries do not build on the system so all the desired optional third party libraries had to be explicitly listed. Also, the system OpenGL implementation was outdated and `--mesagl` had to be included to provide an OpenGL implementation suitable for VisIt. Lastly, the Uintah library was built to enable building the Uintah reader.

This built the third party libraries and generated a `login1.cmake` config site file. The Setup VISITHOME & VISITARCH variables. section was changed to

```
## Setup VISITHOME & VISITARCH variables.
##
SET(VISITHOME /autofs/nccs-svm1_home1/brugger1/visit/thirdparty_shared/3.0.1)
SET(VISITARCH linux-ppc64le_gcc-6.4)
VISIT_OPTION_DEFAULT(VISIT_SLIVR TRUE TYPE BOOL)
```

The Parallel build Setup. section was changed to

```
## Parallel Build Setup.
##
VISIT_OPTION_DEFAULT(VISIT_PARALLEL ON TYPE BOOL)
VISIT_OPTION_DEFAULT(VISIT_MPI_CXX_FLAGS -I/autofs/nccs-svm1_sw/summit/.swci/1-compute/opt/spack/20180914/linux-rhel7-ppc64le/gcc-6.4.0/spectrum-mpi-10.3.0.1-20190611-<cyagenta>20190611-cyaenjgora6now2nusxzkfil14mzjnudx/include</cyagenta> TYPE STRING)
VISIT_OPTION_DEFAULT(VISIT_MPI_C_FLAGS -I/autofs/nccs-svm1_sw/summit/.swci/1-compute/opt/spack/20180914/linux-rhel7-ppc64le/gcc-6.4.0/spectrum-mpi-10.3.0.1-20190611-<cyagenta>20190611-cyaenjgora6now2nusxzkfil14mzjnudx/include</cyagenta> TYPE STRING)
VISIT_OPTION_DEFAULT(VISIT_MPI_LD_FLAGS "-L/autofs/nccs-svm1_sw/summit/.swci/1-compute/opt/spack/20180914/linux-rhel7-ppc64le/gcc-6.4.0/spectrum-mpi-10.3.0.1-20190611-<cyagenta>20190611-cyaenjgora6now2nusxzkfil14mzjnudx/lib</cyagenta> -Wl,-rpath=/autofs/nccs-svm1_sw/summit/.swci/1-compute/opt/spack/20180914/linux-rhel7-ppc64le/gcc-6.4.0/spectrum-mpi-10.3.0.1-20190611-<cyagenta>20190611-cyaenjgora6now2nusxzkfil14mzjnudx/lib</cyagenta> TYPE STRING)
```
VISIT_OPTION_DEFAULT(VISIT_MPI_LIBS mpi_ibm)
VISIT_OPTION_DEFAULT(VISIT_PARALLEL_RPATH "/autofs/nccs-svml_sw/summit/.swci/1-compute/opt/spack/20180914/linux-rhel7-ppc64le/gcc-6.4.0/spectrum-mpi-10.3.0.1-20190611-cyaenjgora6now2nuszxkfli4mzjnudx/lib")

The compiler didn’t like one of the boost header files, so it was manually patched.

```
vi /autofs/nccs-svml_home1/brugger1/visit/thirdparty_shared/3.0.1/boost/1_67_0/linux-ppc64le_gcc-6.4/include/boost/numeric/interval/detail/ppc_rounding_control.hpp
```

line 99:
namespace detail {
   typedef union {
      ::boost::long_long_type imode;
      ::boost::ulong_long_type imode;
      double dmode;
   } rounding_mode_struct;

VisIt was then manually built with the following steps.

```
tar zxf visit3.0.1.tar.gz
cp login1.cmake visit3.0.1/src/config-site
cd visit3.0.1
mkdir build
cd build
/autofs/nccs-svml_sw/summit/.swci/0-core/opt/spack/20171006/linux-rhel7-ppc64le/gcc-4.8.5/cmake-3.9.2-lnpnk356fyio3b6rq5bdhr2djjiertsxk/bin/cmake \
../src -DCMAKE_BUILD_TYPE:STRING=Release \
-DVISIT_INSTALL_THIRD_PARTY:BOOL=ON
make -j 8 package
```

Trinity, a Cray KNL cluster

The system is set up to support the Intel compiler by default so we need to swap out the Intel environment for the GNU environment.

```
module swap PrgEnv-intel/6.0.4 PrgEnv-gnu/6.0.4
```

The Cray compiler wrappers are set up to do static linking, which causes a problem with building parallel hdf5. The linking can be changed to link dynamically by setting a couple of environment variables.

```
export XTPE_LINK_TYPE=dynamic
export CRAYPE_LINK_TYPE=dynamic
```

The linker has a bug that prevents VTK from building, which is fixed with the linker in binutils 2.32. Binutils was then manually built with the following steps.

```
wget https://mirrors.ocf.berkeley.edu/gnu/binutils/binutils-2.32.tar.gz
mkdir /usr/projects/views/visit/thirdparty_shared/3.0.1/binutils
tar zxf binutils-2.32.tar.gz
cd binutils-2.32
./configure --prefix=/usr/projects/views/visit/thirdparty_shared/3.0.1/binutils
make
make install
```
The following lines in `build_visit`

```
vopts="\$\{vopts\} -DCMAKE_C_FLAGS:STRING=\"\$\{C_OPT_FLAGS\}\""
vopts="\$\{vopts\} -DCMAKE_CXX_FLAGS:STRING=\"\$\{CXX_OPT_FLAGS\}\""
```

were changed to

```
vopts="\$\{vopts\} -DCMAKE_C_FLAGS:STRING=\"\$\{C_OPT_FLAGS\} -B/usr/projects/views/visit/thirdparty_shared/3.0.1/binutils/bin\""
vopts="\$\{vopts\} -DCMAKE_CXX_FLAGS:STRING=\"\$\{CXX_OPT_FLAGS\} -B/usr/projects/views/visit/thirdparty_shared/3.0.1/binutils/bin\"
```

to build VTK with the linker from binutils 2.32.

`build_visit` was run to generate the third party libraries. In this case the system MPI was used, so information about the system MPI had to be provided with environment variables and the `--parallel` flag had to be specified. In this case, all the required and optional third party libraries built without problem, so `--required --optional` could be used. Also, the system OpenGL implementation was outdated and `--mesagl` had to be included to provide an OpenGL implementation suitable for VisIt.

```
env PAR_COMPILER=/opt/cray/pe/craype/2.5.16/bin/cc
  PAR_COMPILER_CXX=/opt/cray/pe/craype/2.5.16/bin/CC
  PAR_INCLUDE=-I/opt/cray/pe/mpt/7.7.4/gni/mpich-gnu/7.1/include
  PAR_LIBS="-L/opt/cray/pe/mpt/7.7.4/gni/mpich-gnu/7.1/lib -Wl,-rpath=/opt/cray/pe/
  -mpt/7.7.4/gni/mpich-gnu/7.1/lib -lmpich"
./build_visit3_0_1 --required --optional --mesagl --parallel
--no-visit --thirdparty-path /usr/projects/views/visit/thirdparty_shared/3.0.1
--makeflags -j6
```

This built the third party libraries and generated a `tr-fe2.cmake` config site file. The Setup VISITHOME & VISITARCH variables. section was changed to

```bash
## Setup VISITHOME & VISITARCH variables.
##
SET(VISITHOME /usr/projects/views/visit/thirdparty_shared/3.0.1)
SET(VISITARCH linux-x86_64_gcc-8.2)
VISIT_OPTION_DEFAULT(VISIT_SLIVR TRUE TYPE BOOL)
```

The VISIT_C_FLAGS and VISIT_CXX_FLAGS were changed to

```
VISIT_OPTION_DEFAULT(VISIT_C_FLAGS " -m64 -fPIC -fvisibility=hidden -B/usr/projects/
  views/visit/thirdparty_shared/3.0.1/binutils/bin" TYPE STRING)
VISIT_OPTION_DEFAULT(VISIT_CXX_FLAGS " -m64 -fPIC -fvisibility=hidden -B/usr/projects/
  views/visit/thirdparty_shared/3.0.1/binutils/bin" TYPE STRING)
```

The Parallel build Setup. section was changed to

```
##
## Parallel Build Setup.
##
VISIT_OPTION_DEFAULT(VISIT_PARALLEL ON TYPE BOOL)
VISIT_OPTION_DEFAULT(VISIT_MPI_CXX_FLAGS -I/opt/cray/pe/mpt/7.7.4/gni/mpich-gnu/7.1/
  include TYPE STRING)
VISIT_OPTION_DEFAULT(VISIT_MPI_C_FLAGS -I/opt/cray/pe/mpt/7.7.4/gni/mpich-gnu/7.1/
  include TYPE STRING)
VISIT_OPTION_DEFAULT(VISIT_MPI_LD_FLAGS "-L/opt/cray/pe/mpt/7.7.4/gni/mpich-gnu/7.1/lib 
  -Wl,-rpath=/opt/cray/pe/mpt/7.7.4/gni/mpich-gnu/7.1/lib" TYPE STRING)
```

(continues on next page)
VisIt was then manually built with the following steps.

```
tar zxf visit3.0.1.tar.gz
cp tr-fe2.cmake visit3.0.1/src/config-site
cd visit3.0.1
mkdir build
cd build
/usr/projects/views/visit/thirdparty_shared/3.0.1/cmake/3.9.3/linux-x86_64_gcc-8.2/bin/cmake \
../src -DCMAKE_BUILD_TYPE:STRING=Release \ 
-DVISIT_INSTALL_THIRD_PARTY:BOOL=ON -DVISIT_PARADIS:BOOL=ON
make -j 8 package
```

## 1.19 Building on Windows

In this chapter, we will discuss how to build visit on Windows.

### 1.19.1 Prerequisites

**VisIt's Source Code**

**For a released version**

If you want to build a released version of VisIt, you can download a windows installer that contains all that is necessary from the source code downloads page. Look for the *VisIt Windows sources* link for the particular version you want.

**For the latest development version**

If you want to build the latest development version from our repository, you need to obtain source from the visit repo, and the pre-built third party dependencies from the visit-deps repo on GitHub.

**Other Software**

1. **CMake** version 3.8 or greater.
   - Don’t use the CMake included with cygwin if you plan on using the pre-built thirdparty libraries.
2. Visual Studio 2017 64-bit
   - Needed if you want to use our pre-built thirdparty libraries.
3. **NSIS** *Optional*
   - For creating an installer for VisIt. NSIS 2 is known to work. NSIS 3 hasn’t been tested.
4. **7zip** *Optional*
   - Used to untar testdata files.
5. Microsoft MPI. *Optional*

- For building VisIt’s parallel engine. Redistributable binaries and SDK’s are needed, so download and install both msmpisdk.msi and msmpisetup.exe.

### 1.19.2 Configuring With CMake GUI

Run cmake-gui.exe, which will display this window. Figure 1.333

![CMake-gui](image)

Fig. 1.333: CMake-gui
Locating Source and Build Directories

Fill in the location of VisIt’s src directory in the *Where is the source code:* section.

Then tell CMake where you want the build to go by filling in *Where to build the binaries.* It is best to create a new build directory somewhere other than inside the *src* or *windowsbuild* directories. This is called *out-of-source build* and it prevents pollution your src directory.

The Browse buttons come in handy here.

If you are building from a clone of the github repository, it is recommended to do the build in a directory outside the repo (eg peer to *visit*) to keep your checkout clean. Figure 1.334

![CMake Interface](image)

**Fig. 1.334: Setting source and build directories**
Location of windowsbuild Directory

For a released version of VisIt’s source code, the windowsbuild directory containing the pre-built thirdparty binaries is located peer to src. CMake generation should locate this directory automatically. Figure 1.335

![Directory structure with source from a released version](image1)

For development build cloned or downloaded from the github repositories, in order for CMake to locate the directory automatically, visit-deps should be peer to visit. Figure 1.336

![Expected directory structure with source from GitHub repo](image2)

If neither of the above is true for your situation, use the CMake gui to set VISIT_WINDOWS_DIR to the location of the windowsbuild directory. Figure 1.337

Limiting Plugins

By default, most of the supported database reader plugins are built, which can slow down loading of the solution in the Visual Studio IDE, and slow down the build. If you want to reduce the number of plugins built, add a CMake var using the Add Entry Button. If you are producing a version of VisIt that you plan to distribute, you should skip this step so all database reader plugins are built. Figure 1.338

To limit the database plugins to a specific set of plugins, set the Name: to VISIT_SELECTED_DATABASE_PLUGINS. The Type: should be STRING. The Value: should be a ‘;’ separated list of database plugins names. Case must match
Fig. 1.337: Setting VISIT_WINDOWS_DIR
Click OK when finished.

**Configuring**

Before configuring, you may want to suppress warnings. From the **Options** menu, choose **Warnings**. Check the **Developer Warnings** and **Deprecated Warnings** in the **Suppress Warnings** section. Click **OK**. Figure 1.339

In the main CMake Window, click the **Configure** button.
If the build directory does not exist, you will be prompted to allow its creation.

You will also be prompted to choose a generator. On Windows, this corresponds to the version of Visual Studio for which you plan to generate a solution and projects.

Currently, only Visual Studio version 2017 64-bit is supported by the prebuilt third-party libraries. Choose Visual Studio 15 2017 Win64 from the dropdown and add host=x64 to use the full 64-bit toolset. Figure 1.340

![Fig. 1.340: Choosing the generator](image)

CMakeCache entries will be displayed after the initial configure. All entries at this point will be highlighted reddish orange — a signal that you may want to modify some of them. Subsequent clicks of the Configure button highlight only entries that contain errors or entries that are new since the last configure.

You can modify how many entries are seen, and how they are viewed by selecting the: Grouped, and/or Advanced buttons. Grouped option groups similarly named items, Advanced option shows all the entries. Using both is probably the easiest to navigate for use with VisIt. Mouse-hover over individual entries (not groups) will generate a brief description. Figure 1.341

Most of the default settings should be fine, though you may want to change CMAKE_INSTALL_PREFIX from its default location within the Build directory. If you’ve grouped the entries, click the + button next to CMAKE, find CMAKE_INSTALL_PREFIX and modify it as desired.

### Parallel

If you have an MPI implementation installed (Microsoft’s MPI), you can choose to create a parallel build. Expand the VISIT section within the CMake gui, then check the box for VISIT_PARALLEL. You will have to scroll to find it.

Click the Configure button again to have CMake check the prerequisites for building parallel VisIt. If the prerequisites are met then some new cache entries related to MPI will be created. If not, the MPI entries may have to be modified by hand.
Fig. 1.341: After first configure

1.19. Building on Windows

VisIt User Manual Documentation, Release 3.1
Suppressing Regeneration

The solution file that CMake creates has a project called ZERO_CHECK that is occasionally invoked to regenerate the projects. This can be highly undesirable during development, since it may be triggered during a build and can cause numerous projects to be reloaded into the VS IDE, wasting time unnecessarily. To avoid this behavior, you can create a new CMake cache entry named CMAKE_SUPPRESS_REGENERATION, with type BOOL and make sure that it is checked. If you made this change click Configure again.

You can automate this step in your host.cmake file by adding this line to your host.cmake file:

```cmake
set(CMAKE_SUPPRESS_REGENERATION TRUE)
```

Note that setting this flag means that CMake won’t automatically reconfigure from within the VS IDE when changes are made to the build scripts (CMakeLists.txt) or Cache entries. You will have to manually reconfigure. Once reconfigured, Visual Studio will notify you the project files have been modified and prompt you to reload.

Generate

The Generate step creates the Visual Studio project and solution files. Make sure any changes made to the cache entries have been Configured and that no entries remain red, then click the Generate button.

Compile

Open the generated VisIt.sln file with Visual Studio (it may take awhile to load all the project file). Select desired Configuration and Build solution.

1.20 Acknowledgments

This document was prepared as an account of work sponsored by an agency of the United States government. Neither the United States government nor Lawrence Livermore National Security, LLC, nor any of their employees makes any warranty, expressed or implied, or assumes any legal liability or responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product, or process disclosed, or represents that its use would not infringe privately owned rights. Reference herein to any specific commercial product, process, or service by trade name, trademark, manufacturer, or otherwise does not necessarily constitute or imply its endorsement, recommendation, or favoring by the United States government or Lawrence Livermore National Security, LLC. The views and opinions of authors expressed herein do not necessarily state or reflect those of the United States government or Lawrence Livermore National Security, LLC, and shall not be used for advertising or product endorsement purposes.

1.21 Glossary

AAN

**Always, Auto, Never** Various features in VisIt support an **Always, Auto, Never** choice. A setting of **Never** means to never enable the feature and a setting of **Always** means to always enable the feature. A setting of **Auto**, which is typically the default, means the allow VisIt to decide when it thinks it is best to enable or disable the feature.

Node

**Point**

**Vertex** These terms refer to the corners or ends of mesh elements.

Node-centered
Point-centered These terms refer to a piecewise-linear (one degree of freedom at each of mesh element *corner*) interpolation scheme used to define a variable on a mesh. VTK tends to use the *point* terminology whereas VisIt tends to use the *node* terminology.

Parallel task Although developers are working to enhance VisIt to support a variety of fine-grained parallelism methods (e.g. MC or GPU) and although some portions of VisIt have supported multi-threaded processing for several years, in the currently available implementations, a parallel task is an MPI (Message Passing Interface) rank.

SIL

Subset Inclusion Lattice A Subset Inclusion Lattice or SIL is a term used to describe the often complex, graph like relationships among a variety of subsets defined for a mesh. A SIL describes which subsets and categories of subsets are contained within other subsets and subset categories. The Subset Window is the part of VisIt GUI that displays the contents of a SIL and allows the user to browse subsets and subset categories and turn subsets (and trees of subsets) on and off in visualizations.

SR

SR mode SR is an abbreviation for Scalable Rendering. This is a mode of operation where the VisIt engine performs scalable, parallel rendering and ships the final rendered image (e.g. pixels) to the viewer. This is in contrast to standard mode where the engine ships polygons to the viewer to be rendered there.

Zone

Cell These terms refer to the the individual computational elements comprising a mesh.

Zone-centered

Cell-centered These terms refer to a piecewise-constant (single degree of freedom for an entire zone) interpolation scheme used to define a field variable on a mesh. VTK tends to use the *cell* terminology whereas VisIt tends to use the *zone* terminology.

1.22 Contributing

This is a short contributing guide on the VisIt project’s use of Sphinx for documentation.

You can check out the Sphinx manual with:

```
svn co svn+ssh://<USERNAME>@edison.nersc.gov/project/projectdirs/visit/svn/visit/
    →trunk/docs/SphinxDocs
```

If you have Sphinx You can build the html manual locally using the command:

```
sphinx-build -b html . _build -a
```

You can then browse the root of the manual by pointing your browser to ./_build/index.html. The -a forces a re-build of everything. Remove it when you are constantly revising and rebuilding.

Your changes to any files in trunk/docs/SphinxDocs will go live here at approximately 50 minutes passed every even numbered hour.

1.22.1 Quick Reference

Note that the original source of most of the content here is the OpenOffice document produced with heroic effort by Brad Whitlock. A conversion tool was used to move most of the content there to Sphinx. As such, most of the Sphinx usage conventions adopted here were driven by whatever the conversion tool produced. There are numerous opportunities for adjusting this to make better use of Sphinx as we move forward. These are discussed at the *end* of this section.
• A few documents about reStructuredText and Sphinx are useful:
  – reStructuredText Primer
  – Sphinx Documentation
  – reStructuredText Markup Specification
  – reStructuredText Reference Documentation

• Sphinx uses blank lines as block separators and 2 or 4 spaces of indentation to guide parsing and interpretation of content. So, be sure to pay careful attention to blank lines and indentation. They are not there merely for style. They need to be there for Sphinx to parse and interpret the content correctly.

• Line breaks within reStructuredText inline markup constructs often cause build errors.

• Create headings by a sequence of separator characters immediately underneath and the same length as the heading. Different types of separator characters define different levels of headings

```
First Level Heading
===================
This is an example of some text under the heading...

Second Level Heading
---------------------
This is an example of some text under the heading...

Third Level Heading
~~~~~~~~~~~~~~~~~~
This is an example of some text under the heading...

Fourth level heading
""""""""""""""""""
This is an example of some text under the heading...
```

yields these headings...

20. First Level Heading

This is an example of some text under the heading...

20.1. Second Level Heading

This is an example of some text under the heading...

20.1.1. Third Level Heading

This is an example of some text under the heading...

20.1.1. Fourth level heading

This is an example of some text under the heading...

• If you want to divide sections and subsections across multiple .rst files, you can link them together using the .. toctree:: directive as is done for example in the section on VisIt Plots
Plots
=====

This chapter explains the concept of a plot and goes into detail about each of VisIt's different plot types.

.. toctree::
   :maxdepth: 1

   Working_with_Plots
   PlotTypes/index

Note that the files listed in the .. toctree:: block do not include their .rst extensions.

- Wherever possible, keep lines in .rst files to 80 columns or less.
- Avoid contractions such as isn't, can't and you've.
- Avoid hyphenation of words.
- Use VisIt_ or VisIt_'s when referring to VisIt by name.
- Use upper case for all letters in acronyms (MPI, VTK)
- Use case conventions of product names (QuickTime, TotalView, Valgrind).
- Bracket word(s) with two stars (**some words**) for bold.
- Bracket word(s) with one star (*word*) for italics.
- Bracket word(s) with two backticks (`some words`) for literal.
- Bracketed word(s) should not span line breaks.
- Use bold to refer to VisIt Widget names, Operator or Plot names and other named objects part of VisIt’s interface.
- Use the following terminology when referring to widget names.
- Avoid use of bold for other purposes. Instead use italics.
- Use literals for code, commands, arguments, file names, etc.
- Use :term:`a glossary term` at least for the first use of a glossary term in a section.
- Use :abbr:`ABR (Long Form)` at least for the first use of an acronym or abbreviation in a section.
- Subscripting, H\_2\_O, and superscripting, E = mc\^2, are supported:

```
Subscripting, H\_2\_O, and superscripting, E = mc\^2, are supported
```

Note the use of backslashed spaces so Sphinx treats it all as one word.

- Use .. figure:: and not .. image::, include captions with figures and use :scale: % to adjust image size where needed (see more below).
- LaTeX style equations can be included too (see below).
- Spell checking is supported too (see below) but you need to have PyEnchant and sphinx-contrib.spelling installed.
- Begin a line with .. followed by space for single line comments:
<table>
<thead>
<tr>
<th>Component</th>
<th>Example</th>
<th>Usage</th>
</tr>
</thead>
<tbody>
<tr>
<td>button</td>
<td><img src="image" alt="Example" /></td>
<td>The button is referred to by the text on the button. E.g. To create a new color table click on the &quot;New&quot; button.</td>
</tr>
<tr>
<td>check box</td>
<td><img src="image" alt="Example" /></td>
<td>The check box is referred to by the text to the right of the check box. E.g. To group tables by category check the &quot;Group tables by Category&quot; check box.</td>
</tr>
<tr>
<td>label</td>
<td><img src="image" alt="Example" /></td>
<td>The label is referred to by the text in the label. E.g. Click on the color table menu next to the &quot;Continuous&quot; label.</td>
</tr>
<tr>
<td>list</td>
<td><img src="image" alt="Example" /></td>
<td>Lists don’t typically have labels associated with them, so the name is chosen to be descriptive of the list. E.g. The &quot;Color table&quot; list.</td>
</tr>
<tr>
<td>menu</td>
<td><img src="image" alt="Example" /></td>
<td>The menu is referred to by the text to the left of the menu. E.g. To enable linear smoothing select the &quot;Linear&quot; option from the &quot;Smoothing&quot; menu.</td>
</tr>
<tr>
<td>option</td>
<td><img src="image" alt="Example" /></td>
<td>The option is referred to by the name in the menu and the word “option” isn’t included. E.g. To enable cubic smoothing select the &quot;Cubic Spline&quot; option from the &quot;Smoothing&quot; menu.</td>
</tr>
<tr>
<td>panel</td>
<td><img src="image" alt="Example" /></td>
<td>The panel is referred to by the text in the box surrounding the panel. E.g. The &quot;Manager&quot; panel has controls for managing color tables.</td>
</tr>
<tr>
<td>radio box</td>
<td><img src="image" alt="Example" /></td>
<td>The radio box is referred to by the text to the left of the radio buttons. E.g. Select the &quot;Continuous&quot; radio button from the &quot;Color table type&quot; radio box.</td>
</tr>
<tr>
<td>spin box</td>
<td><img src="image" alt="Example" /></td>
<td>The spin box is referred to by the text to the left of the spin box. E.g. Set the number of colors by changing the value in the &quot;Number of colors&quot; spin box.</td>
</tr>
<tr>
<td>tab</td>
<td><img src="image" alt="Example" /></td>
<td>The tab is referred to by the text at the top of the tab. E.g. The controls for setting the 3D view are located in the &quot;3D view&quot; tab.</td>
</tr>
<tr>
<td>text field</td>
<td><img src="image" alt="Example" /></td>
<td>The text field is referred to by the text to the left of the text field. E.g. Enter the name of the new color table in the &quot;Name&quot; text field.</td>
</tr>
</tbody>
</table>
.. this is a single line comment

.. This is a multi-line comment

- Define anchors ahead of sections or paragraphs you want to cross reference:

  .. _my_anchor:

  Section Heading

  -------------

  Note that the leading underscore is not part of the anchor name.

- Make anchor names unique over all pages of documentation by using the convention of prepending heading and subheading names.

- Link to anchors within this documentation like this one:

  Link to anchors *within* this documentation like :ref:`this one <my_anchor>`

- Link to other documents elsewhere online like visitusers.org:

  Link to other documents elsewhere online like `visitusers.org <https://www.visitusers.org/>`

- Link to numbered figures or tables within this documentation like Fig. 1.342:

  Link to numbered* figures or tables *within* this documentation like :numref:`Fig. %s <my_figure2>`

- Link to a downloadable file within this documentation like this one:

  Link to a downloadable file *within* this documentation like :download:`this one <../Quantitative/VerdictManual-revA.pdf>`

- If you are having trouble getting the formatting for a section worked out and the time involved to re-gen the documentation is too much, you could try an on-line, real-time reStructuredText Renderer to quickly try different things and see how they work.

### 1.22.2 More on Images

Try to use PNG formatted images. We plan to use the Sphinx generated documentation both for online HTML and for printed PDF. So, images sizes cannot be too big or they will slow HTML loads but not so small they are unusable in PDF.

Some image formats wind up enforcing physical dimensions instead of just pixel dimensions. This can have the effect of causing a nicely sized image (from pixel dimensions perspective anyways), to either be unusually large or unusually small in HTML or PDF output. In these cases, you can use the Sphinx :scale: and :width: or :height: options for a .. figure:: block. Also, be sure to use a .. figure:: directive instead of an .. image:: directive for embedding images. This is because the .. figure:: directive also supports anchoring for cross referencing.

Although all images get copied into a common directory during generation, Sphinx takes care of remapping names so there is no need to worry about collisions in image file names potentially used in different subdirectories within the source tree.
An ordinary image...

```latex
.. figure:: images/array_compose_with_bins.png
```

Same image with `scale: 50%` option

```latex
.. figure:: images/array_compose_with_bins.png
   :scale: 50%
```

Same image with an anchor for cross referencing...

```latex
.. _my_figure:

.. figure:: images/array_compose_with_bins.png
   :scale: 50%
```

which can now be cross referenced using an inline Fig. ?? like so...

```latex
Which can now be cross referenced using an inline :numref:`Fig. %s <my_figure>` like so...
```

Note the anchor has a leading underscore which the reference does not include.

Same image (different anchor though because anchors need to be unique) with a caption.
1.22.3 Tables

Sphinx supports a variety of mechanisms for defining tables. The conversion tool used to convert this documentation from its original OpenOffice format converted all tables to the grid style of table which is kinda sorta like ascii art. Large tables can result in individual lines that span many widths of the editor window. It is cumbersome to deal with but rich in capabilities. Often, the best answer is to NOT use tables and instead use definition lists as is used in the documentation on expressions.

1.22.4 Math

We add the Sphinx builtin extension sphinx.ext.mathjax to the extensions variable in conf.py. This allows Sphinx to use mathjax to do LaTeX like math equations in our documentation. For example, this LaTeX code

\[ x = \frac{-b \pm \sqrt{b^2 - 4ac}}{2a} \]
produces...

\[ x = \frac{-b \pm \sqrt{b^2 - 4ac}}{2a} \]

You can find a few examples in *Expressions*. Search there for `:`math:`. Also, this *LaTeX* Wiki page has a lot of useful information on various math symbols available in *LaTeX* and this wiki book has a lot of guidance on constructing math equations with LaTeX.

### 1.22.5 Spell Checking Using Aspell

You can do a pretty good job of spell checking using the Unix/Linux `aspell` command.

1. Run `aspell` looking for candidate miss-spelled words.

```bash
find . -name '*.rst' -exec cat {} \; | \\
grep -v '^[^.. image:\|figure:\|code:\|_]' | \\
tr `\` '@' | sed -e 's/(@.*@)//' | \\
aspell -p ./aspell.en.pws list | \\
sort | uniq > maybe_bad.out
```

The `find` command will find all `.rst` files. Succeeding `grep, tr` and `sed` pipes filter some of the `.rst` syntax away. The final pipe through `aspell` uses the personal word list (also called the personal dictionary) option, `-p ./aspell.en.pws` (*note:* the `./` is critical so don’t ignore it), to specify a file containing a list of words we allow that `aspell` would otherwise flag as incorrect. The `sort` and `uniq` pipes ensure the result doesn’t contain duplicates. But, be aware that a given miss-spelling can have multiple occurrences. The whole process produces a list of candidate miss-spelled words in `maybe_bad.out`.

2. Examine `maybe_bad.out` for words that you think are correctly spelled. If you find any, remove them from `maybe_bad.out` and add them to the end of `aspell.en.pws` being careful to update the total word count in the first line of file where, for example 572 is the word count shown in that line, `personal_ws-1.1 en 572` when this was written.

3. To find instances of remaining (miss-spelled words), use the following command.

```bash
find . -name '*.rst' -exec grep -wnHFf maybe_bad.out {} \;
```

4. It may be necessary to iterate through these steps a few times to find and correct all the misspellings.

It would be nice to create a `make spellcheck` target that does much of the above automatically. However, that involves implementing the above steps as a `cmake` program and involves more effort than available when this was implemented.

### 1.22.6 Things To Consider Going Forward

- Decide what to do about compound words such as `timestep`, `time step` or `time-step`. There are many instances to consider such as `keyframe`, `checkbox`, `pulldown`, `submenu`, `sublauncher`, etc.
- Need to populate glossary with more VisIt specific terms such as...
- Mixed materials, Species, OnionPeel, Mesh, Viewer, cycle, timestep Client-server, CMFE, Zone-centering, Node-centering, etc.
- Decide upon and then make consistent the usage of terms like `zonelcell/element` and `node/point/vertex`
- We will need to support `versions` of the manual with each release. RTD can do that. We just need to implement it.
  - If we have tagged content, then those would also represent different `versions` of the manual.
• All VisIt manuals should probably be hosted at a URL like visit.readthedocs.io and from there users can find manuals for GUI, CLI Getting Data Into VisIt, etc.

• Change name of docs dir to Sphinx and not Sphynx.

• Add at least another LLNL person to RTD project so we have coverage to fix issues as they come up.

• Additional features of Sphinx to consider adopting...
  – :guilable: role for referring to GUI widgets.
  – :command: role for OS level commands.
  – :file: role for referring to file names.
  – :menuselection: role for referring to widget paths in GUI menus. Example: Controls → View → Advanced.
  – :kbd: role for specifying a sequence of key strokes.
  – .. deprecated:: directive for deprecated functionality
  – .. versionadded:: directive for new functionality
  – .. versionchanged:: directive for when functionality changed
  – .. note:, .. warning: and/or .. danger: directives to call attention to the reader.
  – .. only:: directives for audience specific (e.g. tagged) content
    * Could use to also include developer related content but have it not appear in the user manual output
  – ..seealso:: directive for references
  – Substitutions for names of products and projects we refer to frequently such as VTK or VisIt (as is used throughout this section) or for frequently used text such as Viewer Window:

| Substitutions for names of products and projects we refer to frequently such as VTK_ or VisIt_ (as is used throughout this section) or for frequently used text such as |viswin|. |
|---|
| .. _VisIt: https://visit.llnl.gov |
| .. _VTK: https://www.vtk.org  |
| .. |viswin| replace:: **Viewer Window** |

Note that the .. _VisIt: ... substitution is already defined for the whole doctree in the rst_prolog variable in conf.py.

• Possible method for embedding python code to generate and capture images (both of the GUI and visualization images produced by VisIt) automatically
  – With the following pieces....
    * VisIt python CLI
    * pyscreenshot
      * A minor adjustment to VisIt GUI to allow a python CLI instance which used OpenGUI(args...) to inform the GUI that widgets are to be raised/mapped on state changes.
      * We can include python code directly in these .rst documents (prefaced by .. only:: directives to ensure the code does not actually appear in the generated manual) that does the work and just slurp this code out of these documents to actually run for automatic image generation.
- Generate and save VisIt visualization images.
- Use diffs on screen captured images to grab and even annotate images of GUI widgets.

```python
import pyscreenshot
import PIL

# The arg (not yet implemented) sets flag in GUI to map windows
# on state changes
OpenGUI(MapWidgetsOnStateChanges=True)
base_gui_image = pyscreenshot.grab()

OpenDatabase('visit_data_path()/%s' % siro_hdf5_test_data/globe.silo())
AddPlot("Pseudocolor","dx")
DrawPlots()

# Save VisIt rendered image for manual
SaveWindow('Plots/PlotTypes/Pseudocolor/images/figure15.png')
ClearPlots()

# Change something in PC atts to force it to map
pcatts = PseudocolorAttributes()
pcatts.colorTableName = 'Blue'
SetPlotOptions(pcatts) # Causes widget to map due to state change
pcatts.colorTableName = 'hot'
SetPlotOptions(pcatts) # Causes widget to map due to state change
gui_image = pyscreenshot.grab()

# Save image of VisIt PC Attr window
# - computes diff between gui_image and base_gui_image, bounding box
# - around it and then saves that bounding box from gui_image
diff_bbox = BBoxedDiffImage(gui_image, gui_image_base)
SaveBBoxedImage(gui_image, diff_bbox, 'Plots/PlotTypes/Pseudocolor/images/pcatts_window.png')

# Make a change to another PC att, capture and save it
pcatts.limitsMode = pcatts.CurrentPlot
SetPlotOptions(pcatts) # Causes widget to map due to state change
gui_image = pyscreenshot.grab()
SaveBBoxedImage(gui_image, diff_bbox, 'Plots/PlotTypes/Pseudocolor/images/pcatts_limit_mode_window.png')
```
CHAPTER 2

VisIt Python (CLI) Interface Manual

2.1 Introduction to VisIt

2.1.1 Overview

VisIt is a distributed, parallel, visualization tool for visualizing data defined on two and three-dimensional structured and unstructured meshes. VisIt’s distributed architecture allows it to leverage both the compute power of a large parallel computer and the graphics acceleration hardware of a local workstation. Another benefit of the distributed architecture is that VisIt can visualize the data where it is generated, eliminating the need to move data. VisIt can be controlled by a Graphical User Interface (GUI) or through the Python scripting language. More information about VisIt’s Graphical User Interface can be found in the VisIt User’s Manual.
2.1.2 Manual chapters

This manual is broken down into the following chapters:

<table>
<thead>
<tr>
<th>Chapter title</th>
<th>Chapter description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Introduction to VisIt</td>
<td>This chapter.</td>
</tr>
<tr>
<td>Python</td>
<td>Describes the basic features of the Python programming language.</td>
</tr>
<tr>
<td>Quick Recipes</td>
<td>Describes common patterns for scripting using the VisIt Python Interface.</td>
</tr>
<tr>
<td>Functions</td>
<td>Describes functions in the VisIt Python Interface.</td>
</tr>
<tr>
<td>Attributes References</td>
<td>Describes attributes for setting common operations, as well as for VisIt’s plugins</td>
</tr>
<tr>
<td>CLI Events</td>
<td>Describes possible events for callbacks.</td>
</tr>
</tbody>
</table>

2.1.3 Understanding how VisIt works

VisIt visualizes data by creating one or more plots in a visualization window, also known as a vis window. Examples of plots include Mesh plots, Contour plots and Pseudocolor plots. Plots take as input one or more mesh, material, scalar, or tensor variables. It is possible to modify the variables by applying one or more operators to the variables before passing them to a plot. Examples of operators include arithmetic operations or taking slices through the mesh. It is also possible to restrict the visualization of the data to subsets of the mesh. VisIt provides Python bindings to all of its plots and operators so they may be controlled through scripting. Each plot or operator plugin provides a function, which is added to the VisIt namespace, to create the right type of plot or operator attributes. The attribute object can then be modified by setting its fields and then it can be passed to a general-purpose function to set the plot or operator attributes. To display a complete list of functions in the VisIt Python Interface, you can type `dir()` at the Python prompt. Similarly, to inspect the contents of any object, you can type its name at the Python prompt. VisIt supports up to 16 visualization windows, also called vis windows. Each vis window is independent of the other vis windows and VisIt Python functions generally apply only to the currently active vis window. This manual explains how to use the VisIt Python Interface which is a Python extension module that controls VisIt’s viewer. In that way, the VisIt Python Interface fulfills the same role as VisIt’s GUI. The difference is that the viewer is totally controlled through Python scripting, which makes it easy to write scripts to create visualizations and even movies. Since the VisIt module controls VisIt’s viewer, the Python interpreter currently has no direct mechanism for passing data to the compute engine (see Figure [fig:architecture]). If you want to write a script that generates simulation data and have that script pass data to the compute engine, you must pass the data through a file on disk. The VisIt Python Interface comes packaged in two varieties: the extension module and the Command Line Interface (CLI). The extension module version of the VisIt Python Interface is imported into a standard Python interpreter using the import directive. VisIt’s command line interface (CLI) is essentially a Python interpreter where the VisIt Python Interface is built-in. The CLI is provided to simplify the process of running VisIt Python scripts.

2.1.4 Starting VisIt

You can invoke VisIt’s command line interface from the command line by typing:

```bash
visit -cli
```

VisIt provides a separate Python module if you instead wish to include VisIt functions in an existing Python script. In that case, you must first import the VisIt module into Python and then call the `Launch()` function to make VisIt launch and dynamically load the rest of the VisIt functions into the Python namespace. VisIt adopts this somewhat unusual approach to module loading since the lightweight “visit” front-end module can be installed as one of your Python’s site packages yet still dynamically load the real control functions from different versions of VisIt selected by the user.
Fig. 2.1: VisIt’s architecture
If you do not install the visit.so module as a Python site package, you can tell the Python interpreter where it is located by appending a new path to the sys.path variable. Be sure to substitute the correct path to visit.so on your system.

```python
import sys
sys.path.append("/path/to/visit/<version>/<architecture>/lib/site-packages")
```

Here is how to import all functions into the global Python namespace:

```python
from visit import *
Launch()
```

Here is how to import all functions into a “visit” module namespace:

```python
import visit
visit.Launch()
```

### 2.1.5 Getting started

VisIt is a tool for visualizing 2D and 3D scientific databases. The first thing to do when running VisIt is select databases to visualize. To select a database, you must first open the database using the OpenDatabase function. After a window has an open database, any number of plots and operators can be added. To create a plot, use the AddPlot function. After adding a plot, call the DrawPlots function to make sure that all of the new plots are drawn.

Example:

```python
OpenDatabase("/usr/local/visit/data/multi_curv3d.silo")
AddPlot("Pseudocolor", "u")
DrawPlots()
```

To see a list of the available plots and operators when you use the VisIt Python Interface, use the Operator Plugins and Plot Plugins functions. Each of those functions returns a tuple of strings that contain the names of the currently loaded plot or operator plugins. Each plot and operator plugin provides a function for creating an attributes object to set the plot or operator attributes. The name of the function is the name of the plugin in the tuple returned by the OperatorPlugins or PlotPlugins functions plus the word “Attributes”. For example, the “Pseudocolor” plot provides a function called PseudocolorAttributes. To set the plot attributes or the operator attributes, first use the attributes creation function to create an attributes object. Assign the newly created object to a variable name and set the fields in the object. Each object has its own set of fields. To see the available fields in an object, print the name of the variable at the Python prompt and press the Enter key. This will print the contents of the object so you can see the fields contained by the object. After setting the appropriate fields, pass the object to either the SetPlotOptions function or the SetOperatorAttributes function.

Example:

```python
OpenDatabase("/usr/local/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
AddOperator("Slice")
p = PseudocolorAttributes()
p.colorTableName = "rainbow"
p.opacity = 0.5
SetPlotOptions(p)
a = SliceAttributes()
a.originType = a.Point
a.normal, a.upAxis = (1,1,1), (-1,1,-1)
SetOperatorOptions(a)
DrawPlots()
```
That’s all there is to creating a plot using VisIt’s Python Interface. For more information on creating plots and performing specific actions in VisIt, refer to the documentation for each function later in this manual.

## 2.2 Python

### 2.2.1 Overview

Python is a general purpose, interpreted, extensible, object-oriented scripting language that was chosen for VisIt’s scripting language due to its ease of use and flexibility. VisIt’s Python interface was implemented as Python module and it allows you to enhance your Python scripts with coding to control VisIt. This chapter explains some of Python’s syntax so it will be more familiar when you examine the examples found in this document. For more information on programming in Python, there are a number of good references, including on the Internet at [http://www.python.org](http://www.python.org).

### 2.2.2 Indentation

One of the most obvious features of Python is its use of indentation for new scopes. You must take special care to indent all program logic consistently or else the Python interpreter may halt with an error, or worse, not do what you intended. You must increase indentation levels when you define a function, use an if/elif/else statement, or use any loop construct.

Note the different levels of indentation:

```python
def example_function(n):
    v = 0
    if n > 2:
        print "n greater than 2."
    else:
        v = n * n
    return v
```

### 2.2.3 Comments

Like all good programming languages, Python supports the addition of comments in the code. Comments begin with a pound character (#) and continue to the end of the line.

```python
# This is a comment
a = 5 * 5
```

### 2.2.4 Identifiers

The Python interpreter accepts any identifier that contains letters ‘A’-‘Z’, ‘a’-‘z’ and numbers ’0’-’9’ as long as the identifier does not begin with a number. The Python interpreter is case-sensitive so the identifier “case” would not be the same identifier as “CASE”. Be sure to case consistently throughout your Python code since the Python interpreter will instantiate any identifier that it has not seen before and mixing case would cause the interpreter to use multiple identifiers and cause problems that you might not expect. Identifiers can be used to refer to any type of object since Python is flexible in its treatment of types.
2.2.5 Data types

Python supports a wide variety of data types and allows you to define your own data types readily. Most types are created from a handful of building-block types such as integers, floats, strings, tuples, lists, and dictionaries.

Strings

Python has built-in support for strings and you can create them using single quotes or double quotes. You can even use both types of quotes so you can create strings that include quotes in case quotes are desired in the output. Strings are sequence objects and support operations that can break them down into characters.

```python
s = 'using single quotes'
s2 = "using double quotes"
s3 = 'nesting the "spiffy" double quotes'
```

Tuples

Python supports tuples, which can be thought of as a read-only set of objects. The members of a tuple can be of different types. Tuples are commonly used to group multiple related items into a single object that can be passed around more easily. Tuples support a number of operations. You can subscript a tuple like an array to access its individual members. You can easily determine whether an object is a member of a tuple. You can iterate over a tuple. There are many more uses for tuples. You can create tuples by enclosing a comma-separated list of objects in parenthesis.

```python
# Create a tuple
a = (1,2,3,4,5)
print "The first value in a is ": a[0]
# See if 3 is in a using the "in" operator.
print "3 is in a": 3 in a
# Create another tuple and add it to the first one to create c.
b = (6,7,8,9)
c = a + b
# Iterate over the items in the tuple
for value in c:
    print "value is: ", value
```

Lists

Lists are just like tuples except they are not read-only and they use square brackets [] to enclose the items in the list instead of using parenthesis.

```python
# Start with an empty list.
L = []
for i in range(10):
    # Add i to the list L
    L = L + [i]
print L
# Assign a value into element 6
L[5] = 1000
print L
```
## Dictionaries

Dictionaries are Python containers that allow you to store a value that is associated with a key. Dictionaries are convenient for mapping one set to another set since they allow you to perform easy lookups of values. Dictionaries are declared using curly braces and each item in the dictionary consists of a key: value pair with the key and values being separated by a colon. To perform a lookup using a dictionary, provide the key whose value you want to look up to the subscript [] operator.

```python
colors = {"red" : "rouge", "orange" : "orange", "yellow" : "jaune", "green" : "vert", "blue" : "bleu"}
```

```python
# Perform lookups using the keys.
for c in colors.keys():
    print "%s in French is: %s" % (c, colors[c])
```

### 2.2.6 Control flow

Python, like other general-purpose programming languages provides keywords that implement control flow. Control flow is an important feature to have in a programming language because it allows complex behavior to be created using a minimum amount of scripting.

#### if/elif/else

Python provides if/elif/else for conditional branching. The if statement takes any expression that evaluates to an integer and it takes the if branch if the integer value is 1 other wise it takes the else branch if it is present.

```python
# Example 1
if condition:
    do_something()

# Example 2
if condition:
    do_something()
else:
    do_something_else()

# Example 3
if condition:
    do_something()
elif condition:
    do_something_n()
else:
    do_something_else()
```

#### For loop

Python provides a for loop that allows you to iterate over all items stored in a sequence object (tuples, lists, strings). The body of the for loop executes once for each item in the sequence object and allows you to specify the name of an identifier to use in order to reference the current item.

```python
# Iterating through the characters of a string
for c in "characters":
    print c
```

(continues on next page)
# Iterating through a tuple

```python
for value in ("VisIt", "is", "coolness", "times", 100):
    print value
```

# Iterating through a list

```python
for value in ["VisIt", "is", "coolness", "times", 100]:
    print value
```

# Iterating through a range of numbers \(0, N\) created with \(\text{range}(N)\).

```python
N = 100
for i in range(N):
    print i, i*i
```

## While loop

Python provides a while loop that allows you to execute a loop body indefinitely based on some condition. The while loop can be used for iteration but can also be used to execute more complex types of loops.

```python
token = get_next_token()
while token != ":"
    do_something(token)
    token = get_next_token()
```

## 2.2.7 Functions

Python comes with many built-in functions and modules that implement additional functions. Functions can be used to execute bodies of code that are meant to be re-used. Functions can optionally take arguments and can optionally return values. Python provides the `def` keyword, which allows you to define a function. The `def` keyword is followed by the name of the function and its arguments, which should appear as a tuple next to the name of the function.

```python
# Define a function with no arguments and no return value.
def my_function():
    print "my function prints this..."

# Define a function with arguments and a return value.
def n_to_the_d_power(n, d):
    value = 1
    if d > 0:
        for i in range(d):
            value = value * n
    elif d < 0:
        value = 1. / float(n_to_the_d_power(n, -d))
    return value
```

## 2.3 Quick Recipes

### 2.3.1 Overview

This manual contains documentation for over two hundred functions and several dozen extension object types. Learning to combine the right functions in order to accomplish a visualization task without guidance would involve hours
of trial and error. To maximize productivity and start creating visualizations using VisIt’s Python Interface as fast as possible, this chapter provides some common patterns, or “quick recipes” that you can combine to quickly create complex scripts.

2.3.2 How to start

The most important question when developing a script is: “Where do I start?”. You can either use session files that you used to save the state of your visualization to initialize the plots before you start scripting or you can script every aspect of plot initialization.

Using session files

VisIt’s session files contain all of the information required to recreate plots that have been set up in previous interactive VisIt sessions. Since session files contain all of the information about plots, etc., they are natural candidates to make scripting easier since they can be used to do the hard part of setting up the complex visualization, leaving the bulk of the script to animate through time or alter the plots in some way. To use session files within a script, use the RestoreSession function.

```python
# Import a session file from the current working directory.
RestoreSession("my_visualization.session", 0)
# Now that VisIt has restored the session, animate through time.
for states in range(TimeSliderGetNStates()):
    SetTimeSliderState(state)
    SaveWindow()
```

Getting something on the screen

If you don’t want to use a session file to begin the setup for your visualization then you will have to dive into opening databases, creating plots, and animating through time. This is where all of hand-crafted scripts begin. The first step in creating a visualization is opening a database. VisIt provides the OpenDatabase function to open a database. Once a database has been opened, you can create plots from its variables using the AddPlot function. The AddPlot function takes a plot plugin name and the name of a variable from the open database. Once you’ve added a plot, it is in the new state, which means that it has not yet been submitted to the compute engine for processing. To make sure that the plot gets drawn, call the DrawPlots function.

```python
# Step 1: Open a database
OpenDatabase("/usr/local/visit/data/wave.visit")

# Step 2: Add plots
AddPlot("Pseudocolor", "pressure")
AddPlot("Mesh", "quadmesh")

# Step 3: Draw the plots
DrawPlots()

# Step 4: Animate through time and save images
for states in range(TimeSliderGetNStates()):
    SetTimeSliderState(state)
    SaveWindow()
```
2.3.3 Saving images

Much of the time, the entire purpose of using VisIt’s Python Interface is to create a script that can save out images of a time-varying database for the purpose of making movies. Saving images using VisIt’s Python Interface is a straight-forward process, involving just a few functions.

Setting the output image characteristics

VisIt provides a number of options for saving files, including: format, fileName, and image width and height, to name a few. These attributes are grouped into the SaveWindowAttributes object. To set the options that VisIt uses to save files, you must create a SaveWindowAttributes object, change the necessary attributes, and call the SetSaveWindowAttributes function. Note that if you want to create images using a specific image resolution, the best way is to use the -geometry command line argument with VisIt’s Command Line Interface and tell VisIt to use screen capture. If you instead require your script to be capable of saving several different image sizes then you can turn off screen capture and set the image resolution in the SaveWindowAttributes object.

```python
# Save a BMP file at 1024x768 resolution
s = SaveWindowAttributes()
s.format = s.BMP
s.fileName = "mybmpfile"
s.width, s.height = 1024, 768
s.screenCapture = 0
SetSaveWindowAttributes(s)
```

Saving an image

Once you have set the SaveWindowAttributes to your liking, you can call the SaveWindow function to save an image. The SaveWindow function returns the name of the image that is saved so you can use that for other purposes in your script.

```python
# Save images of all timesteps and add each image filename to a list.
names = []
for state in range(TimeSliderGetNStates()):
    SetTimeSliderState(state)
    # Save the image
    n = SaveWindow()
    names = names + [n]
print names
```

2.3.4 Working with databases

VisIt allows you to open a wide array of databases both in terms of supported file formats and in terms how databases treat time. Databases can have a single time state or can have multiple time states. Databases can natively support multiple time states or sets of single time states files can be grouped into time-varying databases using .visit files or using virtual databases. Working with databases gets even trickier if you are using VisIt to visualize a database that is still being generated by a simulation. This section describes how to interact with databases.

Opening a database

Opening a database is a relatively simple operation - most complexities arise in how the database treats time. If you only want to visualize a single time state or if your database format natively supports multiple timestates per file then opening a database requires just a single call to the OpenDatabase function.
Opening a database at time state 0

OpenDatabase("/usr/local/visit/data/allinone00.pdb")

Opening a database at a later time state

Opening a database at a later time state is done just the same as opening a database at time state zero except that you must specify the time state at which you want to open the database. There are a number of reasons for opening a database at a later time state. The most common reason for doing so, as opposed to just changing time states later, is that VisIt uses the metadata from the first opened time state to describe the contents of the database for all time states (except for certain file formats that don’t do this, i.e. SAMRAI). This means that the list of variables found for the first time state that you open is used for all time states. If your database contains a variable at a later time state that does not exist at earlier time states, you must open the database at a later time state to gain access to the transient variable.

OpenDatabase("/usr/local/visit/data/wave.visit", 17)

Opening a virtual database

VisIt provides two ways for accessing a set of single time-state files as a single time-varying database. The first method is a .visit file, which is a simple text file that contains the names of each file to be used as a time state in the time-varying database. The second method uses “virtual databases”, which allow VisIt to exploit the file naming conventions that are often employed by simulation codes when they create their dumps. In many cases, VisIt can scan a specified directory and determine which filenames look related. Filenames with close matches are grouped as individual time states into a virtual database whose name is based on the more abstract pattern used to create the filenames.

Opening first file in series wave0000.silo, wave0010.silo, ...
OpenDatabase("/usr/local/visit/data/wave0000.silo")

Opening a virtual database representing all wave*.silo files.
OpenDatabase("/usr/local/visit/data/wave*.silo database.")

Opening a remote database

VisIt supports running the client on a local computer while also allowing you to process data in parallel on a remote computer. If you want to access databases on a remote computer using VisIt’s Python Interface, the only difference to accessing a database on a local computer is that you must specify a host name as part of the database name.

Opening a file on a remote computer by giving a host name
# Also, open the database to a later time slice (17)
OpenDatabase("thunder:/usr/local/visit/data/wave.visit", 17)

Opening a compute engine

Sometimes it is advantageous to open a compute engine before opening a database. When you tell VisIt to open a database using the OpenDatabase function, VisIt also launches a compute engine and tells the compute engine to open the specified database. When the VisIt Python Interface is run with a visible window, the Engine Chooser Window will present itself so you can select a host profile. If you want to design a script that must specify parallel options, etc in batch mode where there is no Engine ChooserWindow then you have few options other than to open a compute engine before opening a database. To open a compute engine, use the OpenComputeEngine function. You can pass
the name of the host on which to run the compute engine and any arguments that must be used to launch the engine such as the number of processors.

```python
# Open a remote, parallel compute engine before opening a database
OpenComputeEngine("mcr", ("-np", "4", "-nn", "2"))
OpenDatabase("mcr:/usr/local/visit/data/multi_u3d.silo")
```

## 2.3.5 Working with plots

Plots are viewable objects, created from a database, that can be displayed in a visualization window. VisIt provides several types of plots and each plot allows you to view data using different visualization techniques. For example, the Pseudocolor plot allows you to see the general shape of a simulated object while painting colors on it according to the values stored in a variable’s scalar field. The most important functions for interacting with plots are covered in this section.

### Creating a plot

The function for adding a plot in VisIt is: `AddPlot`. The `AddPlot` function takes the name of a plot type and the name of a variable that is to be plotted and creates a new plot and adds it to the plot list. The name of the plot to be created corresponds to the name of one of VisIt’s plot plugins, which can be queried using the `PlotPlugins` function. The variable that you pass to the `AddPlot` function must be a valid variable for the opened database. New plots are not realized, meaning that they have not been submitted to the compute engine for processing. If you want to force VisIt to process the new plot you must call the `DrawPlots` function.

```python
# Names of all available plot plugins
print PlotPlugins()
# Create plots
AddPlot("Pseudocolor", "pressure")
AddPlot("Mesh", "quadmesh")
# Draw the plots
DrawPlots()
```

### Plotting materials

Plotting materials is a common operation in VisIt. The Boundary and FilledBoundary plots enable you to plot material boundaries and materials, respectively.

```python
# Plot material boundaries
AddPlot("Boundary", "mat1")
# Plot materials
AddPlot("FilledBoundary", "mat1")
```

### Setting plot attributes

Each plot type has an attributes object that controls how the plot generates its data or how it looks in the visualization window. The attributes object for each plot contains different fields. You can view the individual object fields by printing the object to the console. Each plot type provides a function that creates a new instance of one of its attribute objects. The function name is always of the form: `plotname + “Attributes”`. For example, the attributes object creation function for the Pseudocolor plot would be: `PseudocolorAttributes`. To change the attributes for a plot, you create an attributes object using the appropriate function, set the properties in the returned object, and tell VisIt to use the new plot attributes by passing the object to the `SetPlotOptions` function. Note that you should set a plot’s attributes
before calling the DrawPlots method to realize the plot since setting a plot’s attributes can cause the compute engine to recalculate the plot.

```python
# Creating a Pseudocolor plot and setting min/max values.
AddPlot("Pseudocolor", "pressure")
p = PseudocolorAttributes()
# Look in the object
print p
# Set the min/max values
p.min, p.minFlag = 0.0, 1
p.max, p.maxFlag = 10.0, 1
SetPlotOptions(p)
```

## Working with multiple plots

When you work with more than one plot, it is sometimes necessary to set the active plots because some of VisIt’s functions apply to all of the active plots. The active plot is usually the last plot that was created unless you’ve changed the list of active plots. Changing which plots are active is useful when you want to delete or hide certain plots or set their plot attributes independently. When you want to set which plots are active, use the SetActivePlots function. If you want to list the plots that you’ve created, call the ListPlots function.

```python
# Create more than 1 plot of the same type
AddPlot("Pseudocolor", "pressure")
AddPlot("Pseudocolor", "density")

# List the plots. The second plot should be active.
ListPlots()

# Draw the plots
DrawPlots()

# Hide the first plot
SetActivePlots(0)
HideActivePlots()

# Set both plots' color table to "hot"
p = PseudocolorAttributes()
p.colorTableName = "hot"
SetActivePlots((0,1))
SetPlotOptions(p)

# Show the first plot again.
SetActivePlots(0)
HideActivePlots()

# Delete the second plot
SetActivePlots(1)
DeleteActivePlots()
ListPlots()
```

## Plots in the error state

When VisIt’s compute engine cannot process a plot, the plot is put into the error state. Once a plot is in the error state, it no longer is displayed in the visualization window. If you are generating a movie, plots entering the error state can be a serious problem because you most often want all of the plots that you have created to animate through time and
not disappear in the middle of the animation. You can add extra code to your script to prevent plots from disappearing (most of the time) due to error conditions by adding a call to the DrawPlots function.

```python
# Save an image and take care of plots that entered the error state.
drawThePlots = 0
for state in range(TimeSliderGetNStates()):
    if SetTimeSliderState(state) == 0:
        drawThePlots = 1
    if drawThePlots == 1:
        if DrawPlots() == 0:
            print "VisIt could not draw plots for state: %d" % state
        else:
            drawThePlots = 0
    SaveWindow()
```

### 2.3.6 Operators

Operators are filters that are applied to database variables before the compute engine uses them to create plots. Operators can be linked one after the other to form chains of operators that can drastically transform the data before plotting it.

#### Adding operators

Adding an operator is similar to adding a plot in that you call a function with the name of the operator to be added. The list of available operators is returned by the OperatorPlugins function. Any of the names returned in that plugin can be used to add an operator using the AddOperator function. Operators are added to the active plots by default but you can also force VisIt to add them to all plots in the plot list.

```python
# Print available operators
print OperatorPlugins()
# Create a plot
AddPlot("Pseudocolor")
# Add an Isovolume operator and a Slice operator
AddOperator("Isovolume")
AddOperator("Slice")
DrawPlots()
```

#### Setting operator attributes

Each plot gets its own instance of an operator which means that you can set each plot’s operator attributes independently. Like plots, operators use objects to set their attributes. These objects are returned by functions whose names are of the form: `operatorname + “Attributes”`. Once you have created an operator attributes object, you can pass it to the SetOperatorOptions to set the options for an operator. Note that setting the attributes for an operator nearly always causes the compute engine to recalculate the operator. You can use the power of VisIt’s Python Interface to create complex operator behavior such as in the following code example, which moves slice planes through a Pseudocolor plot.

```python
OpenDatabase("/usr/local/visit/data/noise.silo")
AddPlot("Pseudocolor", "hardyglocal")
AddOperator("Slice")
s = SliceAttributes()
s.originType = s.Percent
s.project2d = 0
```

(continues on next page)
SetOperatorOptions(s)
DrawPlots()

nSteps = 20
for axis in (0, 1, 2):
    s.axisType = axis
    for step in range(nSteps):
        t = float(step) / float(nSteps - 1)
        s.originPercent = t * 100.
    SetOperatorOptions(s)
    SaveWindow()

2.3.7 Quantitative operations

This section focuses on some of the operations that allow you to examine your data more quantitatively.

Defining expressions

VisIt allows you to create derived variables using its powerful expressions language. You can plot or query variables created using expressions just as you would if they were read from a database. VisIt’s Python Interface allows you to create new scalar, vector, tensor variables using the DefineScalarExpression, DefineVectorExpression, and DefineTensorExpression functions.

# Creating a new expression
OpenDatabase("/usr/local/visit/data/noise.silo")
AddPlot("Pseudocolor", "hardyglobal")
DrawPlots()
DefineScalarExpression("newvar", "sin(hardyglobal) + cos(shepardglobal")
ChangeActivePlotsVar("newvar")

Pick

VisIt allows you to pick on cells, nodes, and points within a database and return information for the item of interest. To that end, VisIt provides several pick functions. Once a pick function has been called, you can call the GetPickOutput function to get a string that contains the pick information. The information in the string could be used for a multitude of uses such as building a test suite for a simulation code.

OpenDatabase("/usr/local/visit/data/noise.silo")
AddPlot("Pseudocolor", "hgslice")
DrawPlots()
s = []
# Pick by a node id
PickbyNode(300)
s = s + [GetPickOutput()]
# Pick by a cell id
PickByZone(250)
s = s + [GetPickOutput()]
# Pick on a cell using a 3d point
Pick((-2., 2., 0.))
s = s + [GetPickOutput()]
# Pick on the node closest to (-2,2,0)
NodePick((-2,2,0))

(continues on next page)
Lineout

VisIt allows you to extract data along a line, called a lineout, and plot the data using a Curve plot.

```python
OpenDatabase("/usr/local/visit/data/noise.silo")
AddPlot("Pseudocolor", "hgslice")
DrawPlots()
Lineout((-5,-3), (5,8))
# Specify a number of sample points
Lineout((-5,-4), (5,7))
```

Query

VisIt can perform a number of different queries based on values calculated about plots or their originating database.

```python
OpenDatabase("/usr/local/visit/data/noise.silo")
AddPlot("Pseudocolor", "hardyglobal")
DrawPlots()
Query("NumNodes")
print "The float value is: \$g\" % GetQueryOutputValue()
Query("NumNodes")
```

Finding the min and the max

A common operation in debugging a simulation code is examining the min and max values. Here is a pattern that allows you to print out the min and the max values and their locations in the database and also see them visually.

```python
# Define a helper function to get the id's of the MinMax query.
def GetMinMaxIds():
    Query("MinMax")
    import string
    s = string.split(GetQueryOutputString(), " ")
    retval = []
    nextGood = 0
    idType = 0
    for token in s:
        if token == "(zone" or token == "(cell":
            idType = 1
            nextGood = 1
            continue
        elif token == "(node":
            idType = 0
            nextGood = 1
            continue
        if nextGood == 1:
            nextGood = 0
            retval = retval + [(idType, int(token))]
    return retval
```

(continues on next page)
# Set up a plot
OpenDatabase("/usr/local/visit/data/noise.silo")
AddPlot("Pseudocolor", "hgslice")
DrawPlots()

# Do picks on the ids that were returned by MinMax.
for ids in GetMinMaxIds():
    idType = ids[0]
    id = ids[1]
    if idType == 0:
        PickByNode(id)
    else:
        PickByZone(id)

## 2.3.8 Subsetting

VisIt allows the user to turn off subsets of the visualization using a number of different methods. Databases can be divided up any number of ways: domains, materials, etc. This section provides some details on how to remove materials and domains from your visualization.

### Turning off domains

VisIt’s Python Interface provides the TurnDomainsOn and TurnDomainsOff functions to make it easy to turn domains on and off.

```
OpenDatabase("/usr/local/visit/data/multi_rect2d.silo")
AddPlot("Pseudocolor", "d")
DrawPlots()
# Turning off all but the last domain
d = GetDomains()
for dom in d[:-1]:
    TurnDomainsOff(dom)
# Turn all domains off
TurnDomainsOff()
# Turn on domains 3,5,7
TurnDomainsOn((d[3], d[5], d[7]))
```

### Turning off materials

VisIt’s Python Interface provides the TurnMaterialsOn and TurnMaterialsOff functions to make it easy to turn materials on and off.

```
OpenDatabase("/usr/local/visit/data/multi_rect2d.silo")
AddPlot("FilledBoundary", "mat1")
DrawPlots()
# Print the materials are:
GetMaterials()
# Turn off material 2
TurnMaterialsOff("2")
```
2.3.9 View

Setting up the view in your Python script is one of the most important things you can do to ensure the quality of your visualization because the view concentrates attention on an object of interest. VisIt provides different methods for setting the view, depending on the dimensionality of the plots in the visualization window but despite differences in how the view is set, the general procedure is basically the same.

Setting the 2D view

The 2D view consists of a rectangular window in 2D space and a 2D viewport in the visualization window. The window in 2D space determines what parts of the visualization you will look at while the viewport determines where the images will appear in the visualization window. It is not necessary to change the viewport most of the time.

```
OpenDatabase("/usr/local/visit/data/noise.silo")
AddPlot("Pseudocolor", "hgslice")
AddPlot("Mesh", "Mesh2D")
AddPlot("Label", "hgslice")
DrawPlots()
print "The current view is:", GetView2D()
# Get an initialized 2D view object.
v = GetView2D()
v.windowCoords = (-7.67964, -3.21856, 2.66766, 7.87724)
SetView2D(v)
```

Setting the 3D view

The 3D view is much more complex than the 2D view. For information on the actual meaning of the fields in the View3DAttributes object, refer to page 214 or the VisIt User’s Manual. VisIt automatically computes a suitable view for 3D objects and it is best to initialize new View3DAttributes objects using the GetView3D function so most of the fields will already be initialized. The best way to get new views to use in a script is to interactively create the plot and repeatedly call GetView3D() after you finish rotating the plots with the mouse. You can paste the printed view information into your script and modify it slightly to create sophisticated view transitions.

```
OpenDatabase("/usr/local/visit/data/noise.silo")
AddPlot("Pseudocolor", "hardyglobal")
AddPlot("Mesh", "Mesh")
DrawPlots()
v = GetView3D()
pn = "The view is: ", v
v.viewNormal = (-0.571619, 0.405393, 0.713378)
v.viewUp = (0.308049, 0.911853, -0.271346)
SetView3D(v)
```

Flying around plots

Flying around plots is a commonly requested feature when making movies. Fortunately, this is easy to script. The basic method used for flying around plots is interpolating the view. VisIt provides a number of functions that can interpolate View2DAttributes and View3DAttributes objects. The most useful of these functions is the EvalCubicSpline function. The EvalCubicSpline function uses piece-wise cubic polynomials to smoothly interpolate between a tuple of N like items. Scripting smooth view changes using EvalCubicSpline is rather like keyframing in that you have a set of views that are mapped to some distance along the parameterized space [0., 1.]. When the parameterized space is sampled with some number of samples, VisIt calculates the view for the specified parameter value and returns a smoothly
interpolated view. One benefit over keyframing, in this case, is that you can use cubic interpolation whereas VisIt’s keyframing mode currently uses linear interpolation.

```python
# Do a pseudocolor plot of u.
OpenDatabase("/usr/local/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
DrawPlots()

# Create the control points for the views.
c0 = View3DAttributes()
c0.viewNormal = (0, 0, 1)
c0.focus = (0, 0, 0)
c0.viewUp = (0, 1, 0)
c0.viewAngle = 30
cc0.parallelScale = 17.3205
c0.nearPlane = 17.3205
c0.farPlane = 81.9615
c0.perspective = 1

c1 = View3DAttributes()
c1.viewNormal = (-0.499159, 0.475135, 0.724629)
c1.focus = (0, 0, 0)
c1.viewUp = (0.196284, 0.876524, -0.439521)
c1.viewAngle = 30
cc1.parallelScale = 14.0932
c1.nearPlane = 15.276
c1.farPlane = 69.917
c1.perspective = 1

c2 = View3DAttributes()
c2.viewNormal = (-0.522881, 0.831168, -0.189092)
c2.focus = (0, 0, 0)
c2.viewUp = (0.783763, 0.556011, 0.27671)
c2.viewAngle = 30
cc2.parallelScale = 11.3107
c2.nearPlane = 14.8914
c2.farPlane = 59.5324
c2.perspective = 1

c3 = View3DAttributes()
c3.viewNormal = (-0.438771, 0.523661, -0.730246)
c3.focus = (0, 0, 0)
c3.viewUp = (-0.0199911, 0.80676, 0.590541)
c3.viewAngle = 30
cc3.parallelScale = 8.28257
c3.nearPlane = 3.5905
c3.farPlane = 48.2315
c3.perspective = 1

c4 = View3DAttributes()
c4.viewNormal = (0.286142, -0.342802, -0.894768)
c4.focus = (0, 0, 0)
c4.viewUp = (-0.0382056, 0.928989, -0.36813)
c4.viewAngle = 30
cc4.parallelScale = 10.4152
c4.nearPlane = 1.5495
c4.farPlane = 56.1905
c4.perspective = 1
```

(continues on next page)
c5 = View3DAttributes()
c5.viewNormal = (0.974296, -0.223599, -0.0274086)
c5.focus = (0, 0, 0)
c5.viewUp = (0.222245, 0.97394, -0.0452541)
c5.viewAngle = 30
     c5.parallelScale = 1.1052
     c5.nearPlane = 24.1248
     c5.farPlane = 58.7658
     c5.perspective = 1

c6 = c0

# Create a tuple of camera values and x values. The x values
# determine where in [0,1] the control points occur.
cpts = (c0, c1, c2, c3, c4, c5, c6)
x=[]
for i in range(7):
    x = x + [float(i) / float(6.)]

# Animate the view using EvalCubicSpline.
nsteps = 100
for i in range(nsteps):
    t = float(i) / float(nsteps - 1)
    c = EvalCubicSpline(t, x, cpts)
    c.nearPlane = -34.461
    c.farPlane = 34.461
    SetView3D(c)

2.3.10 Working with annotations

Adding annotations to your visualization improve the quality of the final visualization in that you can refine the colors
that you use, add logos, or highlight features of interest in your plots. This section provides some recipes for creating
annotations using scripting.

Using gradient background colors

VisIt’s default white background is not necessarily the best looking background color for presentations. Adding a
gradient background under your plots is an easy way to add a small professional touch to your visualizations. VisIt
provides a few different styles of gradient background: radial, top to bottom, bottom to top, left to right, and right
to left. The gradient style is set using the gradientBackgroundStyle member of the AnnotationAttributes object. The
before and after results are shown in Figure [fig:annotations1].

# Set a blue/black, radial, gradient background.
a = AnnotationAttributes()
a.backgroundMode = a.Gradient
a.gradientBackgroundStyle = a.Radial
a.gradientColor1 = (0,0,255,255) # Blue
a.gradientColor2 = (0,0,0,255) # Black
SetAnnotationAttributes(a)
Adding a banner

Banners are useful for providing titles for a visualization or for marking its content (see Figure [fig:annotations2]). To add an “Unclassified” banner to a visualization, use the following bit of Python code:

```python
# Create a text object that we’ll use to indicate that our
# visualization is unclassified.
banner = CreateAnnotationObject("Text2D")
banner.text = "Unclassified"
banner.position = (0.37, 0.95)
banner.fontBold = 1
# print the attributes that you can set in the banner object.
print banner
```

Adding a time slider

Time sliders are important annotations for movies since they convey how much progress an animation has made as well as how many more frames have yet to be seen. The time slider is also important for showing the simulation time as the animation progresses so users can get a sense of when in the simulation important events occur. VisIt’s time slider annotation object is shown in Figure [fig:annotations3].

```python
# Add a time slider in the lower left corner
slider = CreateAnnotationObject("TimeSlider")
slider.height = 0.07
# Print the options that are available in the time slider object
print slider
```

Adding a logo

Adding a logo to a visualization is an important part of project identification for movies and other visualizations created with VisIt. If you have a logo image file stored in TIFF, JPEG, BMP, or PPM format then you can use it with

---

Fig. 2.2: Before and after image of adding a gradient background.
Fig. 2.3: Adding a banner

Fig. 2.4: Time slider annotation in the lower left corner
VisIt as an image annotation (see Figure \textit{fig:annotations4}). Note that this approach can also be used to insert images of graphs, plots, portraits, diagrams, or any other form of image data into a visualization.

```python
# Incorporate LLNL logo image (llnl.jpeg) as an annotation
image = CreateAnnotationObject("Image")
image.image = "llnl.jpeg"
image.position = (0.02, 0.02)
# Print the other image annotation options
print image
```

Fig. 2.5: Image annotation used to incorporate LLNL logo

## 2.4 Functions

Many functions return an integer where 1 means success and 0 means failure. This behavior is represented by the type CLI\_return\_t in an attempt to distinguish it from functions that may utilize the full range of integers.

### 2.4.1 ActivateDatabase

**Synopsis:**

```python
ActivateDatabase(argument) \rightarrow integer
```

**argument** [string] The name of the database to be activated.

**return type** [CLI\_return\_t] ActivateDatabase returns 1 on success and 0 on failure.

**Description:**

The ActivateDatabase function is used to set the active database to a database that has been previously opened. The ActivateDatabase function only works when you are using it to activate a database that you have previously opened. You do not need to use this function unless you frequently toggle between more than one database when making plots or changing time states. While the OpenDatabase function can also
be used to set the active database, the ActivateDatabase function does not have any side effects that would cause the time state for the new active database to be changed.

Example:

```python
#!/usr/bin/env python
import visit
visit.AddArgument("-nowin")
```
AddOperator(operator) -> integer
AddOperator(operator, all) -> integer

operator [string] The name of the operator to be applied.
all [integer] This is an optional integer argument that applies the operator to all plots if the value of the argument is not zero.

return type [CLI_return_t] The AddOperator function returns an integer value of 1 for success and 0 for failure.

Description:
The AddOperator function adds a VisIt operator to the active plots. The operator argument is a string containing the name of the operator to be added to the active plots. The operator name must be a valid operator plugin name that is a member of the tuple returned by the OperatorPlugins function. The all argument is an integer that determines whether or not the operator is applied to all plots. If the all argument is not provided, the operator is only added to active plots. Once the AddOperator function is called, the desired operator is added to all active plots unless the all argument is a non-zero value. When the all argument is a non-zero value, the operator is applied to all plots regardless of whether or not they are selected. Operator attributes are set through the SetOperatorOptions function.

Example:
```bash
#!/usr/bin/env python
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
AddPlot("Mesh", "mesh1")
AddOperator("Slice", 1) # Slice both plots
DrawPlots()
```

2.4.5 AddPlot

Synopsis:
AddPlot(plotType, variableName) -> integer
AddPlot(plotType, variableName, inheritSIL) -> integer
AddPlot(plotType, variableName, inheritSIL, applyOperators) -> integer

plotType [string] The name of a valid plot plugin type.
variableName [string] A valid variable name for the open database.
inheritSIL [integer] An integer flag indicating whether the plot should inherit the active plot’s SIL restriction.
applyOperators [integer] An integer flag indicating whether the operators from the active plot should be applied to the new plot.

return type [CLI_return_t] The AddPlot function returns an integer value of 1 for success and 0 for failure.

Description:
The AddPlot function creates a new plot of the specified type using a variable from the open database. The plotType argument is a string that contains the name of a valid plot plugin type which must be a member of the string tuple that is returned by the PlotPlugins function. The variableName argument is a string that contains the name of a variable in the open database. After the AddPlot function is called, a new plot is created and it is made the sole active plot.

Example:
2.4.6 AddWindow

Synopsis:

AddWindow()

Description:

The AddWindow function creates a new visualization window and makes it the active window. This function can be used to create up to 16 visualization windows. After that, the AddWindow function has no effect.

Example:

```python
import visit
visit.Launch()
visit.AddWindow()  # Create window #2
visit.AddWindow()  # Create window #3
```

2.4.7 AlterDatabaseCorrelation

Synopsis:

AlterDatabaseCorrelation(name, databases, method) -> integer

name [string] The name of the database correlation to be altered.

databases [tuple or list of strings] The databases argument must be a tuple or list of strings containing the fully qualified database names to be used in the database correlation.

method [integer] The method argument must be an integer in the range [0,3].

<table>
<thead>
<tr>
<th>Correlation method</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>IndexForIndexCorrelation</td>
<td>0</td>
</tr>
<tr>
<td>StretchedIndexCorrelation</td>
<td>1</td>
</tr>
<tr>
<td>TimeCorrelation</td>
<td>2</td>
</tr>
<tr>
<td>CycleCorrelation</td>
<td>3</td>
</tr>
</tbody>
</table>

return type [CLI_return_t] The AlterDatabaseCorrelation function returns 1 on success and 0 on failure.

Description:

The AlterDatabaseCorrelation method alters an existing database correlation. A database correlation is a VisIt construct that relates the time states for two or more databases in some way. You would use the AlterDatabaseCorrelation function if you wanted to change the list of databases used in a database correlation or if you wanted to change how the databases are related - the correlation method. The name argument is a string that is the name of the database correlation to be altered. If the name that you pass is not a valid database correlation then the AlterDatabaseCorrelation function fails. The databases argument is a list or tuple of string objects containing the fully-qualified (host:/path/filename) names of the databases.
to be involved in the database query. The method argument allows you to specify a database correlation method.

Example:

```python
dbs = ("/usr/gapps/visit/data/wave.visit", "/usr/gapps/visit/data/wave*.silo database -")
OpenDatabase(dbs[0])
AddPlot("Pseudocolor", "pressure")
OpenDatabase(dbs[1])
AddPlot("Pseudocolor", "d")
# VisIt created an index for index database correlation but we
# want a cycle correlation.
AlterDatabaseCorrelation("Correlation01", dbs, 3)
```

### 2.4.8 ApplyNamedSelection

**Synopsis:**

`ApplyNamedSelection(name) -> integer`

- **name** [string] The name of a named selection. (This should have been previously created with a CreateNamedSelection call.)
- **return type** [CLI_return_t] The ApplyNamedSelection function returns 1 for success and 0 for failure.

**Description:**

Named Selections allow you to select a group of elements (or particles). One typically creates a named selection from a group of elements and then later applies the named selection to another plot (thus reducing the set of elements displayed to the ones from when the named selection was created).

**Example:**

```python
#% visit -cli
db = "/usr/gapps/visit/data/wave*.silo database"
OpenDatabase(db)
AddPlot("Pseudocolor", "pressure")
AddOperator("Clip")
c = ClipAttributes()
c.plane1Origin = (0,0.6,0)
c.plane1Normal = (0,-1,0)
SetOperatorOption(c)
DrawPlots()
CreateNamedSelection("els_above_at_time_0")
SetTimeSliderState(40)
RemoveLastOperator()
ApplyNamedSelection("els_above_at_time_0")
```

### 2.4.9 ChangeActivePlotsVar

**Synopsis:**

`ChangeActivePlotsVar(variableName) -> integer`

- **variableName** [string] The name of the new plot variable.

2.4. Functions 447
return type [CLI_return_t] The `ChangeActivePlotsVar` function returns an integer value of 1 for success and 0 for failure.

Description:
The `ChangeActivePlotsVar` function changes the plotted variable for the active plots. This is a useful way to change what is being visualized without having to delete and recreate the current plots. The `variableName` argument is a string that contains the name of a variable in the open database.

Example:
```bash
## visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
DrawPlots()
SaveWindow()
ChangeActivePlotsVar("v")
```

### 2.4.10 CheckForNewStates

Synopsis:
```
CheckForNewStates(name) -> integer
```

name [string] The name of a database that has been opened previously.

return type [CLI_return_t] The `CheckForNewStates` function returns 1 for success and 0 for failure.

Description:
Calculations are often run at the same time as some of the preliminary visualization work is being performed. That said, you might be visualizing the leading time states of a database that is still being created. If you want to force VisIt to add any new time states that were added since you opened the database, you can use the `CheckForNewStates` function. The `name` argument must contain the name of a database that has been opened before.

Example:
```bash
## visit -cli
db = "/usr/gapps/visit/data/wave*.silo database"
OpenDatabase(db)
AddPlot("Pseudocolor", "pressure")
DrawPlots()
SetTimeSliderState(TimeSliderGetNStates() - 1)
# More files appear on disk
CheckForNewStates(db)
SetTimeSliderState(TimeSliderGetNStates() - 1)
```

### 2.4.11 ChooseCenterOfRotation

Synopsis:
```
ChooseCenterOfRotation() -> integer
ChooseCenterOfRotation(screenX, screenY) -> integer
```

screenX [double] A double that is the X coordinate of the pick point in normalized [0,1] screen space.

screenY [double] A double that is the Y coordinate of the pick point in normalized [0,1] screen space.
return type [CLI_return_t] The ChooseCenterOfRotation function returns 1 if successful and 0 if it fails.

Description:
The ChooseCenterOfRotation function allows you to pick a new center of rotation, which is the point about which plots are rotated when you interactively rotate plots. The function can either take zero arguments, in which case you must interactively pick on plots, or it can take two arguments that correspond to the X and Y coordinates of the desired pick point in normalized screen space. When using the two argument version of the ChooseCenterOfRotation function, the X and Y values are floating point values in the range [0,1]. If the ChooseCenterOfRotation function is able to actually pick on plots, yes there must be plots in the vis window, then the center of rotation is updated and the new value is printed to the console.

Example:
```
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlots("Pseudocolor", "u")
DrawPlots()
# Interactively choose the center of rotation
ChooseCenterOfRotation()
# Choose a center of rotation using normalized screen coordinates and print the value.
ResetView()
ChooseCenterOfRotation(0.5, 0.3)
print("The new center of rotation is:\ \{}\ ".format(GetView3D().centerOfRotation))
```

2.4.12 ClearAllWindows

Synopsis:
ClearAllWindows() -> integer

return type [CLI_return_t] 1 on success, 0 on failure.

Description:
The ClearAllWindows function is used to clear out the plots from the active visualization window. The plots are removed from the visualization window but are left in the plot list so that subsequent calls to the DrawPlots function regenerate the plots in the plot list. The ClearAllWindows function preforms the same work as the ClearWindow function except that all windows are cleared of their plots.

Example:
```
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
DrawPlots()
SetActiveWindow(2)  # Make window 2 active
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Subset", "mat1")
DrawPlots()
ClearWindow()  # Clear the plots in window 2.
DrawPlots()  # Redraw the plots in window 2.
ClearAllWindows()  # Clear the plots from all windows.
```
2.4.13 ClearCache

Synopsis:

ClearCache(host) -> integer
ClearCache(host, simulation) -> integer

host  [string] The name of the computer where the compute engine is running.
simulation  [string] The name of the simulation being processed by the compute engine.

return type  [CLI_return_t] 1 on success and 0 on failure.

Description:

Sometimes during extended VisIt runs, you might want to periodically clear the compute engine’s network cache to reduce the amount of memory being used by the compute engine. Clearing the network cache is also useful when you want to change what the compute engine is working on. For example, you might process a large database and then decide to process another large database. Clearing the network cache beforehand will free up more resources for the compute engine so it can more efficiently process the new database. The host argument is a string object containing the name of the computer on which the compute engine is running. The simulation argument is optional and only applies to when you want to instruct a simulation that is acting as a VisIt compute engine to clear its network cache. If you want to tell more than one compute engine to clear its cache without having to call ClearCache multiple times, you can use the ClearCacheForAllEngines function.

Example:

```%visit -cli
OpenDatabase("localhost:very_large_database")
# Do a lot of work
ClearCache("localhost")
OpenDatabase("localhost:another_large_database")
# Do more work
OpenDatabase("remotehost:yet_another_database")
# Do more work
ClearCacheForAllEngines()
```

2.4.14 ClearCacheForAllEngines

Synopsis:

ClearCacheForAllEngines() -> integer

return type  [CLI_return_t] 1 on success and 0 on failure.

Description:

Sometimes during extended VisIt runs, you might want to periodically clear the compute engine’s network cache to reduce the amount of memory being used by the compute engine. Clearing the network cache is also useful when you want to change what the compute engine is working on. For example, you might process a large database and then decide to process another large database. Clearing the network cache beforehand will free up more resources for the compute engine so it can more efficiently process the new database. The host argument is a string object containing the name of the computer on which the compute engine is running. The simulation argument is optional and only applies to when you want to instruct a simulation that is acting as a VisIt compute engine to clear its network cache. If you want to tell more than one compute engine to clear its cache without having to call ClearCache multiple times, you can use the ClearCacheForAllEngines function.
Example:

```csharp
#visit -cli
OpenDatabase("localhost:very_large_database")
# Do a lot of work
ClearCache("localhost")
OpenDatabase("localhost:another_large_database")
# Do more work
OpenDatabase("remotehost:yet_another_database")
# Do more work
ClearCacheForAllEngines()
```

### 2.4.15 ClearMacros

**Synopsis:**

ClearMacros()

**Description:**

The ClearMacros function clears out the list of registered macros and sends a message to the gui to clear the buttons from the Macros window.

**Example:**

ClearMacros()

### 2.4.16 ClearPickPoints

**Synopsis:**

ClearPickPoints()

**Description:**

The ClearPickPoints function removes pick points from the active visualization window. Pick points are the letters that are added to the visualization window where the mouse is clicked when the visualization window is in pick mode.

**Example:**

```csharp
# visit -cli
# Put the visualization window into pick mode using the popup menu and add some pick points.
# Clear the pick points.
ClearPickPoints()
```

### 2.4.17 ClearReferenceLines

**Synopsis:**

ClearReferenceLines()

**Description:**
The ClearReferenceLines function removes reference lines from the active visualization window. Reference lines are the lines that are drawn on a plot to show where you have performed lineouts.

Example:

```bash
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/curv2d.silo")
AddPlot("Pseudocolor", "d")
Lineout((-3.0, 2.0), (2.0, 4.0), ("default", "u", "v"))
ClearReferenceLines()
```

### 2.4.18 ClearViewKeyframes

**Synopsis:**

ClearViewKeyframes() -> integer

**Return type:** [CLI_return_t] The ClearViewKeyframes function returns 1 on success and 0 on failure.

**Description:**

The ClearViewKeyframes function clears any view keyframes that may have been set. View keyframes are used to create complex view behavior such as fly-throughs when VisIt is in keyframing mode.

Example:

```bash
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
k = KeyframeAttributes()
k.enabled, k.nFrames, k.nFramesWasUserSet = 1,10,1
SetKeyframeAttributes(k)
DrawPlots()
SetViewKeyframe()
v1 = GetView3D()
v1.viewNormal = (-0.66609, 0.337227, 0.665283)
v1.viewUp = (0.157431, 0.935425, -0.316537)
SetView3D(v1)
SetTimeSliderState(9)
SetViewKeyframe()
ToggleCameraViewMode()
for i in range(10):
    SetTimeSliderState(i)
ClearViewKeyframes()
```

### 2.4.19 ClearWindow

**Synopsis:**

ClearWindow() -> integer

**Return type:** [CLI_return_t] 1 on success, 0 on failure.

**Description:**

The ClearWindow function is used to clear out the plots from the active visualization window. The plots are removed from the visualization window but are left in the plot list so that subsequent calls to the
The DrawPlots function regenerate the plots in the plot list. The ClearAllWindows function performs the same work as the ClearWindow function except that all windows are cleared of their plots.

Example:

```c
# visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
DrawPlots()
AddWindow()
SetActiveWindow(2) # Make window 2 active
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Subset", "mat1")
DrawPlots()
ClearWindow() # Clear the plots in window 2.
DrawPlots() # Redraw the plots in window 2.
ClearAllWindows() # Clear the plots from all windows.
```

### 2.4.20 CloneWindow

**Synopsis:**

CloneWindow() -> integer

**return type** [CLI_return_t] The CloneWindow function returns an integer value of 1 for success and 0 for failure.

**Description:**

The CloneWindow function tells the viewer to create a new window, based on the active window, that contains the same plots, annotations, lights, and view as the active window. This function is useful for when you have a window set up like you want and then want to do the same thing in another window using a different database. You can first clone the window and then replace the database.

**Example:**

```c
# visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
DrawPlots()
v = ViewAttributes()
v.camera = (-0.505893, 0.32034, 0.800909)
v.viewUp = (0.1314, 0.946269, -0.295482)
v.parallelScale = 14.5472
v.nearPlane = -34.641
v.farPlane = 34.641
v.perspective = 1
SetView3D() # Set the view
a = AnnotationAttributes()
a.backgroundColor = (0, 0, 255, 255)
SetAnnotationAttributes(a) # Set the annotation properties
CloneWindow() # Create a clone of the active window
DrawPlots() # Make the new window draw its plots
```

### 2.4.21 Close

**Synopsis:**

2.4. Functions
Close()

**Description:**

The Close function terminates VisIt’s viewer. This is useful for Python scripts that only need access to VisIt’s capabilities for a short time before closing VisIt.

**Example:**

```python
import visit
visit.Launch()
visit.Close()  # Close the viewer
```

### 2.4.22 CloseComputeEngine

**Synopsis:**

CloseComputeEngine() -> integer  
CloseComputeEngine(hostName) -> integer  
CloseComputeEngine(hostName, simulation) -> integer

- **hostName** [string] Optional name of the computer on which the compute engine is running.  
- **simulation** [string] Optional name of a simulation.  
- **return type** [CLI_return_t] The CloseComputeEngine function returns an integer value of 1 for success and 0 for failure.

**Description:**

The CloseComputeEngine function tells the viewer to close the compute engine running a specified host. The hostName argument is a string that contains the name of the computer where the compute engine is running. The hostName argument can also be the name “localhost” if you want to close the compute engine on the local machine without having to specify its name. It is not necessary to provide the hostName argument. If the argument is omitted, the first compute engine in the engine list will be closed. The simulation argument can be provided if you want to close a connection to a simulation that is acting as a VisIt compute engine. A compute engine can be launched again by creating a plot or by calling the OpenComputeEngine function.

**Example:**

```python
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")  # Launches an engine
AddPlot("Pseudocolor", "u")
DrawPlots()
CloseComputeEngine()  # Close the compute engine
```

### 2.4.23 CloseDatabase

**Synopsis:**

CloseDatabase(name) -> integer

- **name** [string] The name of the database to close.  
- **return type** [CLI_return_t] The CloseDatabase function returns 1 on success and 0 on failure.
Description:

The CloseDatabase function is used to close a specified database and free all resources that were devoted to keeping the database open. This function has an effect similar to ClearCache but it does more in that in addition to clearing the compute engine’s cache, which it only does for the specified database, it also removes all references to the specified database from tables of cached metadata, etc. Note that the CloseDatabase function will fail and the database will not be closed if any plots reference the specified database.

Example:

```python
#$ visit -cli
db = "\usr\gapps\visit\data\globe.silo"
OpenDatabase(db)
AddPlot("Pseudocolor", "u")
DrawPlots()
print("This won’t work: retval = \$d" % CloseDatabase(db))
DeleteAllPlots()
print("Now it works: retval = \$d" % CloseDatabase(db))
```

2.4.24 ColorTableNames

Synopsis:

ColorTableNames() -> tuple

return type [tuple] The ColorTableNames function returns a tuple of strings containing the names of the color tables that have been defined.

Description:

The ColorTableNames function returns a tuple of strings containing the names of the color tables that have been defined. This method can be used in case you want to iterate over several color tables.

Example:

```python
#$ visit -cli
OpenDatabase("\usr\gapps\visit\data\curv2d.silo")
AddPlot("Pseudocolor", "u")
DrawPlots()
for ct in ColorTableNames():
    p = PseudocolorAttributes()
    p.colorTableName = ct
    SetPlotOptions(p)
```

2.4.25 ConstructDataBinning

Synopsis:

ConstructDataBinning(options) -> integer

options [ConstructDataBinningAttributes object] An object of type ConstructDataBinningAttributes. This object specifies the options for constructing a data binning.

return type [CLI_return_t] Returns 1 on success, 0 on failure.

Description:
The ConstructDataBinning function creates a data binning function for the active plot. Data Binnings place data from a data set into bins and reduce that data. They are used to either be incorporated with expressions to make new derived quantities or to be directly visualized.

Example:

```python
# visit -cli
OpenDatabase("/usr/gapps/visit/data/curv3d.silo")
AddPlot("Pseudocolor", "d")
DrawPlots()
# Set the construct data binning attributes.
i = ConstructDataBinningAttributes()
i.name = "db1"
i.binningScheme = i.Uniform
i.varnames = ("u", "w")
i.binBoundaries = (-1, 1, -1, 1) # minu, maxu, minw, maxw
i.numSamples = (25, 25)
i.reductionOperator = i.Average
i.varForReductionOperator = "v"
ConstructDataBinning(i)
# Example of binning using spatial coordinates
i.varnames = ("X", "u") # X is added as a placeholder to maintain indexing
i.binType = (1, 0) # 1 = X, 2 = Y, 3 = Z, 0 = variable
```

### 2.4.26 CopyAnnotationsToWindow

**Synopsis:**

CopyAnnotationsToWindow(source, dest) -> integer

**source** [integer] The index (an integer from 1 to 16) of the source window.

**dest** [integer] The index (an integer from 1 to 16) of the destination window.

**return type** [CLI_return_t] 1 for success and 0 for failure.

**Description:**

The Copy functions copy attributes from one visualization window to another visualization window. The CopyAnnotationsToWindow function copies the annotations from a source visualization window to a destination visualization window.

**Example:**

```python
# visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
DrawPlots()
AddWindow()
SetActiveWindow(2)
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Mesh", "mesh1")
# Copy window 1's Pseudocolor plot to window 2.
CopyPlotsToWindow(1, 2)
DrawPlots() # Window 2 will have 2 plots
# Spin the plots around in window 2 using the mouse.
CopyViewToWindow(2, 1) # Copy window 2's view to window 1.
```
2.4.27 CopyLightingToWindow

Synopsis:

CopyLightingToWindow(source, dest) -> integer

**source** [integer] The index (an integer from 1 to 16) of the source window.

**dest** [integer] The index (an integer from 1 to 16) of the destination window.

**return type** [CLI_return_t] 1 for success and 0 for failure.

Description:

The Copy functions copy attributes from one visualization window to another visualization window. The CopyLightingAttributes function copies lighting.

Example:

```bash
#!/usr/bin/env python

OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
DrawPlots()
AddWindow()
SetActiveWindow(2)
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Mesh", "mesh1")
# Copy window 1's Pseudocolor plot to window 2.
CopyPlotsToWindow(1, 2)
DrawPlots() # Window 2 will have 2 plots
# Spin the plots around in window 2 using the mouse.
CopyViewToWindow(2, 1) # Copy window 2's view to window 1.
```

2.4.28 CopyPlotsToWindow

Synopsis:

CopyPlotsToWindow(source, dest) -> integer

**source** [integer] The index (an integer from 1 to 16) of the source window.

**dest** [integer] The index (an integer from 1 to 16) of the destination window.

**return type** [CLI_return_t] 1 for success and 0 for failure.

Description:

The Copy functions copy attributes from one visualization window to another visualization window. The CopyPlotsToWindow function copies the plots from one visualization window to another visualization window but does not also force plots to generate so after copying plots with the CopyPlotsToWindow function, you should also call the DrawPlots function.

Example:

```bash
#!/usr/bin/env python

OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
DrawPlots()
AddWindow()
```

(continues on next page)
SetActiveWindow(2)
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Mesh", "mesh1")
# Copy window 1’s Pseudocolor plot to window 2.
CopyPlotsToWindow(1, 2)
DrawPlots() # Window 2 will have 2 plots
# Spin the plots around in window 2 using the mouse.
CopyViewToWindow(2, 1) # Copy window 2’s view to window 1.

### 2.4.29 CopyViewToWindow

**Synopsis:**

`CopyViewToWindow(source, dest) -> integer`

- **source** [integer] The index (an integer from 1 to 16) of the source window.
- **dest** [integer] The index (an integer from 1 to 16) of the destination window.
- **return type** [CLL_return_t] The Copy functions return an integer value of 1 for success and 0 for failure.

**Description:**

The Copy functions copy attributes from one visualization window to another visualization window. The `CopyViewToWindow` function copies the view.

**Example:**

```python
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
DrawPlots()
AddWindow()
SetActiveWindow(2)
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Mesh", "mesh1")
# Copy window 1’s Pseudocolor plot to window 2.
CopyPlotsToWindow(1, 2)
CopyViewToWindow(2, 1) # Copy window 2’s view to window 1.
```

### 2.4.30 CreateAnnotationObject

**Synopsis:**

`CreateAnnotationObject(annotType[,annotName,visibleFlag]) -> annotation object`

- **annotType** [string] The name of the type of annotation object to create.
**Annotation type**

<table>
<thead>
<tr>
<th>Annotation type</th>
<th>String</th>
</tr>
</thead>
<tbody>
<tr>
<td>2D text annotation</td>
<td>Text2D</td>
</tr>
<tr>
<td>3D text annotation</td>
<td>Text3D</td>
</tr>
<tr>
<td>Time slider annotation</td>
<td>TimeSlider</td>
</tr>
<tr>
<td>Image annotation</td>
<td>Image</td>
</tr>
<tr>
<td>Line/arrow annotation</td>
<td>Line2D</td>
</tr>
</tbody>
</table>

**annotName** [string] A user-defined name of the annotation object to create. By default, VisIt creates names like ‘newObject0’, ‘newObject1’, ….

**visibleFlag** [integer] An optional integer to indicate if the annotation object should be created with initial visibility on or off. Pass 0 for off and non-zero for on. By default, VisIt creates annotation objects with visibility on. If you wish only to pass the visibleFlag argument, there is no need to also pass the annotName argument.

**return type** [annotation object] CreateAnnotationObject is a factory function that creates annotation objects of different types. The return value, if a valid annotation type is provided, is an annotation object. If the function fails, VisItException is raised.

**Description:**

CreateAnnotationObject is a factory function that creates different kinds of annotation objects. The annotType argument is a string containing the name of the type of annotation object to create. Each type of annotation object has different properties that can be set. Setting the different properties of an Annotation objects directly modifies annotations in the vis window. Currently there are 5 types of annotation objects:

**Example:**

```python
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/wave.visit", 17)
AddPlot("Pseudocolor", "pressure")
DrawPlots()
slider = CreateAnnotationObject("TimeSlider")
print(slider)
slider.startColor = (255,0,0,255)
slider.endColor = (255,255,0,255)
```

### 2.4.31 CreateDatabaseCorrelation

**Synopsis:**

CreateDatabaseCorrelation(name, databases, method) -> integer

**name** [string] The name of the database correlation to be created.

**databases** [tuple or list of strings] Tuple or list of strings containing the names of the databases to involve in the database correlation.

**method** [integer] An integer in the range [0,3] that determines the correlation method.

<table>
<thead>
<tr>
<th>Correlation method</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>IndexForIndexCorrelation</td>
<td>0</td>
</tr>
<tr>
<td>StretchedIndexCorrelation</td>
<td>1</td>
</tr>
<tr>
<td>TimeCorrelation</td>
<td>2</td>
</tr>
<tr>
<td>CycleCorrelation</td>
<td>3</td>
</tr>
</tbody>
</table>

**return type** [CLI_return_t] The CreateDatabaseCorrelation function returns 1 on success and 0 on failure.
Description:

The CreateDatabaseCorrelation function creates a database correlation, which is a VisIt construct that relates the time states for two or more databases in some way. You would use the CreateDatabaseCorrelation function if you wanted to put plots from more than one time-varying database in the same Vis window and then move them both through time in some synchronized way. The name argument is a string that is the name of the database correlation to be created. You will use the name of the database correlation to set the active time slider later so that you can change time states. The databases argument is a list or tuple of string objects containing the fully-qualified (host/path/filename) names of the databases to be involved in the database query. The method argument allows you to specify a database correlation method. Each database correlation has its own time slider that can be used to set the time state of databases that are part of a database correlation. Individual time-varying databases have their own trivial database correlation, consisting of only 1 database. When you call the CreateDatabaseCorrelation function, VisIt creates a new time slider with the same name as the database correlation and makes it be the active time slider.

Example:

```python
# visit -cli
dbs = ('/usr/gapps/visit/data/dbA00.pdb',
       '/usr/gapps/visit/data/dbB00.pdb')
OpenDatabase(dbs[0])
AddPlot("FilledBoundary", "material(mesh)")
OpenDatabase(dbs[1])
AddPlot("FilledBoundary", "material(mesh)")
DrawPlots()
CreateDatabaseCorrelation("common", dbs, 1)
# Creating a new database correlation also creates a new time slider and makes it be active.
w = GetWindowInformation()
print("Active time slider: %s" % w.timeSliders[w.activeTimeSlider])
# Animate through time using the "common" database correlation's time slider.
for i in range(TimeSliderGetNStates()):
    SetTimeSliderState(i)
```

2.4.32 CreateNamedSelection

Synopsis:

CreateNamedSelection(name) -> integer
CreateNamedSelection(name, properties) -> integer

name [string] The name of a named selection.

properties [SelectionProperties object] This optional argument lets you pass a SelectionProperties object containing the properties that will be used to create the named selection. When this argument is omitted, the named selection will always be associated with the active plot. You can use this argument to set up more complex named selections that may be associated with plots or databases.

return type [CLI_return_t] The CreateNamedSelection function returns 1 for success and 0 for failure.

Description:

Named Selections allow you to select a group of elements (or particles). One typically creates a named selection from a group of elements and then later applies the named selection to another plot (thus reducing the set of elements displayed to the ones from when the named selection was created).

Example:
## 2.4.33 DatabasePlugins

**Synopsis:**

DatabasePlugins() -> dictionary  
DatabasePlugins(host) -> dictionary

**host** [string] The name of the host for which we want database plugins.  
**return type** [dictionary] The DatabasePlugins functions returns a dictionary.

**Description:**

The DatabasePlugins function returns a dictionary containing the names of the database plugins for the specified host. If no host is given, localhost is assumed. The dictionary contains two keys: “host” and “plugins”.

**Example:**

```bash
## visit -cli  
dbp = DatabasePlugins("localhost")  
print(dbp["host"])  
print(dbp["plugins"])```

## 2.4.34 DeIconifyAllWindows

**Synopsis:**

DeIconifyAllWindows()

**Description:**

The DeIconifyAllWindows function unhides all of the hidden visualization windows. This function is usually called after IconifyAllWindows as a way of making all of the hidden visualization windows visible.

**Example:**

```bash
## visit -cli  
SetWindowLayout(4) # Have 4 windows```
2.4.35 DefineArrayExpression

Synopsis:

```python
DefineArrayExpression(variableName, expression) -> integer
```

**variableName**  [string] The name of the variable to be created.

**expression**  [string] The expression definition as a string.

**return type**  [CLI_return_t] The DefineExpression functions return 1 on success and 0 on failure.

**Description:**

DefineArrayExpression creates new array variables. Expression variables can be plotted like any other variable. The variableName argument is a string that contains the name of the new variable. You can pass the name of an existing expression if you want to provide a new expression definition. The expression argument is a string that contains the definition of the new variable in terms of math operators and pre-existing variable names. Reference the VisIt User’s Manual if you want more information on creating expressions, such as expression syntax, or a list of built-in expression functions.

**Example:**

```python
# visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
DefineScalarExpression("myvar", "sin(u) + cos(w)")
# Plot the scalar expression variable.
AddPlot("Pseudocolor", "myvar")
DrawPlots()
# Plot a vector expression variable.
DefineVectorExpression("myvec", "{u,v,w}\")
AddPlot("Vector", "myvec")
DrawPlots()
```

2.4.36 DefineCurveExpression

Synopsis:

```python
DefineCurveExpression(variableName, expression) -> integer
```

**variableName**  [string] The name of the variable to be created.

**expression**  [string] The expression definition as a string.

**return type**  [CLI_return_t] The DefineExpression functions return 1 on success and 0 on failure.

**Description:**

DefineCurveExpression creates new curve variables. Expression variables can be plotted like any other variable. The variableName argument is a string that contains the name of the new variable. You can pass the name of an existing expression if you want to provide a new expression definition. The expression argument is a string that contains the definition of the new variable in terms of math operators and pre-existing variable names. Reference the VisIt User’s Manual if you want more information on creating expressions, such as expression syntax, or a list of built-in expression functions.
argument is a string that contains the definition of the new variable in terms of math operators and pre-existing variable names. Reference the VisIt User’s Manual if you want more information on creating expressions, such as expression syntax, or a list of built-in expression functions.

Example:

```bash
$ visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
DefineScalarExpression("myvar", "sin(u) + cos(w)")
# Plot the scalar expression variable.
AddPlot("Pseudocolor", "myvar")
DrawPlots()
# Plot a vector expression variable.
DefineVectorExpression("myvec", "{u,v,w}\")
AddPlot("Vector", "myvec")
DrawPlots()
```

### 2.4.37 DefineMaterialExpression

**Synopsis:**

`DefineMaterialExpression(variableName, expression) -> integer`

- **variableName** [string] The name of the variable to be created.
- **expression** [string] The expression definition as a string.
- **return type** [CLI_return_t] The DefineExpression functions return 1 on success and 0 on failure.

**Description:**

The DefineMaterialExpression function creates new material variables. Expression variables can be plotted like any other variable. The variableName argument is a string that contains the name of the new variable. You can pass the name of an existing expression if you want to provide a new expression definition. The expression argument is a string that contains the definition of the new variable in terms of math operators and pre-existing variable names. Reference the VisIt User’s Manual if you want more information on creating expressions, such as expression syntax, or a list of built-in expression functions.

Example:

```bash
$ visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
DefineScalarExpression("myvar", "sin(u) + cos(w)")
# Plot the scalar expression variable.
AddPlot("Pseudocolor", "myvar")
DrawPlots()
# Plot a vector expression variable.
DefineVectorExpression("myvec", "{u,v,w}\")
AddPlot("Vector", "myvec")
DrawPlots()
```

### 2.4.38 DefineMeshExpression

**Synopsis:**

`DefineMeshExpression(variableName, expression) -> integer`

2.4. Functions 463
variableName [string] The name of the variable to be created.

expression [string] The expression definition as a string.

return type [CLI_return_t] The DefineExpression functions return 1 on success and 0 on failure.

Description:

The DefineMeshExpression creates new mesh variables. Expression variables can be plotted like any other variable. The variableName argument is a string that contains the name of the new variable. You can pass the name of an existing expression if you want to provide a new expression definition. The expression argument is a string that contains the definition of the new variable in terms of math operators and pre-existing variable names Reference the VisIt User’s Manual if you want more information on creating expressions, such as expression syntax, or a list of built-in expression functions.

Example:

```python
#!/usr/bin/env python
from visit import *

dg = OpenDatabase("/path/to/dataset.silo")
define_scalar_expression("myvar", "sin(u) + cos(w)")
plot("Pseudocolor", "myvar")
draw() # Draw plots.
```

2.4.39 DefinePythonExpression

Synopsis:

```
DefinePythonExpression(myvar, args, source)
DefinePythonExpression(myvar, args, source, type)
DefinePythonExpression(myvar, args, file)
```

myvar [string] The name of the variable to be created.

args [tuple] A tuple (or list) of strings providing the variable names of the arguments to the Python Expression.

source [string] A string containing the source code for a Python Expression Filter.

file [string] A string containing the path to a Python Expression Filter script file.

type [string] An optional string defining the output type of the expression. Default type - 'scalar' Available types - 'scalar', 'vector', 'tensor', 'array', 'curve' Note - Use only one of the 'source' or 'file' arguments. If both are used the 'source' argument overrides 'file'.

Description:

Used to define a Python Filter Expression.

2.4.40 DefineScalarExpression

Synopsis:

```
DefineScalarExpression(variableName, expression) -> integer
```

variableName [string] The name of the variable to be created.
expression [string] The expression definition as a string.

return type [CLI_return_t] The DefineExpression functions return 1 on success and 0 on failure.

Description:
The DefineScalarExpression function creates a new scalar variable based on other variables from the open database. Expression variables can be plotted like any other variable. The variableName argument is a string that contains the name of the new variable. You can pass the name of an existing expression if you want to provide a new expression definition. The expression argument is a string that contains the definition of the new variable in terms of math operators and pre-existing variable names Reference the VisIt User's Manual if you want more information on creating expressions, such as expression syntax, or a list of built-in expression functions.

Example:
```c
#!/usr/bin/env python
import visit
vis = visit.VisIt()
vis.OpenDatabase("/path/to/your/file.silo")
vis.DefineScalarExpression("new_var", "sin(u) + cos(w)")
vis.AddPlot("Pseudocolor", "new_var")
vis.DrawPlots()
```

2.4.41 DefineSpeciesExpression

Synopsis:

```
DefineSpeciesExpression(variableName, expression) -> integer
```

variableName [string] The name of the variable to be created.

expression [string] The expression definition as a string.

return type [CLI_return_t] The DefineExpression functions return 1 on success and 0 on failure.

Description:
The DefineSpeciesExpression creates new species variables. Expression variables can be plotted like any other variable. The variableName argument is a string that contains the name of the new variable. You can pass the name of an existing expression if you want to provide a new expression definition. The expression argument is a string that contains the definition of the new variable in terms of math operators and pre-existing variable names Reference the VisIt User's Manual if you want more information on creating expressions, such as expression syntax, or a list of built-in expression functions.

Example:
```c
#!/usr/bin/env python
import visit
vis = visit.VisIt()
vis.OpenDatabase("/path/to/your/file.silo")
vis.DefineScalarExpression("new_var", "sin(u) + cos(w)")
vis.AddPlot("Pseudocolor", "new_var")
vis.DrawPlots()
```

(continues on next page)
AddPlot("Vector", "myvec")
DrawPlots()

2.4.42 DefineTensorExpression

Synopsis:

DefineTensorExpression(variableName, expression) -> integer

variableName  [string] The name of the variable to be created.
expression  [string] The expression definition as a string.
return type  [CLI_return_t] The DefineExpression functions return 1 on success and 0 on failure.

Description:

The DefineTensorExpression creates new tensor variables. Expression variables can be plotted like any other variable. The variableName argument is a string that contains the name of the new variable. You can pass the name of an existing expression if you want to provide a new expression definition. The expression argument is a string that contains the definition of the new variable in terms of math operators and pre-existing variable names. Reference the VisIt User’s Manual if you want more information on creating expressions, such as expression syntax, or a list of built-in expression functions.

Example:

```bash
$ visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
DefineScalarExpression("myvar", "sin(u) + cos(w)")
# Plot the scalar expression variable.
AddPlot("Pseudocolor", "myvar")
DrawPlots()
# Plot a vector expression variable.
DefineVectorExpression("myvec", "{u,v,w}"")
AddPlot("Vector", "myvec")
DrawPlots()
```

2.4.43 DefineVectorExpression

Synopsis:

DefineVectorExpression(variableName, expression) -> integer

variableName  [string] The name of the variable to be created.
expression  [string] The expression definition as a string.
return type  [CLI_return_t] The DefineExpression functions return 1 on success and 0 on failure.

Description:

The DefineVectorExpression creates new vector variables. Expression variables can be plotted like any other variable. The variableName argument is a string that contains the name of the new variable. You can pass the name of an existing expression if you want to provide a new expression definition. The expression argument is a string that contains the definition of the new variable in terms of math operators.
and pre-existing variable names. Reference the VisIt User’s Manual if you want more information on creating expressions, such as expression syntax, or a list of built-in expression functions.

Example:

```bash
#!/usr/bin/env python
import visit

# Open a database
visit.OpenDatabase("/usr/gapps/visit/data/globe.silo")

# Define a scalar expression
visit.DefineScalarExpression("myvar", "sin(u) + cos(w)"
# Plot the scalar expression variable.
visit.AddPlot("Pseudocolor", "myvar")
visit.DrawPlots()

# Plot a vector expression variable.
visit.DefineVectorExpression("myvec", 
"\{u,v,w\}
visit.AddPlot("Vector", "myvec")
visit.DrawPlots()
```

2.4.44 DeleteActivePlots

**Synopsis:**

```python
DeleteActivePlots() -> integer
```

**Return type** [CLI_return_t] The Delete functions return an integer value of 1 for success and 0 for failure.

**Description:**

The Delete functions delete plots from the active window’s plot list. The DeleteActivePlots function deletes all of the active plots from the plot list. There is no way to retrieve a plot once it has been deleted from the plot list. The active plots are set using the SetActivePlots function. The DeleteAllPlots function deletes all plots from the active window’s plot list regardless of whether or not they are active.

Example:

```bash
#!/usr/bin/env python
import visit

# Open a database
visit.OpenDatabase("/usr/gapps/visit/data/curv2d.silo")

# Add plots
visit.AddPlot("Pseudocolor", "d")
visit.AddPlot("Contour", "u")
visit.AddPlot("Mesh", "curvmesh2d")
visit.DrawPlots()

# Delete the mesh plot
visit.DeleteActivePlots() # Delete the mesh plot

# Delete all plots
visit.DeleteAllPlots() # Delete the pseudocolor and contour plots.
```

2.4.45 DeleteAllPlots

**Synopsis:**

```python
DeleteAllPlots() -> integer
```

**Return type** [CLI_return_t] The Delete functions return an integer value of 1 for success and 0 for failure.

**Description:**

The Delete functions delete plots from the active window’s plot list. The DeleteActivePlots function deletes all of the active plots from the plot list. There is no way to retrieve a plot once it has been deleted from the plot list. The active plots are set using the SetActivePlots function. The DeleteAllPlots function deletes all plots from the active window’s plot list regardless of whether or not they are active.
Example:

```c
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/curv2d.silo")
AddPlot("Pseudocolor", "d")
AddPlot("Contour", "u")
AddPlot("Mesh", "curvmesh2d")
DrawPlots()
DeleteActivePlots() # Delete the mesh plot
DeleteAllPlots() # Delete the pseudocolor and contour plots.
```

## 2.4.46 DeleteDatabaseCorrelation

### Synopsis:

`DeleteDatabaseCorrelation(name) -> integer`

- **name** [string] The name of the database correlation to delete.
- **return type** [CLI_return_t] The `DeleteDatabaseCorrelation` function returns 1 on success and 0 on failure.

### Description:

The `DeleteDatabaseCorrelation` function deletes a specific database correlation and its associated time slider. If you delete a database correlation whose time slider is being used for the current time slider, the time slider will be reset to the time slider of the next best suited database correlation. You can use the `DeleteDatabaseCorrelation` function to remove database correlations that you no longer need such as when you choose to examine databases that have nothing to do with your current databases.

### Example:

```c
#% visit -cli
dbs = ("dbA00.pdb", "dbB00.pdb")
OpenDatabase(dbs[0])
AddPlot("FilledBoundary", "material(mesh)")
OpenDatabase(dbs[1])
AddPlot("FilledBoundary", "material(mesh)")
DrawPlots()
CreateDatabaseCorrelation("common", dbs, 1)
SetTimeSliderState(10)
DeleteAllPlots()
DeleteDatabaseCorrelation("common")
CloseDatabase(dbs[0])
CloseDatabase(dbs[1])
```

## 2.4.47 DeleteExpression

### Synopsis:

`DeleteExpression(variableName) -> integer`

- **variableName** [string] The name of the expression variable to be deleted.
- **return type** [CLI_return_t] The `DeleteExpression` function returns 1 on success and 0 on failure.

### Description:

468 Chapter 2. VisIt Python (CLI) Interface Manual
The `DeleteExpression` function deletes the definition of an expression. The `variableName` argument is a string containing the name of the variable expression to be deleted. Any plot that uses an expression that has been deleted will fail to regenerate if its attributes are changed.

**Example:**

```plaintext
#!/usr/bin/env python3
import visit

db = visit.OpenDatabase("/usr/gapps/visit/data/globe.silo")
myvar = "sin(u) + cos(w)"
myvarplot = visit.AddPlot("Pseudocolor", myvar)
DrawPlots()
DeleteExpression(myvar)  # Delete the expression variable myvar.
```

### 2.4.48 DeleteNamedSelection

**Synopsis:**

`DeleteNamedSelection(name) -> integer`

**name** [string] The name of a named selection.

**return type** [CLI_return_t] The `DeleteNamedSelection` function returns 1 for success and 0 for failure.

**Description:**

Named Selections allow you to select a group of elements (or particles). One typically creates a named selection from a group of elements and then later applies the named selection to another plot (thus reducing the set of elements displayed to the ones from when the named selection was created). If you have created a named selection that you are no longer interested in, you can delete it with the `DeleteNamedSelection` function.

**Example:**

```plaintext
#!/usr/bin/env python3
import visit

db = visit.OpenDatabase("/usr/gapps/visit/data/wave*.silo database")
AddPlot("Pseudocolor", "pressure")
AddOperator("Clip")
c = ClipAttributes()
c.plane1Origin = (0,0.6,0)
c.plane1Normal = (0,-1,0)
SetOperatorOption(c)
DrawPlots()
CreateNamedSelection("els_above_y")
SetTimeSliderState(40)
DeleteNamedSelection("els_above_y")
CreateNamedSelection("els_above_y")
```

### 2.4.49 DeletePlotDatabaseKeyframe

**Synopsis:**

`DeletePlotDatabaseKeyframe(plotIndex, frame)`

**plotIndex** [integer] A zero-based integer value corresponding to a plot’s index in the plot list.

**frame** [integer] A zero-based integer value corresponding to a database keyframe at a particular animation frame.
Description:

The `DeletePlotDatabaseKeyframe` function removes a database keyframe from a specific plot. A database keyframe represents the database time state that will be used at a given animation frame when VisIt’s keyframing mode is enabled. The `plotIndex` argument is a zero-based integer that is used to identify a plot in the plot list. The frame argument is a zero-based integer that is used to identify the frame at which a database keyframe is to be removed for the specified plot.

Example:

```python
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/wave.visit")
k = GetKeyframeAttributes()
k.enabled, k.nFrames, k.nFramesWasUserSet = 1, 20, 1
SetKeyframeAttributes(k)
AddPlot("Pseudocolor", "pressure")
SetPlotDatabaseState(0, 0, 60)
# Repeat time state 60 for the first few animation frames by adding a
# keyframe at frame 3.
SetPlotDatabaseState(0, 3, 60)
SetPlotDatabaseState(0, 19, 0)
DrawPlots()
ListPlots()
# Delete the database keyframe at frame 3.
DeletePlotDatabaseKeyframe(0, 3)
ListPlots()
```

### 2.4.50 DeletePlotKeyframe

**Synopsis:**

`DeletePlotKeyframe(plotIndex, frame)`

**plotIndex** [integer] A zero-based integer value corresponding to a plot’s index in the plot list.

**frame** [integer] A zero-based integer value corresponding to a plot keyframe at a particular animation frame.

**Description:**

The `DeletePlotKeyframe` function removes a plot keyframe from a specific plot. A plot keyframe is the set of plot attributes at a specified frame. Plot keyframes are used to determine what plot attributes will be used at a given animation frame when VisIt’s keyframing mode is enabled. The `plotIndex` argument is a zero-based integer that is used to identify a plot in the plot list. The frame argument is a zero-based integer that is used to identify the frame at which a keyframe is to be removed.

**Example:**

```python
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/wave.visit")
k = GetKeyframeAttributes()
k.enabled, k.nFrames, k.nFramesWasUserSet = 1, 20, 1
SetKeyframeAttributes(k)
AddPlot("Pseudocolor", "pressure")
# Set up plot keyframes so the Pseudocolor plot's min will change
# over time.
p0 = PseudocolorAttributes()
p0.minFlag, p0.min = 1, 0.0
p1 = PseudocolorAttributes()
```
p1.minFlag, p1.min = 1, 0.5
SetPlotOptions(p0)
SetTimeSliderState(19)
SetPlotOptions(p1)
SetTimeSliderState(0)
DrawPlots()
ListPlots()

# Iterate over all animation frames and wrap around to the first one.
for i in list(range(TimeSliderGetNStates())) + [0]:
    SetTimeSliderState(i)
# Delete the plot keyframe at frame 19 so the min won't
# change anymore.
DeletePlotKeyframe(19)
ListPlots()
SetTimeSliderState(10)

## 2.4.51 DeleteViewKeyframe

### Synopsis:

DeleteViewKeyframe(frame)

**frame** [integer] A zero-based integer value corresponding to a view keyframe at a particular animation frame.

### Description:

The DeleteViewKeyframe function removes a view keyframe at a specified frame. View keyframes are used to determine what view will be used at a given animation frame when VisIt’s keyframing mode is enabled. The frame argument is a zero-based integer that is used to identify the frame at which a keyframe is to be removed.

### Example:

```python
# visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
k = KeyframeAttributes()
k.enabled, k.nFrames, k.nFramesWasUserSet = 1, 10, 1
SetKeyframeAttributes(k)
AddPlot("Pseudocolor", "u")
DrawPlots()

# Set some view keyframes
SetViewKeyframe()
v1 = GetView3D()
v1.viewNormal = (-0.66609, 0.337227, 0.665283)
v1.viewUp = (0.157431, 0.935425, -0.316537)
SetView3D(v1)
SetTimeSliderState(9)
SetViewKeyframe()
ToggleCameraViewMode()

# Iterate over the animation frames to watch the view change.
for i in list(range(10)) + [0]:
    SetTimeSliderState(i)
# Delete the last view keyframe, which is on frame 9.
DeleteViewKeyframe(9)
# Iterate over the animation frames again. The view should stay
# the same.
```
for i in range(10):
    SetTimeSliderState(i)

2.4.52 DeleteWindow

Synopsis:

DeleteWindow() -> integer

return type [CLI_return_t] The DeleteWindow function returns an integer value of 1 for success and 0 for failure.

Description:

The DeleteWindow function deletes the active visualization window and makes the visualization window with the smallest window index the new active window. This function has no effect when there is only one remaining visualization window.

Example:

```python
#% visit -cli
DeleteWindow()  # Does nothing since there is only one window
AddWindow()
DeleteWindow()  # Deletes the new window.
```

2.4.53 DemoteOperator

Synopsis:

DemoteOperator(opIndex) -> integer
DemoteOperator(opIndex, applyToAllPlots) -> integer

opIndex [integer] A zero-based integer corresponding to the operator that should be demoted.
applyToAllPlots [integer] An integer flag that causes all plots in the plot list to be affected when it is non-zero.

return type [CLI_return_t] DemoteOperator returns 1 on success and 0 on failure.

Description:

The DemoteOperator function moves an operator closer to the database in the visualization pipeline. This allows you to change the order of operators that have been applied to a plot without having to remove them from the plot. For example, consider moving a Slice to before a Reflect operator when it had been the other way around. Changing the order of operators can result in vastly different results for a plot. The opposite function is PromoteOperator.

Example:

```python
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/noise.silo")
AddPlot("Pseudocolor", "hardyglobal")
AddOperator("Slice")
s = SliceAttributes()
s.project2d = 0
s.originPoint = (0,5,0)
s.originType=s.Point
(continues on next page)
```
s.normal = (0,1,0)
s.upAxis = (-1,0,0)
SetOperatorOptions(s)
AddOperator("Reflect")
DrawPlots()

# Now reflect before slicing. We'll only get 1 slice plane
# instead of 2.
DemoteOperator(1)
DrawPlots()

2.4.54 DisableRedraw

Synopsis:

DisableRedraw()

Description:

The DisableRedraw function prevents the active visualization window from ever redrawing itself. This is a useful function to call when performing many operations that would cause unnecessary redraws in the visualization window. The effects of this function are undone by calling the RedrawWindow function.

Example:

```bash
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Contour", "u")
AddPlot("Pseudocolor", "w")
DrawPlots()
DisableRedraw()
AddOperator("Slice")
# Set the slice operator attributes
# Redraw now that the operator attributes are set. This will prevent 1 redraw.
RedrawWindow()
```

2.4.55 DrawPlots

Synopsis:

DrawPlots() -> integer

return type [CLI_return_t] The DrawPlots function returns an integer value of 1 for success and 0 for failure.

Description:

The DrawPlots function forces all new plots in the plot list to be drawn. Plots are added and then their attributes are modified. Finally, the DrawPlots function is called to make sure all of the new plots draw themselves in the visualization window. This function has no effect if all of the plots in the plot list are already drawn.

Example:
2.4.56 EnableTool

Synopsis:

EnableTool(toolIndex, activeFlag)

toolIndex [integer] This is an integer that corresponds to an interactive tool. (Plane tool = 0, Line tool = 1, Plane tool = 2, Box tool = 3, Sphere tool = 4, Axis Restriction tool = 5)

activeFlag [integer] An integer value of 1 enables the tool while a value of 0 disables the tool.

return [CLI_return_t] The EnableTool function returns 1 on success and 0 on failure.

Description:
The EnableTool function is used to set the enabled state of an interactive tool in the active visualization window. The toolIndex argument is an integer index that corresponds to a certain tool. The activeFlag argument is an integer value (0 or 1) that indicates whether to turn the tool on or off.

Example:

```c
## visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
DrawPlots()  # Draw the new pseudocolor plot.
EnableTool(0, 1)  # Turn on the line tool.
EnableTool(1, 1)  # Turn on the plane tool.
EnableTool(2, 1)  # Turn on the sphere tool.
EnableTool(2, 0)  # Turn off the sphere tool.
```

2.4.57 EvalCubic

Synopsis:

EvalCubic(t, c0, c1, c2, c3) -> f(t)

t [double] A floating point number in the range [0., 1.] that represents the distance from c0 to c3.

c0 [arithmetic expression object] The first control point. f(0) = c0. Any object that can be used in an arithmetic expression can be passed for c0.

c1 [arithmetic expression object] The second control point. Any object that can be used in an arithmetic expression can be passed for c1.

c2 [arithmetic expression object] The third control point. Any object that can be used in an arithmetic expression can be passed for c2.

c3 [arithmetic expression object] The last control point. f(1) = c3. Any object that can be used in an arithmetic expression can be passed for c3.

return [double] The EvalCubic function returns the interpolated value for t taking into account the control points that were passed in.
Description:
The EvalCubic function takes in four objects and blends them using a cubic polynomial and returns the blended value.

Example:

```csharp
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
DrawPlots()
v0 = GetView3D()
# rotate the plots
v1 = GetView3D()
# rotate the plots again.
v2 = GetView3D()
# rotate the plots one last time.
v3 = GetView3D()
# Fly around the plots using the views that have been specified.
nSteps = 100
for i in range(nSteps):
    t = float(i) / float(nSteps - 1)
    newView = EvalCubic(t, v0, v1, v2, v3)
    SetView3D(newView)
```

2.4.58 EvalCubicSpline

Synopsis:

```
EvalCubicSpline(t, weights, values) -> f(t)
```

t [double] A floating point number in the range [0., 1.] that represents the distance from the first control point to the last control point.

weights [tuple of doubles] A tuple of N floating point values in the range [0., 1.] that represent how far along in parameterized space, the values will be located.

values [tuple of arithmetic expression object] A tuple of N objects to be blended. Any objects that can be used in arithmetic expressions can be passed in.

return [double] The EvalCubicSpline function returns the interpolated value for t considering the objects that were passed in.

Description: The EvalCubicSpline function takes in N objects to be blended and blends them using piece-wise cubic polynomials and returns the blended value.

2.4.59 EvalLinear

Synopsis:

```
EvalLinear(t, value1, value2) -> f(t)
```

t [double] A floating point value in the range [0., 1.] that represents the distance between the first and last control point in parameterized space.

value1 [arithmetic expression object] Any object that can be used in an arithmetic expression. f(0) = value1.

value2 [arithmetic expression object] Any object that can be used in an arithmetic expression. f(1) = value2.
**return**  [double] The EvalLinear function returns an interpolated value for \( t \) based on the objects that were passed in.

**Description:** The EvalLinear function linearly interpolates between two values and returns the result.

**Example:**

```python
# visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
DrawPlots()
c0 = GetView3D()
c1 = GetView3D()
c1.viewNormal = (-0.499159, 0.475135, 0.724629)
c1.viewUp = (0.196284, 0.876524, -0.439521)
nSteps = 100
for i in range(nSteps):
    t = float(i) / float(nSteps - 1)
    v = EvalLinear(t, c0, c1)
    SetView3D(v)
```

---

**2.4.60 EvalQuadratic**

**Synopsis:**

\[ \text{EvalQuadratic}(t, c0, c1, c2) \rightarrow f(t) \]

**t**  [double] A floating point number in the range \([0., 1.]\) that represents the distance from \( c0 \) to \( c3 \).

**c0**  [arithmetic expression object] The first control point. \( f(0) = c0 \). Any object that can be used in an arithmetic expression can be passed for \( c0 \).

**c1**  [arithmetic expression object] The second control point. Any object that can be used in an arithmetic expression can be passed for \( c1 \).

**c2**  [arithmetic expression object] The last control point. \( f(1) = c2 \). Any object that can be used in an arithmetic expression can be passed for \( c2 \).

**return**  [double] The EvalQuadratic function returns the interpolated value for \( t \) taking into account the control points that were passed in.

**Description:** The EvalQuadratic function takes in four objects and blends them using a cubic polynomial and returns the blended value.

**Example:**

```python
# visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
DrawPlots()
v0 = GetView3D()
# rotate the plots
v1 = GetView3D()
# rotate the plots one last time.
v2 = GetView3D()
# fly around the plots using the views that have been specified.
for i in range(nSteps):
    t = float(i) / float(nSteps - 1)
```

(continues on next page)
newView = EvalQuadratic(t, v0, v1, v2)
SetView3D(newView)

### 2.4.61 ExecuteMacro

#### Synopsis:

```python
ExecuteMacro(name) -> value
```

- **name** [string] The name of the macro to execute.
- **return type** [value] The ExecuteMacro function returns the value returned from the user’s macro function.

#### Description:

The ExecuteMacro function lets you call a macro function that was previously registered using the RegisterMacro method. Once macros are registered with a name, this function can be called whenever the macro function associated with that name needs to be called. The VisIt gui uses this function to tell the Python interface when macros need to be executed in response to user button clicks.

#### Example:

```python
def SetupMyPlots():
    OpenDatabase('noise.silo')
    AddPlot('Pseudocolor', 'hardyglobal')
    DrawPlots()

RegisterMacro('Setup My Plots', SetupMyPlots)
ExecuteMacro('Setup My Plots')
```

### 2.4.62 ExportDatabase

#### Synopsis:

```python
ExportDatabase(e) -> integer
ExportDatabase(e, o) -> integer
```

- **e** [ExportDBAttributes object] An object of type ExportDBAttributes. This object specifies the options for exporting the database.
- **o** [dictionary] A dictionary containing a key/value mapping to set options needed by the database exporter. The default values can be obtained in the appropriate format using GetExportOptions('plugin').
- **return type** [CLI_return_t] Returns 1 on success, 0 on failure.

#### Description:

The ExportDatabase function exports the active plot for the current window to a file. The format of the file, name, and variables to be saved are specified using the ExportDBAttributes argument. Note that this functionality is distinct from the geometric formats of SaveWindow, such as STL. SaveWindow can only save surfaces (triangle meshes), while ExportDatabase can export an entire three dimensional data set.

#### Example:
2.4.63 Expressions

Synopsis:

Expressions() -> tuple of expression tuples

**return type** [tuple of expression tuples] The Expressions function returns a tuple of tuples that contain two strings that give the expression name and definition.

**Description:**

The Expressions function returns a tuple of tuples that contain two strings that give the expression name and definition. This function is useful for listing the available expressions or for iterating through a list of expressions in order to create plots.

**Example:**

```python
#% visit -cli
SetWindowLayout(4)
DefineScalarExpression("sin_u", "sin(u)")
DefineScalarExpression("cos_u", "cos(u)")
DefineScalarExpression("neg_u", "-u")
DefineScalarExpression("bob", "sin_u + cos_u")
for i in range(1,5):
    SetActiveWindow(i)
    OpenDatabase("/usr/gapps/visit/data/globe.silo")
    exprName = Expressions()[i-1][0]
    AddPlot("Pseudocolor", exprName)
    DrawPlots()
```

2.4.64 GetActiveContinuousColorTable

Synopsis:

GetActiveContinuousColorTable() -> string

**return type** [string] Both functions return a string object containing the name of a color table.

**Description:**

A color table is a set of color values that are used as the colors for plots. VisIt supports two flavors of color table: continuous and discrete. A continuous color table is defined by a small set of color control points and the colors specified by the color control points are interpolated smoothly to fill in any gaps. Continuous color tables are used for plots that need to be colored smoothly by a variable (e.g. Pseudocolor plot). A discrete color table is a set of color control points that are used to color distinct regions of a plot.
(e.g. Subset plot). VisIt supports the notion of default continuous and default discrete color tables so plots can just use the “default” color table. This lets you change the color table used by many plots by just changing the “default” color table. The GetActiveContinuousColorTable function returns the name of the default continuous color table. The GetActiveDiscreteColorTable function returns the name of the default discrete color table.

Example:

```python
%% visit -cli
print("Default continuous color table: \$s" % GetActiveContinuousColorTable())
print("Default discrete color table: \$s" % GetActiveDiscreteColorTable())
```

### 2.4.65 GetActiveDiscreteColorTable

**Synopsis:**

GetActiveDiscreteColorTable() -> string

**return type** [string] Both functions return a string object containing the name of a color table.

**Description:**

A color table is a set of color values that are used as the colors for plots. VisIt supports two flavors of color table: continuous and discrete. A continuous color table is defined by a small set of color control points and the colors specified by the color control points are interpolated smoothly to fill in any gaps. Continuous color tables are used for plots that need to be colored smoothly by a variable (e.g. Pseudocolor plot). A discrete color table is a set of color control points that are used to color distinct regions of a plot (e.g. Subset plot). VisIt supports the notion of default continuous and default discrete color tables so plots can just use the “default” color table. This lets you change the color table used by many plots by just changing the “default” color table. The GetActiveContinuousColorTable function returns the name of the default continuous color table. The GetActiveDiscreteColorTable function returns the name of the default discrete color table.

Example:

```python
%% visit -cli
print("Default continuous color table: \$s" % GetActiveContinuousColorTable())
print("Default discrete color table: \$s" % GetActiveDiscreteColorTable())
```

### 2.4.66 GetActiveTimeSlider

**Synopsis:**

GetActiveTimeSlider() -> string

**return type** [string] The GetActiveTimeSlider function returns a string containing the name of the active time slider.

**Description:**

VisIt can support having multiple time sliders when you have opened more than one time-varying database. You can then use each time slider to independently change time states for each database or you can use a database correlation to change time states for all databases simultaneously. Every time-varying database has a database correlation and every database correlation has its own time slider. If you want to query to determine which time slider is currently the active time slider, you can use the GetActiveTimeSlider function.
Example:

```python
# visit -cli
OpenDatabase("dbA00.pdb")
AddPlot("FilledBoundary", "material(mesh)")
OpenDatabase("dbB00.pdb")
AddPlot("FilledBoundary", "materials(mesh)")
print("Active time slider: %s" % GetActiveTimeSlider())
CreateDatabaseCorrelation("common", ("dbA00.pdb", "dbB00.pdb"), 2)
print("Active time slider: %s" % GetActiveTimeSlider())
```

## 2.4.67 GetAnimationAttributes

### Synopsis:

`GetAnimationAttributes() -> AnimationAttributes object`

**return type** [AnimationAttributes object] The `GetAnimationAttributes` function returns an `AnimationAttributes` object.

**Description:**

This function returns the current animation attributes, which contain the animation mode, increment, and playback speed.

**Example:**

```python
a = GetAnimationAttributes()
print(a)
```

## 2.4.68 GetAnimationTimeout

### Synopsis:

`GetAnimationTimeout() -> integer`

**return type** [CLI_return_t] The `GetAnimationTimeout` function returns an integer that contains the time interval, measured in milliseconds, between the rendering of animation frames.

**Description:**

The `GetAnimationTimeout` returns an integer that contains the time interval, measured in milliseconds, between the rendering of animation frames.

**Example:**

```python
# visit -cli
print("Animation timeout = \$d\" % GetAnimationTimeout())
```

## 2.4.69 GetAnnotationAttributes

### Synopsis:

`GetAnnotationAttributes() -> AnnotationAttributes object`

```
**return type** [AnnotationAttributes object] The GetAnnotationAttributes function returns an AnnotationAttributes object that contains the annotation settings for the active visualization window.

**Description:**

The GetAnnotationAttributes function returns an AnnotationAttributes object that contains the annotation settings for the active visualization window. It is often useful to retrieve the annotation settings and modify them to suit the visualization.

**Example:**

```python
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
DrawPlots()
a = GetAnnotationAttributes()
print(a)
a.backgroundColor = a.BACKGROUNDMODE_GRADIENT
a.gradientColor1 = (0, 0, 255)
SetAnnotationAttributes(a)
```

### 2.4.70 GetAnnotationObject

**Synopsis:**

GetAnnotationObject(string) -> Annotation object

**string** [string] The name of the annotation object as returned by GetAnnotationObjectNames.

**return type** [Annotation object] GetAnnotationObject returns a reference to an annotation object that was created using the CreateAnnotationObject function.

**Description:**

GetAnnotationObject returns a reference to an annotation object that was created using the CreateAnnotationObject function. The string argument specifies the name of the desired annotation object. It must be one of the names returned by GetAnnotationObjectNames. This function is not currently necessary unless the annotation object that you used to create an annotation has gone out of scope and you need to create another reference to the object to set its properties. Also note that although this function will apparently also accept an integer index, that mode of access is not reliably and should be avoided.

**Example:**

```python
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/wave.visit")
AddPlot("Pseudocolor", "pressure")
DrawPlots()
a = CreateAnnotationObject("TimeSlider")
GetAnnotationObjectNames()
["plot0000", "TimeSlider1"]
ref = GetAnnotationObject("TimeSlider1")
print(ref)
```

### 2.4.71 GetAnnotationObjectNames

**Synopsis:**


GetAnnotationObjectNames() \rightarrow \text{tuple of strings}

\textbf{return type} [tuple of strings] GetAnnotationObjectNames returns a tuple of strings of the names of all annotation objects defined for the currently active window.

\textbf{Example:}

```python
names = GetAnnotationObjectNames()
names
['plot0000', 'Line2D1', 'TimeSlider1']
```

### 2.4.72 GetCallbackArgumentCount

\textbf{Synopsis:}

\text{GetCallbackArgumentCount}(\text{callbackName}) \rightarrow \text{integer}

\textbf{callbackName} [\text{string}] The name of a callback function. This name is a member of the tuple returned by \text{GetCallbackNames}().

\textbf{return type} [\text{CLI\_return\_t}] The \text{GetCallbackArgumentCount} function returns the number of arguments associated with a particular callback function.

\textbf{Example:}

```python
cbName = 'OpenDatabaseRPC'
count = GetCallbackArgumentCount(cbName)
print('The number of arguments for %s is: %d' % (cbName, count))
```

### 2.4.73 GetCallbackNames

\textbf{Synopsis:}

\text{GetCallbackNames}() \rightarrow \text{tuple of string objects}

\textbf{return type} [tuple of string objects] GetCallbackNames returns a tuple containing the names of valid callback function identifiers for use in \text{RegisterCallback}().

\textbf{Description:}

The GetCallbackNames function returns a tuple containing the names of valid callback function identifiers for use in \text{RegisterCallback}().

\textbf{Example:}

```python
import visit
print(visit.GetCallbackNames())
```

### 2.4.74 GetDatabaseNStates

\textbf{Synopsis:}

\text{GetDatabaseNStates}() \rightarrow \text{integer}

\textbf{return type} [\text{CLI\_return\_t}] Returns the number of time states in the active database or 0 if there is no active database.
Description:

GetDatabaseNStates returns the number of time states in the active database, which is not the same as
the number of states in the active time slider. Time sliders can have different lengths due to database
correlations and keyframing. Use this function when you need the actual number of time states in the
active database.

Example:

```python
%% visit -cli
OpenDatabase("/usr/gapps/visit/data/wave*.silo database")
print("Number of time states: %d" % GetDatabaseNStates())
```

2.4.75 GetDebugLevel

Synopsis:

```
GetDebugLevel() -> integer
```

return type [CLI_return_t] The GetDebugLevel function returns the debug level of the VisIt module.

Description:

The GetDebugLevel and SetDebugLevel functions are used when debugging VisIt Python scripts. The
GetDebugLevel function can be used in Python scripts to alter the behavior of the script. For instance, the
debug level can be used to selectively print values to the console.

Example:

```python
%% visit -cli -debug 2
print("VisIt's debug level is: %d" % GetDebugLevel())
```

2.4.76 GetDefaultFileOpenOptions

Synopsis:

```
GetDefaultFileOpenOptions(pluginName) -> dictionary
```

pluginName [string] The name of a plugin.

return type [dictionary] Returns a dictionary containing the options.

Description:

GetDefaultFileOpenOptions returns the current options used to open new files when a specific plugin is
triggered.

Example:

```python
%% visit -cli
OpenMDServer()
opts = GetDefaultFileOpenOptions("VASP")
opts["Allow multiple timesteps"] = 1
SetDefaultFileOpenOptions("VASP", opts)
OpenDatabase("CHGCAR")
```
2.4.77 GetDomains

Synopsis:

GetDomains() -> tuple of strings

return type [tuple of strings] GetDomains returns a tuple of strings.

Description:

GetDomains returns a tuple containing the names of all of the domain subsets for a plot that was created using a database with multiple domains. This function can be used in specialized logic that iterates over domains to turn them on or off in some programmed way.

Example:

```python
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/multi_ucd3d.silo")
AddPlot("Pseudocolor", "u")
DrawPlots()
doms = GetDomains()
print(doms)
# Turn off all but the last domain, one after the other.
for d in doms[:-1]:
    TurnDomainsOff(d)
```

2.4.78 GetEngineList

Synopsis:

GetEngineList() -> tuple of strings
GetEngineList(flag) -> tuple of tuples of strings

flag [integer] If flag is a non-zero integer then the function returns a tuple of tuples with information about simulations.

return type [tuple of strings] GetEngineList returns a tuple of strings containing the names of the computers on which compute engines are running. If flag is a non-zero integer argument then the function returns a tuple of tuples where each tuple is of length 2. Element 0 contains the names of the computers where the engines are running. Element 1 contains the names of the simulations being run.

Description:

The GetEngineList function returns a tuple of strings containing the names of the computers on which compute engines are running. This function can be useful if engines are going to be closed and opened explicitly in the Python script. The contents of the tuple can be used to help determine which compute engines should be closed or they can be used to determine if a compute engine was successfully launched.

Example:

```python
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
OpenDatabase("mcr:/usr/gapps/visit/data/globe.silo")
AddPlot("Mesh", "mesh1")
DrawPlots()
for name in GetEngineList():
    print("VisIt has a compute engine running on %s\n %s") % name
CloseComputeEngine(GetEngineList()[1])
```
2.4.79 GetEngineProperties

Synopsis:

```plaintext
GetEngineProperties() -> EngineProperties object
GetEngineProperties(engine) -> EngineProperties object
GetEngineProperties(engine, sim) -> EngineProperties object
```

**engine** When engine is passed and it matches one of the computer names returned from GetEngineList() then the EngineProperties object for that engine is returned.

**sim** When both engine and sim arguments are passed, then the EngineProperties object for the simulation is returned.


**Description:**

GetEngineProperties returns an EngineProperties object containing the properties for the specified compute engine/sim. The EngineProperties let you discover information such as number of processors, etc for a compute engine/sim.

**Example:**

```plaintext
#% visit -cli
db = "/usr/gapps/visit/data/globe.silo"
OpenDatabase(db)
props = GetEngineProperties(GetEngineList()[0])
```

2.4.80 GetGlobalAttributes

Synopsis:

```plaintext
GetGlobalAttributes() -> GlobalAttributes object
```

**return type** [GlobalAttributes object] Returns a GlobalAttributes object that has been initialized.

**Description:**

The GetGlobalAttributes function returns a GlobalAttributes object that has been initialized with the current state of the viewer proxy’s GlobalAttributes object. The GlobalAttributes object contains read-only information about the list of sources, the list of windows, and various flags that can be queried.

**Example:**

```plaintext
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
DrawPlots()
g = GetGlobalAttributes()
print(g)
```

2.4.81 GetGlobalLineoutAttributes

Synopsis:

```plaintext
GetGlobalLineoutAttributes() -> GlobalLineoutAttributes object
```
return type [GlobalLineoutAttributes object] Returns an initialized GlobalLineoutAttributes object.

Description:

The GetGlobalLineoutAttributes function returns an initialized GlobalLineoutAttributes object. The GlobalLineoutAttributes, as suggested by its name, contains global properties that apply to all lineouts. You can use the GlobalLineoutAttributes object to turn on lineout sampling, specify the destination window, etc. for curve plots created as a result of performing lineouts. Once you make changes to the object by setting its properties, use the SetGlobalLineoutAttributes function to make VisIt use the modified global lineout attributes.

Example:

```python
#% visit -cli
SetWindowLayout(4)
OpenDatabase("/usr/gapps/visit/data/curv2d.silo")
AddPlot("Pseudocolor", "d")
DrawPlots()
g = GetGlobalLineoutAttributes()
print(g)
g.samplingOn = 1
g.windowId = 4
g.createWindow = 0
g.numSamples = 100
SetGlobalLineoutAttributes(g)
Lineout((-3,2),(3,3),("default"))
```

### 2.4.82 GetInteractorAttributes

Synopsis:

GetInteractorAttributes() -> InteractorAttributes object

return type [InteractorAttributes object] Returns an initialized InteractorAttributes object.

Description:

The GetInteractorAttributes function returns an initialized InteractorAttributes object. The InteractorAttributes object can be used to set certain interactor properties. Interactors, can be thought of as how mouse clicks and movements are translated into actions in the vis window. To set the interactor attributes, first get the interactor attributes using the GetInteractorAttributes function. Once you’ve set the object’s properties, call the SetInteractorAttributes function to make VisIt use the new interactor attributes.

Example:

```python
#% visit -cli
ia = GetInteractorAttributes()
print(ia)
ia.showGuidelines = 0
SetInteractorAttributes(ia)
```

### 2.4.83 GetKeyframeAttributes

Synopsis:

GetKeyframeAttributes() -> KeyframeAttributes object

return type [KeyframeAttributes object] Returns an initialized KeyframeAttributes object.

Description:

The GetKeyframeAttributes function returns an initialized KeyframeAttributes object. The KeyframeAttributes object can be used to set certain keyframe properties. Keyframes, can be thought of as key positions for animations. To set the keyframe attributes, first get the keyframe attributes using the GetKeyframeAttributes function. Once you’ve set the object’s properties, call the SetKeyframeAttributes function to make VisIt use the new keyframe attributes.

Example:

```python
#% visit -cli
ia = GetInteractorAttributes()
print(ia)
ia.showGuidelines = 0
SetInteractorAttributes(ia)
```
**return type**  [KeyframeAttributes object] GetKeyframeAttributes returns an initialized KeyframeAttributes object.

**Description:**

Use the GetKeyframeAttributes function when you want to examine a KeyframeAttributes object so you can determine VisIt’s state when it is in keyframing mode. The KeyframeAttributes object allows you to see whether VisIt is in keyframing mode and, if so, how many animation frames are in the current keyframe animation.

**Example:**

```python
# visit -cli
k = GetKeyframeAttributes()
p = print(k)
k.enabled, k.nFrames, k.nFramesWasUserSet = 1, 100, 1
SetKeyframeAttributes(k)
```

### 2.4.84 GetLastError

**Synopsis:**

GetLastError() -> string

**return type**  [string] GetLastError returns a string containing the last error message that VisIt issued.

**Description:**

The GetLastError function returns a string containing the last error message that VisIt issued.

**Example:**

```python
# visit -cli
OpenDatabase("/this/database/does/not/exist")
p = print("VisIt Error: %s" % GetLastError())
```

### 2.4.85 GetLight

**Synopsis:**

GetLight(index) -> LightAttributes object

**index**  [integer] A zero-based integer index into the light list. Index can be in the range [0,7].

**return type**  [LightAttributes object] GetLight returns a LightAttributes object.

**Description:**

The GetLight function returns a LightAttributes object containing the attributes for a specific light. You can use the LightAttributes object that GetLight returns to set light properties and then you can pass the object to SetLight to make VisIt use the light properties that you’ve set.

**Example:**

```python
# visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "w")
p = PseudocolorAttributes()
p.colorTableName = "xray"
```

(continues on next page)
SetPlotOptions(p)
DrawPlots()
InvertBackgroundColor()
light = GetLight(0)
print(light)
light.enabledFlag = 1
light.direction = (0,-1,0)
light.color = (255,0,0,255)
SetLight(0, light)
light.color,light.direction = (0,255,0,255), (-1,0,0)
SetLight(1, light)

2.4.86 GetLocalHostName

Synopsis:

GetLocalHostName() -> string

return type  [string] Both functions return a string.

Description:

The GetLocalHostName function returns a string that contains the name of the local computer.

Example:

```python
#$ visit -cli
print("Local machine name is: %s" % GetLocalHostName())
print("My username: %s" % GetLocalUserName())
```

2.4.87 GetLocalUserName

Synopsis:

GetLocalUserName() -> string

return type  [string] Both functions return a string.

Description:

The GetLocalUserName function returns a string containing the name of the user running VisIt.

Example:

```python
#$ visit -cli
print("Local machine name is: %s" % GetLocalHostName())
print("My username: %s" % GetLocalUserName())
```

2.4.88 GetMachineProfile

Synopsis:

GetMachineProfile(hostname) -> MachineProfile object
hostname : string


**Description:**

Gets the MachineProfile for a given hostname

### 2.4.89 GetMachineProfileNames

**Synopsis:**

GetMachineProfileNames() -> [hostname1, hostname2, ...]

**return type** [list of strings] A list of MachineProfile hostnames

**Description:**

Returns a list of hostnames that can be used to get a specific MachineProfile

### 2.4.90 GetMaterialAttributes

**Synopsis:**

GetMaterialAttributes() -> MaterialAttributes object

**return type** [MaterialAttributes object] Returns a MaterialAttributes object.

**Description:**

The GetMaterialAttributes function returns a MaterialAttributes object that contains VisIt’s current material interface reconstruction settings. You can set properties on the MaterialAttributes object and then pass it to SetMaterialAttributes to make VisIt use the new material attributes that you’ve specified:

**Example:**

```plaintext
# visit -cli
OpenDatabase("/usr/gapps/visit/data/allinone00.pdb")
AddPlot("Pseudocolor", "mesh/mixvar")
p = PseudocolorAttributes()
p.min, p.minFlag = 4.0, 1
p.max, p.maxFlag = 13.0, 1
SetPlotOptions(p)
DrawPlots()
# Tell VisIt to always do material interface reconstruction.
m = GetMaterialAttributes()
m.forceMIR = 1
SetMaterialAttributes(m)
ClearWindow()
# Redraw the plot forcing VisIt to use the mixed variable information.
DrawPlots()
```

### 2.4.91 GetMaterials

**Synopsis:**
GetMaterials() -> tuple of strings

**return type** [tuple of strings] The GetMaterials function returns a tuple of strings.

**Description:**

The GetMaterials function returns a tuple of strings containing the names of the available materials for the current plot’s database. Note that the active plot’s database must have materials for this function to return a tuple that has any string objects in it. Also, you must have at least one plot. You can use the materials returned by the GetMaterials function for a variety of purposes including turning materials on or off.

**Example:**

```python
#!/usr/bin/env python
import visit

# Open database
visit.OpenDatabase("/usr/gapps/visit/data/allinone00.pdb")

# Add plot
visit.AddPlot("Pseudocolor", "mesh/mixvar")

# Draw plots
visit.DrawPlots()

mats = visit.GetMaterials()

for m in mats[:-1]:
    visit.TurnMaterialOff(m)
```

### 2.4.92 GetMeshManagementAttributes

**Synopsis:**

GetMeshManagementAttributes() -> MeshmanagementAttributes object

**return type** [MeshmanagementAttributes object] Returns a MeshmanagementAttributes object.

**Description:**

The GetMeshmanagementAttributes function returns a MeshmanagementAttributes object that contains VisIt’s current mesh discretization settings. You can set properties on the MeshManagementAttributes object and then pass it to SetMeshManagementAttributes to make VisIt use the new material attributes that you’ve specified.

**Example:**

```python
#!/usr/bin/env python
import visit

# Open database
visit.OpenDatabase("/usr/gapps/visit/data/csg.silo")

# Add plot
visit.AddPlot("Mesh", "csgmesh")

# Draw plots
visit.DrawPlots()

# Tell VisIt to always do material interface reconstruction.
mma = visit.GetMeshManagementAttributes()
mma.discretizationTolernace = (0.01, 0.025)
visit.SetMeshManagementAttributes(mma)

# Clear window
visit.ClearWindow()

# Redraw the plot forcing VisIt to use the mixed variable information.
visit.DrawPlots()
```

### 2.4.93 GetMetaData

**Synopsis:**

GetMetaData(db) -> avtDatabaseMetaData object

GetMetaData(db, ts) -> avtDatabaseMetaData object
The name of the database for which to return metadata.

An optional integer indicating the time state at which to open the database.

The GetMetaData function returns an avtDatabaseMetaData object.

VisIt relies on metadata to populate its variable menus and make important decisions. Metadata can be used to create complex scripts whose behavior adapts based on the contents of the database.

Example:

```python
md = GetMetaData('noise.silo')
for i in range(md.GetNumScalars()):
    AddPlot('Pseudocolor', md.GetScalars(i).name)
DrawPlots()
```

## 2.4.94 GetNumPlots

### Synopsis:

GetNumPlots() -> integer

### Description:

The GetNumPlots function returns the number of plots in the active window.

### Example:

```python
#% visit -cli
print("Number of plots", GetNumPlots())
OpenDatabase("/usr/gapps/visit/data/curv2d.silo")
AddPlot("Pseudocolor", "d")
print("Number of plots", GetNumPlots())
AddPlot("Mesh", "curvmesh2d")
DrawPlots()
print("Number of plots", GetNumPlots())
```

## 2.4.95 GetOperatorOptions

### Synopsis:

GetOperatorOptions(index) -> operator attributes object

The integer index of the operator within the plot’s list of operators.

The GetOperatorOptions function returns an operator attributes object.

### Description:

This function is provided to make it easy to probe the current attributes for a specific operator on the active plot.

### Example:

```python
```

2.4. Functions
AddPlot('Pseudocolor', 'temperature')
AddOperator('Transform')
AddOperator('Transform')
t = GetOperatorOptions(1)
print('Attributes for the 2nd Transform operator:', t)

2.4.96 GetPickAttributes

Synopsis:

GetPickAttributes() -> PickAttributes object

**return type** [PickAttributes object] GetPickAttributes returns a PickAttributes object.

**Description:**

The GetPickAttributes object returns the pick settings that VisIt is currently using when it performs picks. These settings mainly determine which pick information is displayed when pick results are printed out but they can also be used to select auxiliary variables and generate time curves. You can examining the settings and you can set properties on the returned object. Once you’ve changed pick settings by setting properties on the object, you can pass the altered object to the SetPickAttributes function to force VisIt to use the new pick settings.

**Example:**

```python
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/allinone00.pdb")
AddPlot("Pseudocolor", "mesh/ireg")
DrawPlots()
p = GetPickAttributes()
print(p)
p.variables = ("default", "mesh/a", "mesh/mixvar")
SetPickAttributes(p)
# Now do some interactive picks and you'll see pick information
# for more than 1 variable.
p.doTimeCurve = 1
SetPickAttributes(p)
# Now do some interactive picks and you'll get time-curves in
# a new window.
```

2.4.97 GetPickOutput

Synopsis:

GetPickOutput() -> string

**return type** [string] GetPickOutput returns a string containing the output from the last pick.

**Description:**

The GetPickOutput returns a string object that contains the output from the last pick.

**Example:**
2.4.98 GetPickOutputObject

Synopsis:

GetPickOutputObject() \rightarrow \text{dictionary}

return type [dictionary] GetPickOutputObject returns a dictionary produced by the last pick.

Description:

GetPickOutputObject returns a dictionary object containing output from the last pick.

Example:

```python
#$ visit -cli
OpenDatabase("/usr/gapps/visit/data/rect2d.silo")
AddPlot("Pseudocolor", "d")
DrawPlots()
ZonePick(coord=(0.4, 0.6, 0), vars=\{"default", "u", "v"\})
s = GetPickOutputObject()
print(s)
```

2.4.99 GetPipelineCachingMode

Synopsis:

GetPipelineCachingMode() \rightarrow \text{integer}

return type [CLI_return_t] The GetPipelineCachingMode function returns 1 if pipelines are being cached and 0 otherwise.

Description:

The GetPipelineCachingMode function returns whether or not pipelines are being cached in the viewer. For animations of long time sequences, it is often useful to turn off pipeline caching so the viewer does not run out of memory.

Example:

```python
#$ visit -cli
offon = ('off', 'on')
pipeline_caching_is = "\$s" % offon[GetPipelineCachingMode()]
print(pipeline_caching_is)
```

2.4.100 GetPlotInformation

Synopsis:

2.4. Functions 493
GetPlotInformation() -> dictionary

**return type** [dictionary] GetPlotInformation returns a dictionary.

**Description:**

The GetPlotInformation function returns information about the active plot. For example, a Curve plot will return the xy pairs that comprise the curve. The tuple is arranged \(<x_1, y_1, x_2, y_2, \ldots, x_n, y_n>\).

**Example:**

```python
#$ visit -cli
OpenDatabase("/usr/gapps/visit/data/rect2d.silo")
AddPlot("Pseudocolor", "d")
DrawPlots()
Lineout((0, 0), (1, 1))
SetActiveWindow(2)
info = GetPlotInformation()
lineout = info["Curve"]
print("The first lineout point is: [(\%g, \%g) \ " \ % lineout[0], lineout[1])")
```

### 2.4.101 GetPlotList

**Synopsis:**

GetPlotList() -> PlotList object

**return type** [PlotList object] The GetPlotList function returns a PlotList object.

**Description:**

The GetPlotList function returns a copy of the plot list that gets exchanged between VisIt’s viewer and its clients. The plot list object contains the list of plots, along with the databases, and any operators that are applied to each plot. Changing this object has NO EFFECT but it can be useful when writing complex functions that need to know about the plots and operators that exist within a visualization window.

**Example:**

```python
# Copy plots (without operators to window 2)
pl = GetPlotList()
AddWindow()
for i in range(pl.GetNumPlots()):
    AddPlot(PlotPlugins()[pl.GetPlots(i).plotType], pl.GetPlots(i).plotVar)
DrawPlots()
```

### 2.4.102 GetPlotOptions

**Synopsis:**

GetPlotOptions() -> plot attributes object

**return type** [plot attributes object] The GetPlotOptions function returns a plot attributes object whose type varies depending the selected plots.

**Description:**

This function is provided to make it easy to probe the current attributes for the selected plot.
Example:

```python
pc = GetPlotOptions()
pc.legend = 0
SetPlotOptions(pc)
```

### 2.4.103 GetPreferredFileFormats

**Synopsis:**

GetPreferredFileFormats() -> tuple of strings

**return type** [tuple of strings] The GetPreferredFileFormats returns the current list of preferred plugins.

**Description:**

The GetPreferredFileFormats method is a way to get the list of file format reader plugins which are tried before any others. These IDs are full IDs, not just names, and are tried in order.

**Example:**

```python
GetPreferredFileFormats()
# returns ('Silo_1.0',)
```

### 2.4.104 GetQueryOutputObject

**Synopsis:**

GetQueryOutputObject() -> dictionary or value

**return type** [dictionary or value] GetQueryOutputObject returns an xml string produced by the last query.

**Description:**

Both the GetQueryOutputString and GetQueryOutputValue functions return information about the last query to be executed but the type of information returns differs. GetQueryOutputString returns a string containing the output of the last query. GetQueryOutputValue returns a single number or tuple of numbers, depending on the nature of the last query to be executed. GetQueryOutputXML and GetQueryOutputObject expose more complex query output.

**Example:**

```python
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/rect2d.silo")
AddPlot("Pseudocolor", "d")
DrawPlots()
Query("MinMax")
print(GetQueryOutputString())
print("The min is: \$g and the max is: \$g" % GetQueryOutputValue())
```

### 2.4.105 GetQueryOutputString

**Synopsis:**
GetQueryOutputString() -> string

return type  [string] GetQueryOutputString returns a string.

Description:
Both the GetQueryOutputString and GetQueryOutputValue functions return information about the last query to be executed but the type of information returns differs. GetQueryOutputString returns a string containing the output of the last query. GetQueryOutputValue returns a single number or tuple of numbers, depending on the nature of the last query to be executed. GetQueryOutputXML and GetQueryOutputObject expose more complex query output.

Example:

```python
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/rect2d.silo")
AddPlot("Pseudocolor", "d")
DrawPlots()
Query("MinMax")
print(GetQueryOutputString())
print("The min is: %g and the max is: %g" % GetQueryOutputValue())
```

2.4.106 GetQueryOutputValue

Synopsis:

GetQueryOutputValue() -> double, tuple of doubles

return type  [double, tuple of doubles] GetQueryOutputValue returns a single double precision number or a tuple of double precision numbers.

Description:
Both the GetQueryOutputString and GetQueryOutputValue functions return information about the last query to be executed but the type of information returns differs. GetQueryOutputString returns a string containing the output of the last query. GetQueryOutputValue returns a single number or tuple of numbers, depending on the nature of the last query to be executed. GetQueryOutputXML and GetQueryOutputObject expose more complex query output.

Example:

```python
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/rect2d.silo")
AddPlot("Pseudocolor", "d")
DrawPlots()
Query("MinMax")
print(GetQueryOutputString())
print("The min is: %g and the max is: %g" % GetQueryOutputValue())
```

2.4.107 GetQueryOutputXML

Synopsis:

GetQueryOutputXML() -> string

return type  [string] GetQueryOutputXML returns an xml string produced by the last query.
Description:

Both the GetQueryOutputString and GetQueryOutputValue functions return information about the last query to be executed but the type of information returns differs. GetQueryOutputString returns a string containing the output of the last query. GetQueryOutputValue returns a single number or tuple of numbers, depending on the nature of the last query to be executed. GetQueryOutputXML and GetQueryOutputObject expose more complex query output.

Example:

```c
#$ visit -cli
OpenDatabase("/usr/gapps/visit/data/rect2d.silo")
AddPlot("Pseudocolor", "d")
DrawPlots()
Query("MinMax")
print(GetQueryOutputString())
print("The min is: %g and the max is: %g" % GetQueryOutputValue())
```

2.4.108 GetQueryOverTimeAttributes

Synopsis:

GetQueryOverTimeAttributes() -> QueryOverTimeAttributes object

return type [QueryOverTimeAttributes object] GetQueryOverTimeAttributes returns a QueryOverTimeAttributes object.

Description:

The GetQueryOverTimeAttributes function returns a QueryOverTimeAttributes object containing the settings that VisIt currently uses for query over time. You can use the returned object to change those settings by first setting object properties and then by passing the modified object to the SetQueryOverTimeAttributes function.

Example:

```c
#$ visit -cli
SetWindowLayout(4)
OpenDatabase("/usr/gapps/visit/data/allinone00.pdb")
AddPlot("Pseudocolor", "mesh/mixvar")
DrawPlots()
qot = GetQueryOverTimeAttributes()            # Make queries over time go to window 4.
print(qot)
qot.createWindow, q.windowId = 0, 4
SetQueryOverTimeAttributes(qot)              # Make queries over time only use half of the number of time states.
QueryOverTime("Min")
endTime = GetDatabaseNStates() / 2
QueryOverTime("Min", end_time=endTime)
ResetView()
```

2.4.109 GetQueryParameters

Synopsis:
GetQueryParameters(name) -> dictionary

**return type** [dictionary] A python dictionary.

**Description:**

The GetQueryParameters function returns a Python dictionary containing the default parameters for the named query, or None if the query does not accept additional parameters. The returned dictionary (if any) can then be modified if necessary and passed back as an argument to the Query function.

**Example:**

```python
#% visit -cli
minMaxInput = GetQueryParameters("MinMax")
minMaxInput["use_actual_data"] = 1
Query("MinMax", minMaxInput)
xrayInput = GetQueryParameters("XRay Image")
xrayInput["origin"]=(0.5, 2.5, 0.)
xrayInput["image_size"]=(300,300)
xrayInput["vars"]=("p", "d")
Query("XRay Image", xrayInput)
```

### 2.4.110 GetRenderingAttributes

**Synopsis:**

GetRenderingAttributes() -> RenderingAttributes object

**return type** [RenderingAttributes object] Returns a RenderingAttributes object.

**Description:**

The GetRenderingAttributes function returns a RenderingAttributes object that contains the rendering settings that VisIt currently uses. The RenderingAttributes object contains information related to rendering such as whether or not specular highlights or shadows are enabled. The RenderingAttributes object also contains information scalable rendering such as whether or not it is currently in use and the scalable rendering threshold.

**Example:**

```python
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/noise.silo")
AddPlot("Surface", "hgslice")
DrawPlots()
v = GetView3D()
v.viewNormal = (-0.215934, -0.454611, 0.864119)
v.viewUp = (0.973938, -0.163188, 0.157523)
v.imageZoom = 1.64765
SetView3D(v)
light = GetLight(0)
light.direction = (0,1,-1)
SetLight(0, light)
r = GetRenderingAttributes()
r.scalableActivationMode = r.Always
r.doShadowing = 1
SetRenderingAttributes(r)
```
2.4.111 GetSaveWindowAttributes

Synopsis:

```
GetSaveWindowAttributes() -> SaveWindowAttributes object
```

**return type** [SaveWindowAttributes object] This function returns a VisIt SaveWindowAttributes object that contains the attributes used in saving windows.

**Description:**

The GetSaveWindowAttributes function returns a SaveWindowAttributes object that is a structure containing several fields which determine how windows are saved to files. The object that is returned can be modified and used to set the save window attributes.

**Example:**

```python
# visit -cli
s = GetSaveWindowAttributes()
print(s)
s.width = 600
s.height = 600
s.format = s.RGB
print(s)
```

2.4.112 GetSelection

Synopsis:

```
GetSelection(name) -> SelectionProperties object
```

**name** [string] The name of the selection whose properties we want to retrieve.

**return type** [SelectionProperties object] The GetSelection function returns a SelectionProperties object.

**Description:**

Named selections have properties that describe how the selection is defined. This function lets you query those selection properties.

**Example:**

```python
CreateNamedSelection('selection1')
s = GetSelection('selection1')
s.selectionType = s.CumulativeQuerySelection
s.histogramType = s.HistogramMatches
s.combineRule = s.CombineOr
s.variables = ('temperature',)
s.variableMins = (2.9,)
s.variableMaxs = (3.1,)
UpdateNamedSelection('selection1', s)
```

2.4.113 GetSelectionList

Synopsis:
GetSelectionList() -> SelectionList object


Description:
VisIt maintains a list of named selections, which are sets of cells that are used to restrict the cells processed by other plots. This function returns a list of the selections that VisIt knows about, including their properties.

Example:
s = GetSelectionList()

2.4.114 GetSelectionSummary

Synopsis:
GetSelectionSummary(name) -> SelectionSummary object

name [string] The name of the selection whose summary we want to retrieve.

return type [SelectionSummary object] The GetSelectionSummary function returns a SelectionSummary object.

Description:
Named selections have both properties, which describe how the selection is defined, and a summary that describes the data that was processed while creating the selection. The selection summary object contains some statistics about the selection such as how many cells it contains and histograms of the various variables that were used in creating the selection.

Example:
print(GetSelectionSummary('selection1'))

2.4.115 GetTimeSliders

Synopsis:
GetTimeSliders() -> tuple of strings

return type [tuple of strings] GetTimeSliders returns a tuple of strings.

Description:
The GetTimeSliders function returns a tuple of strings containing the names of each of the available time sliders. The list of time sliders contains the names of any open time-varying database, all database correlations, and the keyframing time slider if VisIt is in keyframing mode.

Example:
#% visit -cli
path = "/usr/gapps/visit/data/"
dbs = (path + "/dbA00.pdb", path + "dbB00.pdb", path + "dbC00.pdb")
for db in dbs:
    OpenDatabase(db)
    AddPlot("FilledBoundary", "material(mesh)")
2.4.116 GetUltraScript

Synopsis:

GetUltraScript() -> string

return type  [string] The GetUltraScript function returns a filename.

Description:

Return the name of the file in use by the LoadUltra function. Normal users do not need to use this function.

2.4.117 GetView2D

Synopsis:

GetView2D() -> View2DAttributes object

return type  [View2DAttributes object] Object that represents the 2D view information.

Description:

The GetView functions return ViewAttributes objects which describe the current camera location. The GetView2D function should be called if the active visualization window contains 2D plots.

Example:

```python
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
DrawPlots()
# Change the view interactively using the mouse.
v0 = GetView3D()
# Change the view again using the mouse
v1 = GetView3D()
print(v0)
for i in range(0,20):
    t = float(i) / 19.
    v2 = (1. - t) * v1 + t * v0
SetView3D(v2) # Animate the view back to the first view.
```

2.4.118 GetView3D

Synopsis:

GetView3D() -> View3DAttributes object

return type  [View3DAttributes object] Object that represents the 3D view information.

Description:
The `GetView` functions return `ViewAttributes` objects which describe the current camera location. The `GetView3D` function should be called to get the view if the active visualization window contains 3D plots.

**Example:**

```python
# visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
DrawPlots()
# Change the view interactively using the mouse.
v0 = GetView3D()
# Change the view again using the mouse
v1 = GetView3D()
print(v0)
for i in range(0,20):
    t = float(i) / 19.
    v2 = (1. - t) * v1 + t * v0
    SetView3D(v2)  # Animate the view back to the first view.
```

### 2.4.119 GetViewAxisArray

**Synopsis:**

```python
GetViewAxisArray() -> ViewAxisArrayAttributes object
```

**return type** `[ViewAxisArrayAttributes object]` Object that represents the AxisArray view information.

**Description:**

The `GetView` functions return `ViewAttributes` objects which describe the current camera location. The `GetViewAxisArray` function should be called if the active visualization window contains axis-array plots.

**Example:**

```python
# visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
DrawPlots()
# Change the view interactively using the mouse.
v0 = GetView3D()
# Change the view again using the mouse
v1 = GetView3D()
print(v0)
for i in range(0,20):
    t = float(i) / 19.
    v2 = (1. - t) * v1 + t * v0
    SetView3D(v2)  # Animate the view back to the first view.
```

### 2.4.120 GetViewCurve

**Synopsis:**

```python
GetViewCurve() -> ViewCurveAttributes object
```

**return type** `[ViewCurveAttributes object]` Object that represents the curve view information.
Description:

The GetView functions return ViewAttributes objects which describe the current camera location. The GetViewCurve function should be called if the active visualization window contains 1D curve plots.

Example:

```python
# visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
DrawPlots()
# Change the view interactively using the mouse.
v0 = GetView3D()
# Change the view again using the mouse
v1 = GetView3D()
print(v0)
for i in range(0,20):
    t = float(i) / 19.
    v2 = (1. - t) * v1 + t * v0
    SetView3D(v2)  # Animate the view back to the first view.
```

2.4.121 GetWindowInformation

Synopsis:

```python
getWindowInformation() -> WindowInformation object
```

return type [WindowInformation object] The GetWindowInformation object returns a WindowInformation object.

Description:

The GetWindowInformation object returns a WindowInformation object that contains information about the active visualization window. The WindowInformation object contains the name of the active source, the active time slider index, the list of available time sliders and their current states, as well as certain window flags that determine whether a window’s view is locked, etc. Use the WindowInformation object if you need to query any of these types of information in your script to influence how it behaves.

Example:

```python
path = "/usr/gapps/visit/data/"
dbs = (path + "dbA00.pdb", path + "dbB00.pdb", path + "dbC00.pdb")
for db in dbs:
    OpenDatabase(db)
    AddPlot("FilledBoundary", "material(mesh)")
    DrawPlots()
CreateDatabaseCorrelation("common", dbs, 1)
# Get the list of available time sliders.
tsList = getWindowInformation().timeSliders
# Iterate through "time" on each time slider.
for ts in tsList:
    SetActiveTimeSlider(ts)
for state in range(TimeSliderGetNStates()):
    SetTimeSliderState(state)
# Print the window information to examine the other attributes
# that are available.
getWindowInformation()
```
2.4.122 HideActivePlots

Synopsis:

HideActivePlots() -> integer

return type [CLI_return_t] The HideActivePlots function returns an integer value of 1 for success and 0 for failure.

Description:
The HideActivePlots function tells the viewer to hide the active plots in the active visualization window.

Example:

```vis
#!/usr/bin/env python
import visit

visit.OpenDatabase("/usr/gapps/visit/data/globe.silo")
visit.AddPlot("Pseudocolor", "u")
visit.AddPlot("Mesh", "mesh1")
visit.DrawPlots()
visit.SetActivePlots(0)
visit.HideActivePlots()
visit.AddPlot("FilledBoundary", "mat1")
visit.DrawPlots()
```

2.4.123 HideToolbars

Synopsis:

HideToolbars() -> integer
HideToolbars(allWindows) -> integer

allWindows [integer] An optional integer value that tells VisIt to hide the toolbars for all windows when it is non-zero.

return type [CLI_return_t] The HideToolbars function returns 1 on success and 0 on failure.

Description:
The HideToolbars function tells VisIt to hide the toolbars for the active visualization window or for all visualization windows when the optional allWindows argument is provided and is set to a non-zero value.

Example:

```vis
#!/usr/bin/env python
import visit

visit.SetWindowLayout(4)
visit.HideToolbars()
visit.ShowToolbars()
visit.HideToolbars(1)
```

2.4.124 IconifyAllWindows

Synopsis:

IconifyAllWindows()

Description:

```vis
#!/usr/bin/env python
import visit

visit.IconifyAllWindows()
```
The IconifyAllWindows function minimizes all of the hidden visualization windows to get them out of the way.

Example:

```c
#define visit -cli
SetWindowLayout(4)  # Have 4 windows
IconifyAllWindows();
DeIconifyAllWindows();
```

### 2.4.125 InitializeNamedSelectionVariables

**Synopsis:**

`InitializeNamedSelectionVariables(name) -> integer`

**name** [string] The name of the named selection to initialize.

**return type** [CLI_return_t] The InitializeNamedSelectionVariables function returns 1 on success and 0 on failure.

**Description:**

Complex thresholds are often defined using the Parallel Coordinates plot or the Threshold operator. This function can copy variable ranges from compatible plots and operators into the specified named selection’s properties. This can be useful when setting up Cumulative Query selections.

Example:

```c
InitializeNamedSelectionVariables('selection1');
```

### 2.4.126 InvertBackgroundColor

**Synopsis:**

`InvertBackgroundColor();`

**Description:**

The InvertBackgroundColor function swaps the background and foreground colors in the active visualization window. This function is a cheap alternative to setting the foreground and background colors through the AnnotationAttributes in that it is a simple no-argument function call. It is not adequate to set new colors for the background and foreground, but in the event where the two colors can be exchanged favorably, it is a good function to use. An example of when this function is used is after the creation of a Volume plot.

Example:

```c
#define visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Volume", "u")
DrawPlots();
InvertBackgroundColor();
```
2.4.127 Launch

Synopsis:

Launch() -> integer  
Launch(program) -> integer

**program** [string] The complete path as a string to the top level ‘visit’ script.  
**return type** [CLI_return_t] The Launch functions return 1 for success and 0 for failure

**Description:**

The Launch function is used to launch VisIt’s viewer when the VisIt module is imported into a stand-alone Python interpreter. The Launch function has no effect when a viewer already exists. The difference between Launch and LaunchNowin is that LaunchNowin prevents the viewer from ever creating onscreen visualization windows. The LaunchNowin function is primarily used in Python scripts that want to generate visualizations using VisIt without the use of a display such as when generating movies.

**Example:**

```python
import visit
visit.AddArgument("-nowin")
visit.Launch()
```

2.4.128 LaunchNowin

Synopsis:

LaunchNowin() -> integer  
LaunchNowin(program) -> integer

**program** [string] The complete path as a string to the top level ‘visit’ script.  
**return type** [CLI_return_t] The LaunchNowin functions return 1 for success and 0 for failure

**Description:**

The Launch function is used to launch VisIt’s viewer when the VisIt module is imported into a stand-alone Python interpreter. The Launch function has no effect when a viewer already exists. The difference between Launch and LaunchNowin is that LaunchNowin prevents the viewer from ever creating onscreen visualization windows. The LaunchNowin function is primarily used in Python scripts that want to generate visualizations using VisIt without the use of a display such as when generating movies.

**Example:**

```python
import visit
visit.AddArgument("-geometry")
visit.AddArgument("1024x1024")
visit.LaunchNowin()
```

2.4.129 Lineout

Synopsis:
Lineout(start, end) -> integer
Lineout(start, end, variables) -> integer
Lineout(start, end, samples) -> integer
Lineout(start, end, variables, samples) -> integer
Lineout(keywordarg1=arg1, keywordarg2=arg2,..., keywordargn=argn ) -> integer

**Description:**

The Lineout function extracts data along a given line segment and creates curves from it in a new visualization window. The start argument is a tuple of numbers that make up the coordinate of the lineout’s starting location. The end argument is a tuple of numbers that make up the coordinate of the lineout’s ending location. The optional variables argument is a tuple of strings that contain the variables that should be sampled to create lineouts. The optional samples argument is used to determine the number of sample points that should be taken along the specified line. If the samples argument is not provided then VisIt will sample the mesh where it intersects the specified line instead of using the number of samples to compute a list of points to sample.

**Example:**

```c
# visit -cli
OpenDatabase("/usr/gapps/visit/data/rect2d.silo")
AddPlot("Pseudocolor", "ascii")
DrawPlots()
Lineout((0.2,0.2), (0.8,1.2))
Lineout((0.2,1.2), (0.8,0.2), ("default", "d", "u"))
Lineout((0.6, 0.1), (0.8, 1.2), 100)
Lineout(start_point=(0.6, 0.1), end_point=(0.8, 1.2), use_sampling=1, num_samples=100)
```

### 2.4.130 ListDomains

**Synopsis:**

ListDomains()

**Description:**

ListDomains prints a list of the domains for the active plots, which indicates which domains are on and off. The list functions are used mostly to print the results of restricting the SIL.

**Example:**

```c
# visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
DrawPlots()
```

(continues on next page)
2.4.131 ListMaterials

Synopsis:

ListMaterials()

Description:

ListMaterials prints a list of the materials for the active plots, which indicates which materials are on and off. The list functions are used mostly to print the results of restricting the SIL.

Example:

```python
# % visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
DrawPlots()
TurnMaterialsOff("4") # Turn off material 4
ListMaterials() # List the materials in the SIL restriction
```

2.4.132 ListPlots

Synopsis:

ListPlots() -> string
ListPlots(stringOnly) -> string

return type  [string] The ListPlots function returns a string containing a representation of the plot list.

Description:

Sometimes it is difficult to remember the order of the plots in the active visualization window’s plot list. The ListPlots function prints the contents of the plot list to the output console and returns that string as well.

Example:

```python
# % visit -cli
OpenDatabase("/usr/gapps/visit/data/curv2d.silo")
AddPlot("Pseudocolor", "u")
AddPlot("Contour", "d")
DrawPlots()
ListPlots()
```

2.4.133 LoadAttribute

Synopsis:

LoadAttribute(filename, object)

filename  [string] The name of the XML file to load the attribute from or save the attribute to.
**object**  The object to load or save.

**Description:**

The LoadAttribute and SaveAttribute methods save a single attribute, such as a current plot or operator python object, to a standalone XML file. Note that LoadAttribute requires that the target attribute already be created by other means; it fills, but does not create, the attribute.

**Example:**

```python
# visit -cli
a = MeshPlotAttributes()
SaveAttribute('mesh.xml', a)
b = MeshPlotAttributes()
LoadAttribute('mesh.xml', b)
```

### 2.4.134 **LoadNamedSelection**

**Synopsis:**

```python
LoadNamedSelection(name) -> integer
LoadNamedSelection(name, engineName) -> integer
LoadNamedSelection(name, engineName, simName) -> integer
```

**name**  [string] The name of a named selection.

**engineName**  [string] (optional) The name of the engine where the selection was saved.

**simName**  [string] (optional) The name of the simulation that saved the selection.

**return type**  [CLI_return_t] The LoadNamedSelection function returns 1 for success and 0 for failure.

**Description:**

Named Selections allow you to select a group of elements (or particles). One typically creates a named selection from a group of elements and then later applies the named selection to another plot (thus reducing the set of elements displayed to the ones from when the named selection was created). Named selections only last for the current session. However, if you find a named selection that is particularly interesting, you can save it to a file for use in later sessions. You would use LoadNamedSelection to do the loading.

**Example:**

```python
# visit -cli
db = "/usr/gapps/visit/data/wave*.silo database"
OpenDatabase(db)
AddPlot("Pseudocolor", "pressure")
LoadNamedSelection("selection_from_previous_session")
ApplyNamedSelection("selection_from_previous_session")
```

### 2.4.135 **LoadUltra**

**Synopsis:**

```python
LoadUltra()
```

**Description:**
LoadUltra launches the Ultra command parser, allowing you to enter Ultra commands and have VisIt process them. A new command prompt is presented, and only Ultra commands will be allowed until ‘end’ or ‘quit’ is entered, at which time, you will be returned to VisIt’s cli prompt. For information on currently supported commands, type ‘help’ at the Ultra prompt Please note that filenames/paths must be surrounded by quotes, unlike with Ultra.

**Example:**

```bash
# visit -cli
>>> LoadUltra()
U-> rd "../../../data/distribution.ultra"
U-> select 1
U-> end
>>> 
```

### 2.4.136 LocalNameSpace

**Synopsis:**

`LocalNameSpace()`

**Description:**

The LocalNameSpace function tells the VisIt module to add plugin functions to the global namespace when the VisIt module is imported into a stand-alone Python interpreter. This is the default behavior when using VisIt’s cli program.

**Example:**

```python
import visit
visit.LocalNameSpace()
visit.Launch()
```

### 2.4.137 LongFileName

**Synopsis:**

`LongFileName(filename) -> string`

**filename** [string] A string object containing the short filename to expand.

**return type** [string] The LongFileName function returns a string. This function returns the input argument unless you are on the Windows platform.

**Description:**

On Windows, filenames can have two different sizes: traditional 8.3 format, and long format. The long format, which lets you name files whatever you want, is implemented using the traditional 8.3 format under the covers. Sometimes filenames are given to VisIt in the traditional 8.3 format and must be expanded to long format before it is possible to open them. If you ever find that you need to do this conversion, such as when you process command line arguments, then you can use the LongFileName function to return the longer filename.
2.4.138 MoveAndResizeWindow

Synopsis:

```
MoveAndResizeWindow(win, x, y, w, h) -> integer
```

**win** [integer] The integer id of the window to be moved [1..16].

**x** [integer] The new integer x location for the window being moved.

**y** [integer] The new integer y location for the window being moved.

**w** [integer] The new integer width for the window being moved.

**h** [integer] The new integer height for the window being moved.

**return type** [CLI_return_t] MoveAndResizeWindow returns 1 on success and 0 on failure.

**Description:**

MoveAndResizeWindow moves and resizes a visualization window.

**Example:**

```c
#% visit -cli
MoveAndResizeWindow(1, 100, 100, 300, 600)
```

2.4.139 MovePlotDatabaseKeyframe

Synopsis:

```
MovePlotDatabaseKeyframe(index, oldFrame, newFrame)
```

**index** [integer] An integer representing the index of the plot in the plot list.

**oldFrame** [integer] An integer that is the old animation frame where the keyframe is located.

**newFrame** [integer] An integer that is the new animation frame where the keyframe will be moved.

**Description:**

MovePlotDatabaseKeyframe moves a database keyframe for a specified plot to a new animation frame, which changes the list of database time states that are used for each animation frame when VisIt is in keyframing mode.

**Example:**

```c
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/wave.visit")
k = GetKeyframeAttributes()
nFrames = 20
k.enabled, k.nFrames, k.nFramesWasUserSet = 1, nFrames, 1
AddPlot("Pseudocolor", "pressure")
SetPlotFrameRange(0, 0, nFrames-1)
SetPlotDatabaseKeyframe(0, 0, 70)
SetPlotDatabaseKeyframe(0, nFrames/2, 35)
SetPlotDatabaseKeyframe(0, nFrames-1, 0)
DrawPlots()
for state in list(range(TimeSliderGetNStates())) + [0]:
    SetTimeSliderState(state)
```

(continues on next page)
MovePlotDatabaseKeyframe(0, nFrames/2, nFrames/4)
for state in list(range(TimeSliderGetNStates())) + [0]:
    SetTimeSliderState(state)

2.4.140 MovePlotKeyframe

Synopsis:

MovePlotKeyframe(index, oldFrame, newFrame)

index  [integer]  An integer representing the index of the plot in the plot list.
oldFrame  [integer]  An integer that is the old animation frame where the keyframe is located.
newFrame  [integer]  An integer that is the new animation frame where the keyframe will be moved.

Description:

MovePlotKeyframe moves a keyframe for a specified plot to a new animation frame, which changes the plot attributes that are used for each animation frame when VisIt is in keyframing mode.

Example:

```python
# visit -cli
OpenDatabase("/usr/gapps/visit/data/noise.silo")
AddPlot("Contour", "hgslice")
DrawPlots()
k = GetKeyframeAttributes()
nFrames = 20
k.enabled, k.nFrames, k.nFramesWasUserSet = 1, nFrames, 1
SetKeyframeAttributes(k)
SetPlotFrameRange(0, 0, nFrames-1)
c = ContourAttributes()
c.contourNLevels = 5
SetPlotOptions(c)
SetTimeSliderState(nFrames/2)
c.contourNLevels = 10
SetPlotOptions(c)
c.contourLevels = 25
SetTimeSliderState(nFrames-1)
SetPlotOptions(c)
for state in range(TimeSliderGetNStates()):
    SetTimeSliderState(state)
    SaveWindow()
temp = nFrames-2
MovePlotKeyframe(0, nFrames/2, temp)
MovePlotKeyframe(0, nFrames-1, nFrames/2)
MovePlotKeyframe(0, temp, nFrames-1)
for state in range(TimeSliderGetNStates()):
    SetTimeSliderState(state)
    SaveWindow()
```

2.4.141 MovePlotOrderTowardFirst

Synopsis:
MovePlotOrderTowardFirst(index) -> integer

**index** [integer] The integer index of the plot that will be moved within the plot list.

**return type** [CLI_return_t] The MovePlotOrderTowardFirst function returns 1 on success and 0 on failure.

**Description:**
This function shifts the specified plot one slot towards the start of the plot list.

**Example:**

```
MovePlotOrderTowardFirst(2)
```

### 2.4.142 MovePlotOrderTowardLast

**Synopsis:**

MovePlotOrderTowardLast(index) -> integer

**index** [integer] The integer index of the plot that will be moved within the plot list.

**return type** [CLI_return_t] The MovePlotOrderTowardLast function returns 1 on success and 0 on failure.

**Description:**
This function shifts the specified plot one slot towards the end of the plot list.

**Example:**

```
MovePlotOrderTowardLast(0)
```

### 2.4.143 MoveViewKeyframe

**Synopsis:**

MoveViewKeyframe(oldFrame, newFrame) -> integer

**oldFrame** [integer] An integer that is the old animation frame where the keyframe is located.

**newFrame** [integer] An integer that is the new animation frame where the keyframe will be moved.

**return type** [CLI_return_t] MoveViewKeyframe returns 1 on success and 0 on failure.

**Description:**
MoveViewKeyframe moves a view keyframe to a new animation frame, which changes the view that is used for each animation frame when VisIt is in keyframing mode.

**Example:**

```bash
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/noise.silo")
AddPlot("Contour", "hardyglobal")
DrawPlots()
k = GetKeyframeAttributes()
nFrames = 20
k.enabled, k.nFrames, k.nFramesWasUserSet = 1, nFrames, 1
```

(continues on next page)
SetKeyframeAttributes(k)
SetViewKeyframe()
SetTimeSliderState(nFrames/2)
v = GetView3d()
v.viewNormal = (-0.616518, 0.676972, 0.402014)
v.viewUp = (0.49808, 0.730785, -0.466764)
SetViewKeyframe()
SetTimeSliderState(0)

# Move the view keyframe to the last animation frame.
MoveViewKeyframe(nFrames/2, nFrames-1)

2.4.144 MoveWindow

Synopsis:

MoveWindow(win, x, y) -> integer

win [integer] The integer id of the window to be moved [1..16].
x [integer] The new integer x location for the window being moved.
y [integer] The new integer y location for the window being moved.

return type [CLI_return_t] MoveWindow returns 1 on success and 0 on failure.

Description:

MoveWindow moves a visualization window.

Example:

```c
#!/usr/bin/env python

# visit -cli
MoveWindow(1, 100, 100)
```

2.4.145 NodePick

Synopsis:

NodePick(namedarg1=arg1, namedarg2=arg2, ...) -> dictionary

coord [tuple] A tuple of doubles containing the spatial coordinate (x, y, z).
x [integer] An integer containing the screen X location (in pixels) offset from the left side of the visualization window.
y [integer] An integer containing the screen Y location (in pixels) offset from the bottom of the visualization window.
vars (optional) [tuple] A tuple of strings with the variable names for which to return results. Default is the currently plotted variable.
do_time (optional) [integer] An integer indicating whether to do a time pick. 1 -> do a time pick, 0 (default) -> do not do a time pick.
start_time (optional) [integer] An integer with the starting frame index. Default is 0.
end_time (optional) [integer] An integer with the ending frame index. Default is num_timestates-1.
stride (optional) [integer] An integer with the stride for advancing in time. Default is 1.
**preserve_coord (optional)** [integer] An integer indicating whether to pick an element or a coordinate. 0 -> used picked element (default), 1 -> used picked coordinate.

**curve_plot_type (optional)** [integer] An integer indicating whether the output should be on a single axis or with multiple axes. 0 -> single Y axis (default), 1 -> multiple Y Axes.

**return type** [dictionary] NodePick returns a python dictionary of the pick results, unless do_time is specified, then a time curve is created in a new window.

**Description:**

The NodePick function prints pick information for the node closest to the specified point. The point can be specified as a 2D or 3D point in world space or it can be specified as a pixel location in screen space. If the point is specified as a pixel location then VisIt finds the node closest to a ray that is projected into the mesh. Once the nodal pick has been calculated, you can use the GetPickOutput function to retrieve the printed pick output as a string which can be used for other purposes.

**Example:**

```plaintext
#!/usr/bin/env python
import visit

OpenDatabase("/usr/gapps/visit/data/noise.silo")
AddPlot("Pseudocolor", "hgslice")
DrawPlots()

# Perform node pick in screen space
pick_out = NodePick(x=200,y=200)

# Perform node pick in world space.
pick_out = NodePick(coord=(-5.0, 5.0, 0))
```

### 2.4.146 NumColorTableNames

**Synopsis:**

`NumColorTableNames() -> integer`

**return type** [CLI_return_t] The NumColorTableNames function return an integer.

**Description:**

The NumColorTableNames function returns the number of color tables that have been defined.

**Example:**

```plaintext
#!/usr/bin/env python
import visit

OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
p = PseudocolorAttributes()
p.colorTableName = "default"
SetPlotOptions(p)
DrawPlots()
print("There are \$d color tables." % NumColorTableNames())
for ct in ColorTableNames():
    SetActiveContinuousColorTable(ct)
    SaveWindow()
```

### 2.4.147 NumOperatorPlugins

**Synopsis:**

The NumOperatorPlugins function returns the number of operator plugins that have been defined.
VisIt User Manual Documentation, Release 3.1

**NumOperatorPlugins()** -> integer

**return type** [CLI_return_t] The NumOperatorPlugins function returns an integer.

**Description:**

The NumOperatorPlugins function returns the number of available operator plugins.

**Example:**

```python
#% visit -cli
print("The number of operator plugins is: ", NumOperatorPlugins())
print("The names of the plugins are: ", OperatorPlugins())
```

---

### 2.4.148 NumPlotPlugins

**Synopsis:**

NumPlotPlugins() -> integer

**return type** [CLI_return_t] The NumPlotPlugins function returns an integer.

**Description:**

The NumPlotPlugins function returns the number of available plot plugins.

**Example:**

```python
#% visit -cli
print("The number of plot plugins is: ", NumPlotPlugins())
print("The names of the plugins are: ", PlotPlugins())
```

---

### 2.4.149 OpenComputeEngine

**Synopsis:**

OpenComputeEngine() -> integer
OpenComputeEngine(hostName) -> integer
OpenComputeEngine(hostName, simulation) -> integer
OpenComputeEngine(hostName, args) -> integer
OpenComputeEngine(MachineProfile) -> integer

**hostName** [string] The name of the computer on which to start the engine.

**args** [tuple] Optional tuple of command line arguments for the engine. Alternative arguments - MachineProfile object to load with OpenComputeEngine call

**return type** [CLI_return_t] The OpenComputeEngine function returns an integer value of 1 for success and 0 for failure.

**Description:**

The OpenComputeEngine function is used to explicitly open a compute engine with certain properties. When a compute engine is opened implicitly, the viewer relies on sets of attributes called host profiles. Host profiles determine how compute engines are launched. This allows compute engines to be easily launched in parallel. Since the VisIt Python Interface does not expose VisIt’s host profiles, it provides the OpenComputeEngine function to allow users to launch compute engines. The OpenComputeEngine...
function must be called before opening a database in order to prevent any latent host profiles from taking precedence.

Example:

```python
# Launch parallel compute engine remotely.
args = ("-np", "16", "-nn", "4")
OpenComputeEngine("thunder", args)
OpenDatabase("thunder:/usr/gapps/visit/data/multi_ucd3d.silo")
AddPlot("Pseudocolor", "d")
DrawPlots()
```

### 2.4.150 OpenDatabase

**Synopsis:**

OpenDatabase(databaseName) -> integer
OpenDatabase(databaseName, timeIndex) -> integer
OpenDatabase(databaseName, timeIndex, dbPluginName) -> integer

- **databaseName** [string] The name of the database to open.
- **timeIndex** [integer] This is an optional integer argument indicating the time index at which to open the database. If it is not specified, a time index of zero is assumed.
- **dbPluginName** [string] An optional string containing the name of the plugin to use. Note that this string must also include the plugin’s version number (with few exceptions, almost all plugins’ version numbers are 1.0). Note also that you must capitalize the spelling identically to what the plugin’s GetName() method returns. For example, “XYZ_1.0” is the string you would use for the XYZ plugin.

**Description:**

The OpenDatabase function is one of the most important functions in the VisIt Python Interface because it opens a database so it can be plotted. The databaseName argument is a string containing the full name of the database to be opened. The database name is of the form: computer:/path/filename. The computer part of the filename can be omitted if the database to be opened resides on the local computer.

Example:

```python
# OpenDatabase("/usr/gapps/visit/data/globe.silo")
OpenDatabase("mcr:/usr/gapps/visit/data/multi_ucd3d.silo")
OpenDatabase("file.visit")
OpenDatabase("mcr:/usr/gapps/visit/data/multi_ucd3d.silo", 0,"Silo_1.0")
```

### 2.4.151 OpenMDServer

**Synopsis:**

OpenMDServer() -> integer
OpenMDServer(host) -> integer
OpenMDServer(host, args) -> integer
OpenMDServer(MachineProfile) -> integer
**host** [string] The optional host argument determines the host on which the metadata server is to be launched. If this argument is not provided, “localhost” is assumed.

**args** [tuple] A tuple of strings containing command line flags for the metadata server.

<table>
<thead>
<tr>
<th>Argument</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>-debug #</td>
<td>The -debug argument allows you to specify a debug level.</td>
</tr>
<tr>
<td>-dir visitdir</td>
<td>The -dir argument allows you to specify where VisIt is.</td>
</tr>
</tbody>
</table>

**MachineProfile** [MachineProfile object] MachineProfile object to load with OpenMDServer call

**return type** [CLI_return_t] The OpenMDServer function returns 1 on success and 0 on failure.

**Description:**

The OpenMDServer explicitly launches a metadata server on a specified host. This allows you to provide command line options that influence how the metadata server will run. range [1,5] that VisIt uses to write debug logs to disk, located on a remote computer. This allows you to successfully connect to a remote computer in the absence of host profiles. It also allows you to debug VisIt in distributed mode.

- **-fallback_format <format>** The -fallback_format argument allows you to specify the database plugin that will be used to open files if all other guessing failed. This is useful when the files that you want to open do not have file extensions. -**-assume_format <format>** The -assume_format argument allows you to specify the database plugin that will be used FIRST when attempting to open files. This is useful when the files that you want to open have a file extension which may match multiple file format readers.

**Example:**

```bash
#!/usr/bin/env python
import sys
from visilibity import *

# Open a metadata server before the call to OpenDatabase so we can launch it how we want.
OpenMDServer("thunder", args)
OpenDatabase("thunder:/usr/gapps/visit/data/allinone00.pdb")
OpenMDServer()```

**2.4.152 OperatorPlugins**

**Synopsis:**

`OperatorPlugins() -> tuple of strings`

**return type** [tuple of strings] The OperatorPlugins function returns a tuple of strings.

**Description:**

The OperatorPlugins function returns a tuple of strings that contain the names of the loaded operator plugins. This can be useful for the creation of scripts that alter their behavior based on the available operator plugins.

**Example:**

```bash
#!/usr/bin/env python
import sys
from visilibity import *

#% visit -cli
for plugin in OperatorPlugins():
    print("The %s operator plugin is loaded." % plugin)```
### 2.4.153 OverlayDatabase

**Synopsis:**

```
OverlayDatabase(databaseName) -> integer
OverlayDatabase(databaseName, state) -> integer
```

databaseName [string] The name of the new plot database.
state The time state at which to open the database.

**return type** [CLI_return_t] The OverlayDatabase function returns an integer value of 1 for success and 0 for failure.

**Description:**

VisIt has the concept of overlaying plots which, in the nutshell, means that the entire plot list is copied and a new set of plots with exactly the same attributes but a different database is appended to the plot list of the active window. The OverlayDatabase function allows the VisIt Python Interface to overlay plots. OverlayDatabase takes a single string argument which contains the name of the database. After calling the OverlayDatabase function, the plot list is larger and contains plots of the specified overlay database.

**Example:**

```bash
# % visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
DrawPlots()
OverlayDatabase("riptide:/usr/gapps/visit/data/curv3d.silo")
```

### 2.4.154 PickByGlobalNode

**Synopsis:**

```
PickByGlobalNode(namedarg1=arg1, namedarg2=arg2, ...) -> dictionary
```

element [integer] An integer with the global node id.
vars (optional) [tuple] A tuple of strings with the variable names for which to return results. Default is the currently plotted variable.
do_time (optional) [integer] An integer indicating whether to do a time pick. 1 -> do a time pick, 0 (default) -> do not do a time pick.
start_time (optional) [integer] An integer with the starting frame index. Default is 0.
end_time (optional) [integer] An integer with the ending frame index. Default is num_timestates-1.
stride (optional) [integer] An integer with the stride for advancing in time. Default is 1.
preserve_coord (optional) [integer] An integer indicating whether to pick an element or a coordinate. 0 -> used picked element (default), 1 -> used picked coordinate.
curve_plot_type (optional) [integer] An integer indicating whether the output should be on a single axis or with multiple axes. 0 -> single Y axis (default), 1 -> multiple Y Axes.

**return type** [dictionary] PickByGlobalNode returns a python dictionary of pick results.

**Description:**

The PickByGlobalNode function tells VisIt to perform pick using a specific global node index for the entire problem. Some meshes are broken up into smaller “domains” and then these smaller domains can
employ a global indexing scheme to make it appear as though the mesh was still one large mesh. Not all meshes that have been decomposed into domains provide sufficient information to allow global node indexing. You can use the GetPickOutput function to retrieve a string containing the pick information once you’ve called PickByGlobalNode.

Example:

```python
# visit -cli
OpenDatabase("/usr/gapps/visit/data/global_node.silo")
AddPlot("Pseudocolor", "dist")
DrawPlots()
# Pick on global node 236827
pick_out = PickByGlobalNode(element=246827)
# examine output
print('value of dist at global node 246827: %g' % pick_out['dist'])
print('local domain/node: %d/%d' % (pick_out['domain_id'], pick_out['node_id']))
# get last pick output as string
print('Last pick = ', GetPickOutput())
```

2.4.155 PickByGlobalZone

Synopsis:

```
PickByGlobalZone(namedarg1=arg1, namedarg2=arg2, ...) -> dictionary
```

<table>
<thead>
<tr>
<th>element</th>
<th>[integer] An integer with the global zone id.</th>
</tr>
</thead>
<tbody>
<tr>
<td>vars (optional)</td>
<td>[tuple] A tuple of strings with the variable names for which to return results. Default is the currently plotted variable.</td>
</tr>
<tr>
<td>do_time (optional)</td>
<td>[integer] An integer indicating whether to do a time pick. 1 -&gt; do a time pick, 0 (default) -&gt; do not do a time pick.</td>
</tr>
<tr>
<td>start_time (optional)</td>
<td>[integer] An integer with the starting frame index. Default is 0.</td>
</tr>
<tr>
<td>end_time (optional)</td>
<td>[integer] An integer with the ending frame index. Default is num_timestates-1.</td>
</tr>
<tr>
<td>stride (optional)</td>
<td>[integer] An integer with the stride for advancing in time. Default is 1.</td>
</tr>
<tr>
<td>preserve_coord (optional)</td>
<td>[integer] An integer indicating whether to pick an element or a coordinate. 0 -&gt; used picked element (default), 1 -&gt; used picked coordinate.</td>
</tr>
<tr>
<td>curve_plot_type (optional)</td>
<td>[integer] An integer indicating whether the output should be on a single axis or with multiple axes. 0 -&gt; single Y axis (default), 1 -&gt; multiple Y Axes.</td>
</tr>
<tr>
<td>return type</td>
<td>[dictionary] PickByGlobalZone returns a python dictionary of pick results.</td>
</tr>
</tbody>
</table>

Description:

The PickByGlobalZone function tells VisIt to perform pick using a specific global cell index for the entire problem. Some meshes are broken up into smaller “domains” and then these smaller domains can employ a global indexing scheme to make it appear as though the mesh was still one large mesh. Not all meshes that have been decomposed into domains provide sufficient information to allow global cell indexing. You can use the GetPickOutput function to retrieve a string containing the pick information once you’ve called PickByGlobalZone.

Example:
OpenDatabase("/usr/gapps/visit/data/global_node.silo")
AddPlot("Pseudocolor", "p")
DrawPlots()
# Pick on global zone 237394
pick_out = PickByGlobalZone(element=237394)
# examine output
print('value of p at global zone 237394: %g' % pick_out['p'])
print('local domain/zone: %d/%d' % (pick_out['domain_id'], pick_out['zone_id']))
# get last pick output as string
print('Last pick = ', GetPickOutput())

2.4.156 PickByNode

Synopsis:

```
PickByNode(namedarg1=arg1, namedarg2=arg2, ...) -> dictionary
```

domain [integer] An integer with the domain id.
element [integer] An integer with the node id.
vars (optional) [tuple] A tuple of strings with the variable names for which to return results. Default is the currently plotted variable.
do_time (optional) [integer] An integer indicating whether to do a time pick. 1 -> do a time pick, 0 (default) -> do not do a time pick.
start_time (optional) [integer] An integer with the starting frame index. Default is 0.
end_time (optional) [integer] An integer with the ending frame index. Default is num_timesteps-1.
stride (optional) [integer] An integer with the stride for advancing in time. Default is 1.
preserve_coord (optional) [integer] An integer indicating whether to pick an element or a coordinate. 0 -> used picked element (default), 1-> used picked coordinate.
curve_plot_type (optional) [integer] An integer indicating whether the output should be on a single axis or with multiple axes. 0 -> single Y axis (default), 1 -> multiple Y Axes. Currently, this is only available when performing a pick range.
return type [dictionary] PickByNode returns a python dictionary of the pick results, unless do_time is specified, then a time curve is created in a new window. If the picked variable is zone centered, the variable values are grouped according to incident zone ids.

Description:
The PickByNode function tells VisIt to perform pick using a specific node index in a given domain. Other pick by node variants first determine the node that is closest to some user-specified 3D point but the PickByNode functions cuts out this step and allows you to directly pick on the node of your choice. You can use the GetPickOutput function to retrieve a string containing the pick information once you’ve called PickByNode.

Example:
```
#!/usr/bin/env python

OpenDatabase("/usr/gapps/visit/data/multi_curv2d.silo")
AddPlot("Pseudocolor", "u")
DrawPlots()
# Pick on node 200 in the first domain.

## visit -cli
```
2.4.157 PickByNodeLabel

Synopsis:

| PickByNodeLabel(namedarg1=arg1, namedarg2=arg2, ...) -> dictionary |

- **element_label** [string] An string with the label of the node to pick.
- **vars** [optional] [tuple] A tuple of strings with the variable names for which to return results. Default is the currently plotted variable.
- **do_time** [optional] [integer] An integer indicating whether to do a time pick. 1 -> do a time pick, 0 (default) -> do not do a time pick.
- **start_time** [optional] [integer] An integer with the starting frame index. Default is 0.
- **end_time** [optional] [integer] An integer with the ending frame index. Default is num_timestates-1.
- **stride** [optional] [integer] An integer with the stride for advancing in time. Default is 1.
- **preserve_coord** [optional] [integer] An integer indicating whether to pick an element or a coordinate. 0 -> used picked element (default), 1 -> used picked coordinate.
- **curve_plot_type** [optional] [integer] An integer indicating whether the output should be on a single axis or with multiple axes. 0 -> single Y axis (default), 1 -> multiple Y Axes.
- **return** [dictionary] PickByNodeLabel returns a python dictionary of the pick results, unless do_time is specified, then a time curve is created in a new window. If the picked variable is node centered, the variable values are grouped according to incident node ids.

Description:

The PickByNodeLabel function tells VisIt to perform pick using a specific cell label. Other pick by zone variants first determine the cell that contains some user-specified 3D point but the PickByZone functions cuts out this step and allows you to directly pick on the cell of your choice. You can use the GetPickOutput function to retrieve a string containing the pick information once you've called PickByZone.

Example:

```python
# visit -cli
OpenDatabase("/usr/gapps/visit/data/multi_curv2d.silo")
AddPlot("Pseudocolor", "d")
DrawPlots()
# Pick on node labeled "node 4".
pick_out = PickByNodeLabel(element_label="node 4")
(continues on next page)```
# Pick on cell labeled "node 4" using a python dictionary.

```python
opts = {}
opts["element_label"] = "node 4"
pick_out = PickByNodeLabel(opts)
# examine output
print('value of d at "node 4": %g' % pick_out["d"])
# Pick on node labeled "node 12" return information for two additional
# variables.
pick_out = PickByNodeLabel(element_label="node 12", vars=("d", "u", "v"))
# examine output
print('incident nodes for "node 12": ', pick_out['incident_nodes'])
print('values of u at incident node %d: %g' % (pick_out['incident_nodes'][0], pick_out['u'][str(pick_out['incident_zones'][0])]))
# print results formatted as string
print("Last pick = ", GetPickOutput())
```

## 2.4.158 PickByZone

### Synopsis:

```python
PickByZone(namedarg1=arg1, namedarg2=arg2, ...) -> dictionary
```

- **domain** [integer] An integer with the domain id.
- **element** [integer] An integer with the zone id.
- **vars (optional)** [tuple] A tuple of strings with the variable names for which to return results. Default is the currently plotted variable.
- **do_time (optional)** [integer] An integer indicating whether to do a time pick. 1 -> do a time pick, 0 (default) -> do not do a time pick.
- **start_time (optional)** [integer] An integer with the starting frame index. Default is 0.
- **end_time (optional)** [integer] An integer with the ending frame index. Default is num_timesteps-1.
- **stride (optional)** [integer] An integer with the stride for advancing in time. Default is 1.
- **preserve_coord (optional)** [integer] An integer indicating whether to pick an element or a coordinate. 0 -> used picked element (default), 1-> used picked coordinate.
- **curve_plot_type (optional)** [integer] An integer indicating whether the output should be on a single axis or with multiple axes. 0 -> single Y axis (default), 1 -> multiple Y Axes. Currently, this is only available when performing a pick range.
- **return type** [dictionary] PickByZone returns a python dictionary of the pick results, unless do_time is specified, then a time curve is created in a new window. If the picked variable is node centered, the variable values are grouped according to incident node ids.

### Description:

The PickByZone function tells VisIt to perform pick using a specific cell index in a given domain. Other pick by zone variants first determine the cell that contains some user-specified 3D point but the PickByZone functions cuts out this step and allows you to directly pick on the cell of your choice. You can use the GetPickOutput function to retrieve a string containing the pick information once you’ve called PickByZone.

### Example:
# visit -cli
OpenDatabase("/usr/gapps/visit/data/multi_curv2d.silo")
AddPlot("Pseudocolor", "d")
DrawPlots()
# Pick on cell 200 in the second domain.
pick_out = PickByZone(element=200, domain=2)
# examine output
print('value of d at zone 200: %g' % pick_out['d'])
# Pick on cell 100 in domain 5 and return information for two additional variables.
pick_out = PickByZone(element=100, domain=5, vars=('d', "u", "v"))
# examine output
print('incident nodes for zone 100: ', pick_out['incident_nodes'])
print('values of u at incident zone %d: %g' % (pick_out['incident_nodes'][0], pick_out['u'][str(pick_out['incident_zones'][0])]))
# print results formatted as string
print("Last pick = ", GetPickOutput())

## 2.4.159 PickByZoneLabel

**Synopsis:**

```
PickByZoneLabel(namedarg1=arg1, namedarg2=arg2, ...) -> dictionary
```

- **element_label** [string] An string with the label of the zone to pick.
- **vars (optional)** [tuple] A tuple of strings with the variable names for which to return results. Default is the currently plotted variable.
- **do_time (optional)** [integer] An integer indicating whether to do a time pick. 1 -> do a time pick, 0 (default) -> do not do a time pick.
- **start_time (optional)** [integer] An integer with the starting frame index. Default is 0.
- **end_time (optional)** [integer] An integer with the ending frame index. Default is num_timesteps-1.
- **stride (optional)** [integer] An integer with the stride for advancing in time. Default is 1.
- **preserve_coord (optional)** [integer] An integer indicating whether to pick an element or a coordinate. 0 -> used picked element (default), 1-> used picked coordinate.
- **curve_plot_type (optional)** [integer] An integer indicating whether the output should be on a single axis or with multiple axes. 0 -> single Y axis (default), 1 -> multiple Y Axes.
- **return_type** [dictionary] PickByZoneLabel returns a python dictionary of the pick results, unless do_time is specified, then a time curve is created in a new window. If the picked variable is node centered, the variable values are grouped according to incident node ids.

**Description:**

The PickByZoneLabel function tells VisIt to perform pick using a specific cell label. Other pick by zone variants first determine the cell that contains some user-specified 3D point but the PickByZone functions cuts out this step and allows you to directly pick on the cell of your choice. You can use the GetPickOutput function to retrieve a string containing the pick information once you’ve called PickByZone.

**Example:**
2.4.160 PlotPlugins

Synopsis:

PlotPlugins() -> tuple of strings

return type [tuple of strings] The PlotPlugins function returns a tuple of strings.

Description:

The PlotPlugins function returns a tuple of strings that contain the names of the loaded plot plugins. This can be useful for the creation of scripts that alter their behavior based on the available plot plugins.

Example:

```bash
# visit -cli
for plugin in PluginPlugins():
    print("The \$s plot plugin is loaded." % plugin)
```

2.4.161 PointPick

Synopsis:

PointPick(namedarg1=arg1, namedarg2=arg2, ...) -> dictionary

coord [tuple] A tuple of doubles containing the spatial coordinate (x, y, z).

x [integer] An integer containing the screen X location (in pixels) offset from the left side of the visualization window.

y [integer] An integer containing the screen Y location (in pixels) offset from the bottom of the visualization window.

vars (optional) [tuple] A tuple of strings with the variable names for which to return results. Default is the currently plotted variable.
do_time (optional) [integer] An integer indicating whether to do a time pick. 1 -> do a time pick, 0 (default) -> do not do a time pick.

start_time (optional) [integer] An integer with the starting frame index. Default is 0.

end_time (optional) [integer] An integer with the ending frame index. Default is num_timestates-1.

stride (optional) [integer] An integer with the stride for advancing in time. Default is 1.

preserve_coord (optional) [integer] An integer indicating whether to pick an element or a coordinate. 0 -> used picked element (default), 1-> used picked coordinate.

curve_plot_type (optional) [integer] An integer indicating whether the output should be on a single axis or with multiple axes. 0 -> single Y axis (default), 1 -> multiple Y Axes.

return type [dictionary] PointPick returns a python dictionary of the pick results, unless do_time is specified, then a time curve is created in a new window.

Description:
The PointPick function prints pick information for the node closest to the specified point. The point can be specified as a 2D or 3D point in world space or it can be specified as a pixel location in screen space. If the point is specified as a pixel location then VisIt finds the node closest to a ray that is projected into the mesh. Once the nodal pick has been calculated, you can use the GetPickOutput function to retrieve the printed pick output as a string which can be used for other purposes.

Example:

```python
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/noise.silo")
AddPlot("Pseudocolor", "hgslice")
DrawPlots()
# Perform node pick in screen space
pick_out = PointPick(x=200,y=200)
# Perform node pick in world space.
pick_out = PointPick(coord=(-5.0, 5.0, 0))
```

2.4.162 PrintWindow

Synopsis:

PrintWindow() -> integer

return type [CLI_return_t] The PrintWindow function returns an integer value of 1 for success and 0 for failure.

Description:
The PrintWindow function tells the viewer to print the image in the active visualization window using the current printer settings.

Example:

```python
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/curv2d.silo")
AddPlot("Pseudocolor", "d")
AddPlot("Contour", "u")
DrawPlots()
PrintWindow()
```
2.4.163 PromoteOperator

Synopsis:

<table>
<thead>
<tr>
<th>Function</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>PromoteOperator(opIndex) -&gt; integer</td>
<td>A zero-based integer corresponding to the operator that should be promoted.</td>
</tr>
<tr>
<td>PromoteOperator(opIndex, applyToAllPlots) -&gt; integer</td>
<td>An integer flag that causes all plots in the plot list to be affected when it is non-zero.</td>
</tr>
</tbody>
</table>

**opIndex** [integer] A zero-based integer corresponding to the operator that should be promoted.

**applyToAllPlots** [integer] An integer flag that causes all plots in the plot list to be affected when it is non-zero.

**return type** [CLI_return_t] PromoteOperator returns 1 on success and 0 on failure.

**Description:**

The PromoteOperator function moves an operator closer to the end of the visualization pipeline. This allows you to change the order of operators that have been applied to a plot without having to remove them from the plot. For example, consider moving a Slice to after a Reflect operator when it had been the other way around. Changing the order of operators can result in vastly different results for a plot. The opposite function is DemoteOperator.

**Example:**

```python
# visit -cli
OpenDatabase("/usr/gapps/visit/data/noise.silo")
AddPlot("Pseudocolor", "hardyglobal")
AddOperator("Slice")
s = SliceAttributes()
s.project2d = 0
s.originPoint = (0,5,0)
s.originType=s.Point
s.normal = (0,1,0)
s.upAxis = (-1,0,0)
SetOperatorOptions(s)
AddOperator("Reflect")
DrawPlots()
# Now slice after reflect. We'll only get 1 slice plane instead of 2.
PromoteOperator(0)
DrawPlots()
```

2.4.164 PythonQuery

Synopsis:

<table>
<thead>
<tr>
<th>Function</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>PythonQuery(source='python filter source ...') -&gt; integer</td>
<td>A string containing the source code for a Python Query Filter</td>
</tr>
<tr>
<td>PythonQuery(file='path/to/python_filter_script.py') -&gt; integer</td>
<td>A string containing the path to a Python Query Filter script file. Note: Use only one of the ‘source’ or ‘file’ arguments. If both are used, the ‘source’ argument overrides ‘file’.</td>
</tr>
</tbody>
</table>

**source** [string] A string containing the source code for a Python Query Filter.

**file** [string] A string containing the path to a Python Query Filter script file. Note: Use only one of the ‘source’ or ‘file’ arguments. If both are used, the ‘source’ argument overrides ‘file’.

**return type** [CLI_return_t] The PythonQuery function returns 1 on success and 0 on failure.

**Description:**

Used to execute a Python Filter Query.
2.4.165 Queries

Synopsis:

Queries() -> tuple of strings

return type [tuple of strings] The Queries function returns a tuple of strings.

Description:

The Queries function returns a tuple of strings that contain the names of all of VisIt’s supported queries.

Example:

```python
#% visit -cli
print("supported queries: ", Queries())
```

2.4.166 QueriesOverTime

Synopsis:

QueriesOverTime() -> tuple of strings

return type [tuple of strings] Returns a tuple of strings.

Description:

The QueriesOverTime function returns a tuple of strings that contains the names of all of the VisIt queries that can be executed over time.

Example:

```python
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/allineone00.pdb")
AddPlot("Pseudocolor", "mesh/mixvar")
DrawPlots()
# Execute each of the queries over time on the plots.
for q in QueriesOverTime():
    QueryOverTime(q)
# You can control timestates used in the query via start_time, end_time, and stride as follows:
QueryOverTime("Volume", start_time=5, end_time=250, stride=5)
# (Defaults used if not specified are 0, nStates, 1)
```

2.4.167 Query

Synopsis:

Query(name) -> string
Query(name, dict) -> string
Query(name, namedarg1=arg1, namedarg2=arg2, ...) -> string
Query(name) -> double, tuple of double
Query(name, dict) -> double, tuple of double
Query(name, namedarg1=arg1, namedarg2=arg2, ...) -> double, tuple of double
Query(name) -> dictionary
Query(name, dict) -> dictionary
Query(name, namedarg1=arg1, namedarg2=arg2, ...) -> dictionary
name [string] The name of the query to execute.

dict [dictionary] An optional dictionary containing additional query arguments. namedarg1, namedarg2,... An optional list of named arguments supplying additional query parameters.

return type [see SetQueryOutputToXXX() functions] The Query function returns either a String (default), Value(s), or Object. The return type can be customized via calls to SetQueryOutputToXXX(), where ‘XXX’ is ‘String’, ‘Value’, or ‘Object’. For more information on these return types, see ‘GetQueryOutput’.

Description:
The Query function is used to execute any of VisIt’s predefined queries. The list of queries can be found in the VisIt User’s Manual in the Quantitative Analysis chapter. You can get also get a list of queries using ‘Queries’ function. Since queries can take a wide array of arguments, the Query function takes either a python dictionary or a list of named arguments specific to the given query. To obtain the possible options for a given query, use the GetQueryParameters(name) function. If the query accepts additional arguments beyond its name, this function will return a python dictionary containing the needed variables and their default values. This can be modified and passed back to the Query method, or named arguments can be used instead.

Example:

```cpp
# visit -cli
OpenDatabase("/usr/gapps/visit/data/wave.visit")
AddPlot("Pseudocolor", "pressure")
DrawPlots()
Query("Volume")
Query("MinMax")
Query("MinMax", use_actual_data=1)

hohlraumArgs = GetQueryParameters("Hohlraum Flux")

hohlraumArgs["ray_center"]=(0.5,0.5,0)
hohlraumArgs["vars"]=('a1', "e1")
Query("Hohlraum Flux", hohlraumArgs)
```

2.4.168 QueryOverTime

Synopsis:

<table>
<thead>
<tr>
<th>Function</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>QueryOverTime(name)</td>
<td>integer</td>
</tr>
<tr>
<td>QueryOverTime(name, dict)</td>
<td>integer</td>
</tr>
<tr>
<td>QueryOverTime(name, namedarg1=val1, namedarg2=val2, ...)</td>
<td>integer</td>
</tr>
</tbody>
</table>

name [string] The name of the query to execute.

dict [dictionary] An optional dictionary containing additional query arguments. namedarg1, namedarg2,... An optional list of named arguments supplying additional query parameters.

return type [CLI_return_t] The QueryOverTime function returns 1 on success and 0 on failure.

Description:
The QueryOverTime function is used to execute any of VisIt’s predefined queries. The list of queries can be found in the VisIt User’s Manual in the Quantitative Analysis chapter. You can get also get a list of queries that can be executed over time using ‘QueriesOverTime’ function. Since queries can take a wide array of arguments, the Query function takes either a python dictionary or a list of named arguments specific to the given query. To obtain the possible options for a given query, use the GetQueryParameters(name) function. If the query accepts additional arguments beyond its name, this function will return a python dictionary containing the needed variables and their default values. This can be modified and passed back to the QueryOverTime method, or named arguments can be used instead.
Example:

```python
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/wave.visit")
AddPlot("Pseudocolor", "pressure")
DrawPlots()
for q in QueriesOverTime():
    QueryOverTime(q)
```

## 2.4.169 ReOpenDatabase

### Synopsis:

`ReOpenDatabase(databaseName) -> integer`

* `databaseName` [string] The name of the database to open.
* `return type` [CLI_return_t] The ReOpenDatabase function returns an integer value of 1 for success and 0 for failure.

### Description:

The ReOpenDatabase function reopens a database that has been opened previously with the OpenDatabase function. The ReOpenDatabase function is primarily used for regenerating plots whose database has been rewritten on disk. ReOpenDatabase allows VisIt to access new variables and new time states that have been added since the database was opened using the OpenDatabase function. Note that ReOpenDatabase is expensive since it causes all plots that use the specified database to be regenerated. If you want to ensure that a time-varying database has all of its time states as they are being created by a simulation, try the CheckForNewStates function instead. The `databaseName` argument is a string containing the full name of the database to be opened. The database name is of the form: host:/path/filename. The host part of the filename can be omitted if the database to be reopened resides on the local computer.

Example:

```python
#% visit -cli
OpenDatabase("edge:/usr/gapps/visit/data/wave*.silo database")
AddPlot("Pseudocolor", "pressure")
DrawPlots()
last = TimeSliderGetNStates()
for state in range(last):
    SetTimeSliderState(state)
    SaveWindow()
ReOpenDatabase("edge:/usr/gapps/visit/data/wave*.silo database")
for state in range(last, TimeSliderGetNStates()):
    SetTimeSliderState(state)
    SaveWindow()
```

## 2.4.170 ReadHostProfilesFromDirectory

### Synopsis:

`ReadHostProfilesFromDirectory(directory, clear) -> integer`

* `directory` [string] The name of the directory that contains the host profile XML files.
* `clear` [integer] An integer flag indicating whether the host profile list should cleared first.
**Return type** [CLI_return_t] The ReadHostProfilesFromDirectory function returns an integer value of 1 for success and 0 for failure.

**Description:**

The ReadHostProfilesFromDirectory provides a way to tell VisIt to load host profiles from the XML files in a specified directory. This is needed because the machine profile for host profiles contains client/server options that sometimes cannot be specified via the VisIt command line.

**Example:**

```c
ReadHostProfilesFromDirectory("/usr/gapps/visit/2.8.2/linux-x86_64/resources/hosts/_llnl", 1)
```

### 2.4.171 RecenterView

**Synopsis:**

`RecenterView() -> integer`

**Return type** [CLI_return_t] The RecenterView function returns 1 on success and 0 on failure.

**Description:**

After adding plots to a visualization window or applying operators to those plots, it is sometimes necessary to recenter the view. When the view is recentered, the orientation does not change but the view is shifted to make better use of the screen.

**Example:**

```c
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
DrawPlots()
OpenDatabase("/usr/gapps/visit/data/curv3d.silo")
AddPlot("Pseudocolor", "d")
DrawPlots()
RecenterView()
```

### 2.4.172 RedoView

**Synopsis:**

`RedoView() -> integer`

**Return type** [CLI_return_t] The RedoView function returns 1 on success and 0 on failure.

**Description:**

When the view changes in the visualization window, it puts the old view on a stack of views. VisIt provides the UndoView function that lets you undo view changes. The RedoView function re-applies any views that have been undone by the UndoView function.

**Example:**

```c
# visit -clí
OpenDatabase("/usr/gapps/visit/data/curv2d.silo")
AddPlot("Subset", "mat1")
```
2.4.173 RedrawWindow

Synopsis:

RedrawWindow() -> integer

return type  [CLI_return_t] The RedrawWindow function returns 1 on success and 0 on failure.

Description:

The RedrawWindow function allows a visualization window to redraw itself and then forces the window
to redraw. This function does the opposite of the DisableRedraw function and is used to recover from it.

Example:

```python
#$ visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Contour", "u")
AddPlot("Pseudocolor", "w")
DrawPlots()
DisableRedraw()
AddOperator("Slice")
# Set the slice operator attributes
# Redraw now that the operator attributes are set. This will
# prevent 1 redraw.
RedrawWindow()
```

2.4.174 RegisterCallback

Synopsis:

RegisterCallback(callbackname, callback) --> integer

callbackname  [string] A string object designating the callback that we’re installing. Allowable values are returned by the GetCallbackNames() function.

callback  [python function] A Python function, typically with one argument by which VisIt passes the object that caused the callback to be called.

return type  [CLI_return_t] RegisterCallback returns 1 on success.

Description:

The RegisterCallback function is used to associate a user-defined callback function with the updating of
a state object or execution of a particular rpc

Example:

```python
```
import visit

def print_sliceatts(atts):
    print("SLICEATTS=", atts)

visit.RegisterCallback("SliceAttributes", print_sliceatts)

### 2.4.175 RegisterMacro

**Synopsis:**

RegisterMacro(name, callable)

- **name** [string] The name of the macro.
- **callable** [python function] A Python function that will be associated with the macro name.

**Description:**

The `RegisterMacro` function lets you associate a Python function with a name so when VisIt’s gui calls down into Python to execute a macro, it ends up executing the registered Python function. Macros let users define complex new behaviors using Python functions yet still call them simply by clicking a button within VisIt’s gui. When a new macro function is registered, a message is sent to the gui that adds the known macros as buttons in the Macros window.

**Example:**

```python
def SetupMyPlots():
    OpenDatabase('noise.silo')
    AddPlot('Pseudocolor', 'hardyglobal')
    DrawPlots()

RegisterMacro('Setup My Plots', SetupMyPlots)
```

### 2.4.176 RemoveAllOperators

**Synopsis:**

RemoveAllOperators() -> integer
RemoveAllOperators(all) -> integer

- **all** [integer] An optional integer argument that tells the function to ignore the active plots and use all plots in the plot list if the value of the argument is non-zero.
- **return type** [CLI_return_t] All functions return an integer value of 1 for success and 0 for failure.

**Description:**

The `RemoveAllOperators` function removes all operators from the active plots in the active visualization window. If the all argument is provided and contains a non-zero value, all plots in the active visualization window are affected. If the value is zero or if the argument is not provided, only the active plots are affected.

**Example:**
```python
## visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
AddOperator("Threshold")
AddOperator("Slice")
AddOperator("SphereSlice")
DrawPlots()
RemoveLastOperator() # Remove SphereSlice
RemoveOperator(0) # Remove Threshold
RemoveAllOperators() # Remove the rest of the operators
```

## 2.4.177 RemoveLastOperator

**Synopsis:**

```python
RemoveLastOperator() -> integer
RemoveLastOperator(all) -> integer
```

all [integer] An optional integer argument that tells the function to ignore the active plots and use all plots in the plot list if the value of the argument is non-zero.

**return type** [CLI_return_t] All functions return an integer value of 1 for success and 0 for failure.

**Description:**

The RemoveLastOperator function removes the operator that was last applied to the active plots. If the all argument is provided and contains a non-zero value, all plots in the active visualization window are affected. If the value is zero or if the argument is not provided, only the active plots are affected.

**Example:**

```python
## visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
AddOperator("Threshold")
AddOperator("Slice")
AddOperator("SphereSlice")
DrawPlots()
RemoveLastOperator() # Remove SphereSlice
RemoveOperator(0) # Remove Threshold
RemoveAllOperators() # Remove the rest of the operators
```

## 2.4.178 RemoveMachineProfile

**Synopsis:**

```python
RemoveMachineProfile(hostname) -> integer
```

hostname : string

**Description:**

Removes machine profile with hostname from HostProfileList

**Example:**

```python
## visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
AddOperator("Threshold")
AddOperator("Slice")
AddOperator("SphereSlice")
DrawPlots()
RemoveLastOperator() # Remove SphereSlice
RemoveOperator(0) # Remove Threshold
RemoveAllOperators() # Remove the rest of the operators
```
2.4.179 RemoveOperator

Synopsis:

RemoveOperator(index) -> integer
RemoveOperator(index, all) -> integer

**all** [integer] An optional integer argument that tells the function to ignore the active plots and use all plots in the plot list if the value of the argument is non-zero.

**index** [integer] The zero-based integer index into a plot’s operator list that specifies which operator is to be deleted.

**return type** [CLI_return_t] All functions return an integer value of 1 for success and 0 for failure.

**Description:**

The RemoveOperator functions allow operators to be removed from plots. If the all argument is provided and contains a non-zero value, all plots in the active visualization window are affected. If the value is zero or if the argument is not provided, only the active plots are affected.

**Example:**

```bash
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
AddOperator("Threshold")
AddOperator("Slice")
AddOperator("SphereSlice")
DrawPlots()
RemoveLastOperator() # Remove SphereSlice
RemoveOperator(0) # Remove Threshold
RemoveAllOperators() # Remove the rest of the operators
```

2.4.180 RemovePicks

Synopsis:

RemovePicks()

**Description:**

The RemovePicks function removes a list of pick points from the active visualization window. Pick points are the letters that are added to the visualization window where the mouse is clicked when the visualization window is in pick mode.

**Example:**

```bash
#% visit -cli
# Put the visualization window into pick mode using the popup menu and add some pick points (let's say A -> G).
# Clear the pick points.
RemovePicks('A, B, D')
```

2.4.181 RenamePickLabel

Synopsis:
VisIt User Manual Documentation, Release 3.1

```
RenamePickLabel(oldLabel, newLabel) -> integer
```

**oldLabel** [string] A string that is the old pick label to replace. (e.g. ‘A’, ‘B’).

**newLabel** [string] A string that is the new label to display in place of the old label.

**return type** [CLI_return_t] The RenamePickLabel function returns 1 on success and 0 on failure.

**Description:**

The RenamePickLabel function can be used to replace an automatically generated pick label such as ‘A’ with a user-defined string.

**Example:**

```
RenamePickLabel('A', 'Point of interest')
```

### 2.4.182 ReplaceDatabase

**Synopsis:**

```
ReplaceDatabase(databaseName) -> integer
ReplaceDatabase(databaseName, timeState) -> integer
```

**databaseName** [string] The name of the new database.

**timeState** [integer] A zero-based integer containing the time state that should be made active once the database has been replaced.

**return type** [CLI_return_t] The ReplaceDatabase function returns an integer value of 1 for success and 0 for failure.

**Description:**

The ReplaceDatabase function replaces the database in the current plots with a new database. This is one way of switching timesteps if no “.visit” file was ever created. If two databases have the same variable name then replace is usually a success. In the case where the new database does not have the desired variable, the plot with the variable not contained in the new database does not get regenerated with the new database.

**Example:**

```
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
DrawPlots()
ReplaceDatabase("/usr/gapps/visit/data/curv3d.silo")
SaveWindow()
# Replace with a time-varying database and change the time
# state to 17.
ReplaceDatabase("/usr/gapps/visit/data/wave.visit", 17)
```

### 2.4.183 ResetLineoutColor

**Synopsis:**

```
ResetLineoutColor() -> integer
```

**return type** [CLI_return_t] ResetLineoutColor returns 1 on success and 0 on failure.
Description:
Lineouts on VisIt cause reference lines to be drawn over the plot where the lineout was being extracted. Each reference line uses a different color in a discrete color table. Once the colors in the discrete color table are used up, the reference lines start using the color from the start of the discrete color table and so on. ResetLineoutColor forces reference lines to start using the color at the start of the discrete color table again thus resetting the lineout color.

2.4.184 ResetOperatorOptions

Synopsis:

<table>
<thead>
<tr>
<th>Function</th>
<th>Return Type</th>
</tr>
</thead>
<tbody>
<tr>
<td>ResetOperatorOptions(operatorType)</td>
<td>integer</td>
</tr>
<tr>
<td>ResetOperatorOptions(operatorType, all)</td>
<td>integer</td>
</tr>
</tbody>
</table>

**operatorType** [string] The name of a valid operator type.

**all** [integer] An optional integer argument that tells the function to reset the operator options for all plots regardless of whether or not they are active.

**return type** [CLI_return_t] The ResetOperatorOptions function returns an integer value of 1 for success and 0 for failure.

Description:
The ResetOperatorOptions function resets the operator attributes of the specified operator type for the active plots back to the default values. The operatorType argument is a string containing the name of the type of operator whose attributes are to be reset. The all argument is an optional flag that tells the function to reset the operator attributes for the indicated operator in all plots regardless of whether the plots are active. When non-zero values are passed for the all argument, all plots are reset. When the all argument is zero or not provided, only the operators on active plots are modified.

Example:
```
# visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
DrawPlots()
AddOperator("Slice")
a = SliceAttributes()
a.normal, a.upAxis = (0,0,1), (0,1,0)
SetOperatorOptions(a)
ResetOperatorOptions("Slice")
```

2.4.185 ResetPickLetter

Synopsis:

**ResetPickLetter()** -> integer

**return type** [CLI_return_t] ResetPickLetter returns 1 on success and 0 on failure.

Description:
The ResetPickLetter function resets the pick marker back to “A” so that the next pick will use “A” as the pick letter and then “B” and so on.
2.4.186 ResetPlotOptions

Synopsis:

ResetPlotOptions(plotType) -> integer

**plotType** [string] The name of the plot type.

**return type** [CLI_return_t] The ResetPlotOptions function returns an integer value of 1 for success and 0 for failure.

Description:

The ResetPlotOptions function resets the plot attributes of the specified plot type for the active plots back to the default values. The plotType argument is a string containing the name of the type of plot whose attributes are to be reset.

Example:

```python
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
DrawPlots()
p = PseudocolorAttributes()
p.colorTableName = "calewhite"
p.minFlag, p.maxFlag = 1, 1
p.min, p.max = -5.0, 8.0
SetPlotOptions(p)
ResetPlotOptions("Pseudocolor")
```

2.4.187 ResetView

Synopsis:

ResetView() -> integer

**return type** [CLI_return_t] The ResetView function returns 1 on success and 0 on failure.

Description:

The ResetView function resets the camera to the initial view.

Example:

```python
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/curv3d.silo")
AddPlot("Mesh", "curvmesh3d")
v = ViewAttributes()
v.camera = (-0.45396, 0.401908, 0.79523)
v.focus = (0, 2.5, 15)
v.viewUp = (0.109387, 0.910879, -0.397913)
v.viewAngle = 30
v.setScale = 1
v.parallelScale = 16.0078
v.nearPlane = -32.0156
v.farPlane = 32.0156
v.perspective = 1
SetView3D(v)  # Set the 3D view
DrawPlots()
ResetView()
```
2.4.188 ResizeWindow

Synopsis:

ResizeWindow(win, w, h) -> integer

win [integer] The integer id of the window to be moved [1..16].
w [integer] The new integer width for the window.
h [integer] The new integer height for the window.

return type [CLI_return_t] ResizeWindow returns 1 on success and 0 on failure.

Description:

ResizeWindow resizes a visualization window.

Example:

```bash
# visit -cli
ResizeWindow(1, 300, 600)
```

2.4.189 RestoreSession

Synopsis:

RestoreSession(filename, visitDir) -> integer

filename [string] The name of the session file to restore.

visitDir [integer] An integer flag that indicates whether the filename to be restored is located in the user’s VisIt directory. If the flag is set to 1 then the session file is assumed to be located in the user’s VisIt directory otherwise the filename must contain an absolute path.

return type [CLI_return_t] RestoreSession returns 1 on success and 0 on failure.

Description:

The RestoreSession function is important for setting up complex visualizations because you can design a VisIt session file, which is an XML file that describes exactly how plots are set up, using the VisIt GUI and then use that same session file in the CLI to generate movies in batch. The RestoreSession function takes 2 arguments. The first argument specifies the filename that contains the VisIt session to be restored. The second argument determines whether the session file is assumed to be in the user’s VisIt directory. If the visitDir argument is set to 0 then the filename argument must contain the absolute path to the session file.

Example:

```bash
# visit -cli
# Restore my session file for a time-varying database from my .visit directory.
RestoreSessionFile("visit.session", 1)
for state in range(TimeSliderGetNStates()):
    SetTimeSliderState(state)
    SaveWindow()
```
2.4.190 RestoreSessionWithDifferentSources

Synopsis:

```
RestoreSessionWithDifferentSources(filename, visitDir, mapping) -> integer
```

- **filename** [string] The name of the session file to restore.
- **visitDir** [integer] An integer flag that indicates whether the filename to be restored is located in the user’s VisIt directory. If the flag is set to 1 then the session file is assumed to be located in the user’s VisIt directory otherwise the filename must contain an absolute path.
- **mapping** [tuple] A tuple of strings representing the mapping from sources as specified in the original session file to new sources. Sources in the original session file are numbered starting from 0. So, this tuple of strings simply contains the new names for each of the sources, in order.

Return type [CLI_return_t] RestoreSession returns 1 on success and 0 on failure.

Description:
The RestoreSession function is important for setting up complex visualizations because you can design a VisIt session file, which is an XML file that describes exactly how plots are set up, using the VisIt GUI and then use that same session file in the CLI to generate movies in batch. The RestoreSession function takes 2 arguments. The first argument specifies the filename that contains the VisIt session to be restored. The second argument determines whether the session file is assumed to be in the user’s VisIt directory. If the visitDir argument is set to 0 then the filename argument must contain the absolute path to the session file.

Example:

```bash
#% visit -cli
# Restore my session file for a time-varying database from
# my .visit directory.
RestoreSessionFile("visit.session", 1)
for state in range(TimeSliderGetNStates()):
    SetTimeSliderState(state)
    SaveWindow()
```

2.4.191 SaveAttribute

Synopsis:

```
SaveAttribute(filename, object)
```

- **filename** [string] The name of the XML file to load the attribute from or save the attribute to.
- **object** The object to load or save.

Description:
The LoadAttribute and SaveAttribute methods save a single attribute, such as a current plot or operator python object, to a standalone XML file. Note that LoadAttribute requires that the target attribute already be created by other means; it fills, but does not create, the attribute.

Example:

```bash
#% visit -cli
a = MeshPlotAttributes()
```
2.4.192 SaveNamedSelection

Synopsis:

SaveNamedSelection(name) -> integer

name [string] The name of a named selection.

return type [CLI_return_t] The SaveNamedSelection function returns 1 for success and 0 for failure.

Description:

Named Selections allow you to select a group of elements (or particles). One typically creates a named selection from a group of elements and then later applies the named selection to another plot (thus reducing the set of elements displayed to the ones from when the named selection was created). Named selections only last for the current session. If you create a named selection that you want to use over and over, you can save it to a file with the SaveNamedSelection function.

Example:

```python
#% visit -cli
db = "/usr/gapps/visit/data/wave*.silo database"
OpenDatabase(db)
AddPlot("Pseudocolor", "pressure")
AddOperator("Clip")
c = ClipAttributes()
c.plane1Origin = (0, 0.6, 0)
c.plane1Normal = (0, -1, 0)
SetOperatorOption(c)
DrawPlots()
CreateNamedSelection("els_above_at_time_0")
SaveNamedSelection("els_above_at_time_0")
```

2.4.193 SaveSession

Synopsis:

SaveSession(filename) -> integer

filename [string] The filename argument is the filename that is used to save the session file. The filename is relative to the current working directory.

return type [CLI_return_t] The SaveSession function returns 1 on success and 0 on failure.

Description:

The SaveSession function tells VisIt to save an XML session file that describes everything about the current visualization. Session files are very useful for creating movies and also as shortcuts for setting up complex visualizations.

Example:
2.4.194 SaveWindow

Synopsis:

SaveWindow() -> string

return type [string] The SaveWindow function returns a string containing the name of the file that was saved.

Description:

The SaveWindow function saves the contents of the active visualization window. The format of the saved window is dictated by the SaveWindowAttributes which can be set using the SetSaveWindowAttributes function. The contents of the active visualization window can be saved as TIFF, JPEG, RGB, PPM, PNG images or they can be saved as curve, Alias Wavefront Obj, or VTK geometry files.

Example:

```python
#$ visit -cli
OpenDatabase("/usr/gapps/visit/data/cuv3d.silo")
AddPlot("Pseudocolor", "d")
DrawPlots()
# Set the save window attributes.
s = SaveWindowAttributes()
s.fileName = "test"
s.format = s.JPG
s.progressive = 1
name = SaveWindowAttributes(s)
print("name = "; name)
```

2.4.195 SendSimulationCommand

Synopsis:
SendSimulationCommand(host, simulation, command)
SendSimulationCommand(host, simulation, command, argument)

host  [string] The name of the computer where the simulation is running.
simulation  [string] The name of the simulation being processed at the specified host.
command  [string] A string that is the command to send to the simulation.
argument  An argument to the command.

Description:
The SendSimulationCommand method tells the viewer to send a command to a simulation that is running on the specified host. The host argument is a string that contains the name of the computer where the simulation is running. The simulation argument is a string that contains the name of the simulation to send the command to.

2.4.196 SetActiveContinuousColorTable

Synopsis:
SetActiveContinuousColorTable(name) -> integer

name  [string] The name of the color table to use for the active color table. The name must be present in the tuple returned by the ColorTableNames function.

return type  [CLI_return_t] Both functions return 1 on success and 0 on failure.

Description:
VisIt supports two flavors of color tables: continuous and discrete. Both types of color tables have the same underlying representation but each type of color table is used a slightly different way. Continuous color tables are made of a small number of color control points and the gaps in the color table between two color control points are filled by interpolating the colors of the color control points. Discrete color tables do not use any kind of interpolation and like continuous color tables, they are made up of control points. The color control points in a discrete color table repeat infinitely such that if we have 4 color control points: A, B, C, D then the pattern of repetition is: ABCDABCDABCD... Discrete color tables are mainly used for plots that have a discrete set of items to display (e.g. Subset plot). Continuous color tables are used in plots that display a continuous range of values (e.g. Pseudocolor).

Example:
```bash
#!/usr/gapps/visit/data/noise.silo
AddPlot("Contour", "hgslice")
DrawPlots()
SetActiveDiscreteColorTable("levels")
```

2.4.197 SetActiveDiscreteColorTable

Synopsis:
SetActiveDiscreteColorTable(name) -> integer

name  [string] The name of the color table to use for the active color table. The name must be present in the tuple returned by the ColorTableNames function.

2.4. Functions 543
return type  [CLI_return_t] Both functions return 1 on success and 0 on failure.

Description:
VisIt supports two flavors of color tables: continuous and discrete. Both types of color tables have the same underlying representation but each type of color table is used a slightly different way. Continuous color tables are made of a small number of color control points and the gaps in the color table between two color control points are filled by interpolating the colors of the color control points. Discrete color tables do not use any kind of interpolation and like continuous color tables, they are made up of control points. The color control points in a discrete color table repeat infinitely such that if we have 4 color control points: A, B, C, D then the pattern of repetition is: ABCDABCDABCD... Discrete color tables are mainly used for plots that have a discrete set of items to display (e.g. Subset plot). Continuous color tables are used in plots that display a continuous range of values (e.g. Pseudocolor).

Example:

```bash
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/noise.silo")
AddPlot("Contour", "hgslice")
DrawPlots()
SetActiveDiscreteColorTable("levels")
```

2.4.198 SetActivePlots

Synopsis:

```
SetActivePlots(plots) -> integer
```

plots  [tuple of integers] A tuple of integer plot indices starting at zero. A single integer is also accepted

return type  [CLI_return_t] The SetActivePlots function returns an integer value of 1 for success and 0 for failure.

Description:
Any time VisIt sets the attributes for a plot, it only sets the attributes for plots which are active. The SetActivePlots function must be called to set the active plots. The function takes one argument which is a tuple of integer plot indices that start at zero. If only one plot is being selected, the plots argument can be an integer instead of a tuple.

Example:

```bash
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Subset", "mat1")
AddPlot("Mesh", "mesh1")
AddPlot("Contour", "u")
DrawPlots()
SetActivePlots((0,1,2))  # Make all plots active
SetActivePlots(0)  # Make only the Subset plot active
```

2.4.199 SetActiveTimeSlider

Synopsis:

```
SetActiveTimeSlider(tsName) -> integer
```

tsName  [string] The name of the time slider that should be made active.
return type  [CLI_return_t] SetActiveTimeSlider returns 1 on success and 0 on failure.

Description:
Sets the active time slider, which is the time slider that is used to change time states.

Example:

```python
#% visit -cli
path = "/usr/gapps/visit/data/

dbs = (path + "dbA00.pdb", path + "dbB00.pdb", path + "dbC00.pdb")
for db in dbs:
    OpenDatabase(db)
    AddPlot("FilledBoundary", "material(mesh)"
    DrawPlots()
CreateDatabaseCorrelation("common", dbs, 1)
tsNames = GetWindowInformation().timeSliders
for ts in tsNames:
    SetActiveTimeSlider(ts)
for state in list(range(TimeSliderGetNStates())) + [0]:
    SetTimeSliderState(state)
```

2.4.200 SetActiveWindow

Synopsis:

SetActiveWindow(windowIndex) -> integer
SetActiveWindow(windowIndex, raiseWindow) -> integer

windowIndex  [integer] An integer window index starting at 1.
raiseWindow  [integer] This is an optional integer argument that raises and activates the window if set to 1. If omitted, the default behavior is to raise and activate the window.

return type  [CLI_return_t] The SetActiveWindow function returns an integer value of 1 for success and 0 for failure.

Description:
Most of the functions in the VisIt Python Interface operate on the contents of the active window. If there is more than one window, it is very important to be able to set the active window. To set the active window, use the SetActiveWindow function. The SetActiveWindow function takes a single integer argument which is the index of the new active window. The new window index must be an integer greater than zero and less than or equal to the number of open windows.

Example:

```python
#% visit -cli
SetWindowLayout(2)
SetActiveWindow(2)
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Mesh", "mesh1")
DrawPlots()
```

2.4.201 SetAnimationTimeout

Synopsis:
SetAnimationTimeout(milliseconds) -> integer

**return type** [CLI_return_t] The SetAnimationTimeout function returns 1 for success and 0 for failure.

**Description:**

The SetAnimationTimeout function sets the animation timeout which is a value that governs how fast animations play. The timeout is specified in milliseconds and has a default value of 1 millisecond. Larger timeout values decrease the speed at which animations play.

**Example:**

```
#%visit -cli
# Play a new frame every 5 seconds.
SetAnimationTimeout(5000)
OpenDatabase("/usr/gapps/visit/data/wave.visit")
AddPlot("Pseudocolor", "pressure")
DrawPlots()
# Click the play button in the toolbar
```

### 2.4.202 SetAnnotationAttributes

**Synopsis:**

SetAnnotationAttributes(atts) -> integer

**atts** [AnnotationAttributes object] An AnnotationAttributes object containing the annotation settings.

**return type** [CLI_return_t] Both functions return 1 on success and 0 on failure.

**Description:**

The annotation settings control what bits of text are drawn in the visualization window. Among the annotations are the plot legends, database information, user information, plot axes, triad, and the background style and colors. Setting the annotation attributes is important for producing quality visualizations. The annotation settings are stored in AnnotationAttributes objects. To set the annotation attributes, first create an AnnotationAttributes object using the AnnotationAttributes function and then pass the object to the SetAnnotationAttributes function. To set the default annotation attributes, also pass the object to the SetDefaultAnnotationAttributes function.

**Example:**

```
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/wave.visit")
AddPlot("Pseudocolor", "pressure")
DrawPlots()
a = AnnotationAttributes()
a.gradientBackgroundStyle = a.GRADIENTSTYLE_RADIAL
a.gradientColor1 = (0,255,255)
a.gradientColor2 = (0,0,0)
a.backgroundMode = a.BACKGROUNDMODE_GRADIENT
SetAnnotationAttributes(a)
```

### 2.4.203 SetBackendType

**Synopsis:**
SetBackendType(name) -> integer

name [string] VTK, VTKM.

return type [CLI_return_t] Both functions return 1 on success and 0 on failure.

Description:
The compute back end determines the compute library that is used for processing plots in VisIt. The default is VTK, which supports all VisIt operations. VTKm can be used too but it only supports a fraction of VisIt's functionality. Filters that support VTKm will use those libraries when their compute back end is selected using this function.

Example:
```c
#!/usr/bin/env python
OpenDatabase("/home/visit/data/noise.silo")
SetBackendType("VTKm")
AddPlot("Contour", "radial")
DrawPlots()
```

2.4.204 SetCenterOfRotation

Synopsis:

SetCenterOfRotation(x,y,z) -> integer

x [double] A double that is the x component of the center of rotation.
y [double] A double that is the y component of the center of rotation.
z [double] A double that is the z component of the center of rotation.

return type [CLI_return_t] The SetCenterOfRotation function returns 1 on success and 0 on failure.

Description:
The SetCenterOfRotation function sets the center of rotation for plots in a 3D visualization window. The center of rotation, is the point about which plots are rotated when you interactively spin the plots using the mouse. It is useful to set the center of rotation if you've zoomed in on any 3D plots so in the event that you rotate the plots, the point of interest remains fixed on the screen.

Example:
```c
#!/usr/bin/env python
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
AddPlot("Mesh", "mesh1")
DrawPlots()
v = GetView3D()
v.viewNormal = (-0.409139, 0.631025, 0.6591)
v.viewUp = (0.320232, 0.775678, -0.543851)
v.imageZoom = 4.8006
SetCenterOfRotation(-4.755280, 6.545080, 5.877850)
# Rotate the plots interactively.
```
2.4.205 SetColorTexturingEnabled

Synopsis:

SetColorTexturingEnabled(enabled) -> integer

enabled [integer] A integer value. Non-zero values enable color texturing and zero disables it.

return type [CLI_return_t] The SetColorTexturingEnabled function returns 1 on success and 0 on failure.

Description:

Node-centered variables are drawn on plots such as the Pseudocolor plot such that the nodal value looks interpolated throughout the zone. This can be done by interpolating colors, which can produce some colors that do not appear in a color table. Alternatively, the nodal values can be mapped to a texture coordinate in a 1D texture and those values can be interpolated, with colors being selected after interpolating the texture coordinate. This method always uses colors that are defined in the color table.

Example:

SetColorTexturingEnabled(1)

2.4.206 SetCreateMeshQualityExpressions

Synopsis:

SetCreateMeshQualityExpressions(val) -> integer

val [integer] Either a zero (false) or non-zero (true) integer value to indicate if Mesh Quality expressions should be automatically created when a database is opened.

return type [CLI_return_t] The SetCreateMeshQualityExpressions function returns 1 on success and 0 on failure.

Description:

The SetCreateMeshQualityExpressions function sets a boolean in the global attributes indicating whether or not Mesh Quality expressions should be automatically created. The default behavior is for the expressions to be created, which may slow down VisIt’s performance if there is an extraordinary large number of meshes. Turning this feature off tells VisIt to skip automatic creation of the Mesh Quality expressions.

Example:

#% visit -cli
SetCreateMeshQualityExpressions(1) # turn this feature on
SetCreateMeshQualityExpressions(0) # turn this feature off

2.4.207 SetCreateTimeDerivativeExpressions

Synopsis:

SetCreateTimeDerivativeExpressions(val) -> integer

val [integer] Either a zero (false) or non-zero (true) integer value to indicate if Time Derivative expressions should be automatically created when a database is opened.

return type [CLI_return_t] The SetCreateTimeDerivativeExpressions function returns 1 on success and 0 on failure.

Description:
The `SetCreateTimeDerivativeExpressions` function sets a boolean in the global attributes indicating whether or not Time Derivative expressions should be automatically created. The default behavior is for the expressions to be created, which may slow down VisIt’s performance if there is an extraordinary large number of variables. Turning this feature off tells VisIt to skip automatic creation of the Time Derivative expressions.

**Example:**

```
#% visit -cli
SetCreateTimeDerivativeExpressions(1)  # turn this feature on
SetCreateTimeDerivativeExpressions(0)  # turn this feature off
```

### 2.4.208 SetCreateVectorMagnitudeExpressions

**Synopsis:**

```
SetCreateVectorMagnitudeExpressions(val) -> integer
```

**val** [integer] Either a zero (false) or non-zero (true) integer value to indicate if Vector magnitude expressions should be automatically created when a database is opened.

**return type** [CLI_return_t] The `SetCreateVectorMagnitudeExpressions` function returns 1 on success and 0 on failure.

**Description:**

The `SetCreateVectorMagnitudeExpressions` function sets a boolean in the global attributes indicating whether or not vector magnitude expressions should be automatically created. The default behavior is for the expressions to be created, which may slow down VisIt’s performance if there is an extraordinary large number of vector variables. Turning this feature off tells VisIt to skip automatic creation of the vector magnitude expressions.

**Example:**

```
#% visit -cli
SetCreateVectorMagnitudeExpressions(1)  # turn this feature on
SetCreateVectorMagnitudeExpressions(0)  # turn this feature off
```

### 2.4.209 SetDatabaseCorrelationOptions

**Synopsis:**

```
SetDatabaseCorrelationOptions(method, whenToCreate) -> integer
```

**method** [integer] An integer that tells VisIt what default method to use when automatically creating a database correlation. The value must be in the range [0,3].

<table>
<thead>
<tr>
<th>method</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>IndexForIndexCorrelation</td>
</tr>
<tr>
<td>1</td>
<td>StretchedIndexCorrelation</td>
</tr>
<tr>
<td>2</td>
<td>TimeCorrelation</td>
</tr>
<tr>
<td>3</td>
<td>CycleCorrelation</td>
</tr>
</tbody>
</table>

**whenToCreate** [integer] An integer that tells VisIt when to automatically create database correlations.

2.4. Functions
<table>
<thead>
<tr>
<th>whenToCreate</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>Always create database correlation</td>
</tr>
<tr>
<td>1</td>
<td>Never create database correlation</td>
</tr>
<tr>
<td>2</td>
<td>Create database correlation only if the new time-varying database has the same length as another time-varying database already being used in a plot.</td>
</tr>
</tbody>
</table>

**Description:**

VisIt provides functions to explicitly create and alter database correlations but there are also a number of occasions where VisIt can automatically create a database correlation. The SetDatabaseCorrelationOptions function allows you to tell VisIt the default correlation method to use when automatically creating a new database correlation and it also allows you to tell VisIt when database correlations can be automatically created. The same length as another time-varying database already being used in a plot.

**Example:**

```python
#$ visit -cli
OpenDatabase("/usr/gapps/visit/data/dbA00.pdb")
AddPlot("FilledBoundary", "material(mesh)")
DrawPlots()
# Always create a stretched index correlation.
SetDatabaseCorrelationOptions(1, 0)
OpenDatabase("/usr/gapps/visit/data/dbB00.pdb")
AddPlot("FilledBoundary", "material(mesh)")
# The AddPlot caused a database correlation to be created.
DrawPlots()
wi = GetWindowInformation()
print("Active time slider: ", wi.timeSliders[wi.activeTimeSlider])
# This will set time for both databases since the database correlation is the active time slider.
SetTimeSliderState(5)
```

### 2.4.210 SetDebugLevel

**Synopsis:**

SetDebugLevel(level)

**level** [string] A string ‘1’, ‘2’, ‘3’, ‘4’, ‘5’ with an optional ‘b’ suffix to indicate whether the output should be buffered. A value of ‘1’ is a low debug level, which should be used to produce little output while a value of 5 should produce a lot of debug output.

**Description:**

The GetDebugLevel and SetDebugLevel functions are used when debugging VisIt Python scripts. The SetDebugLevel function sets the debug level for VisIt’s viewer thus it must be called before a Launch method. The debug level determines how much detail is written to VisIt’s execution logs when it executes.

**Example:**

```bash
#$ visit -cli -debug 2
print("VisIt's debug level is: ", % GetDebugLevel())
```
2.4.211 SetDefaultAnnotationAttributes

Synopsis:

```python
SetDefaultAnnotationAttributes(atts) -> integer
```

**atts** [AnnotationAttributes object] An AnnotationAttributes object containing the annotation settings.

**return type** [CLI_return_t] Both functions return 1 on success and 0 on failure.

Description:

The annotation settings control what bits of text are drawn in the visualization window. Among the annotations are the plot legends, database information, user information, plot axes, triad, and the background style and colors. Setting the annotation attributes is important for producing quality visualizations. The annotation settings are stored in AnnotationAttributes objects. To set the annotation attributes, first create an AnnotationAttributes object using the AnnotationAttributes function and then pass the object to the SetAnnotationAttributes function. To set the default annotation attributes, also pass the object to the SetDefaultAnnotationAttributes function.

Example:

```bash
# visit -cli
OpenDatabase("/usr/gapps/visit/data/wave.visit")
AddPlot("Pseudocolor", "pressure")
DrawPlots()
a = AnnotationAttributes()
a.gradientBackgroundStyle = a.GRADIENTSTYLE_RADIAL
a.gradientColor1 = (0,255,255)
a.gradientColor2 = (0,0,0)
a.backgroundMode = a.BACKGROUNDMODE_GRADIENT
SetAnnotationAttributes(a)
```

2.4.212 SetDefaultFileOpenOptions

Synopsis:

```python
SetDefaultFileOpenOptions(pluginName, options) -> integer
```

**pluginName** [string] The name of a plugin.

**options** [dictionary] A dictionary containing the new default options for that plugin.

**return type** [CLI_return_t] The SetDefaultFileOpenOptions function returns 1 on success and 0 on failure.

Description:

SetDefaultFileOpenOptions sets the current options used to open new files when a specific plugin is triggered.

Example:

```bash
# visit -cli
OpenMDServer()
opts = GetDefaultFileOpenOptions("VASP")
opts["Allow multiple timesteps"] = 1
SetDefaultFileOpenOptions("VASP", opts)
OpenDatabase("CHGCAR")
```
2.4.213 SetDefaultInteractorAttributes

Synopsis:

SetDefaultInteractorAttributes(atts) -> integer

atts [InteractorAttributes object] An InteractorAttributes object that contains the new interactor attributes that you want to use.

return type [CLI_return_t] SetInteractorAttributes returns 1 on success and 0 on failure.

Description:

The SetInteractorAttributes function is used to set certain interactor properties. Interactors, can be thought of as how mouse clicks and movements are translated into actions in the vis window. To set the interactor attributes, first get the interactor attributes using the GetInteractorAttributes function. Once you’ve set the object’s properties, call the SetInteractorAttributes function to make VisIt use the new interactor attributes.

The SetDefaultInteractorAttributes function sets the default interactor attributes, which are used for new visualization windows. The default interactor attributes can also be saved to the VisIt configuration file to ensure that future VisIt sessions have the right default interactor attributes.

Example:

```python
# visit -cli
ia = GetInteractorAttributes()
print(ia)
ia.showGuidelines = 0
SetInteractorAttributes(ia)
```

2.4.214 SetDefaultMaterialAttributes

Synopsis:

SetDefaultMaterialAttributes(atts) -> integer

atts [MaterialAttributes object] A MaterialAttributes object containing the new settings.

return type [CLI_return_t] Both functions return 1 on success and 0 on failure.

Description:

The SetMaterialAttributes function takes a MaterialAttributes object and makes VisIt use the material settings that it contains. You use the SetMaterialAttributes function when you want to change how VisIt performs material interface reconstruction. The SetDefaultMaterialAttributes function sets the default material attributes, which are saved to the config file and are also used by new visualization windows.

Example:

```python
# visit -cli
OpenDatabase("/usr/gapps/visit/data/allinone00.pdb")
AddPlot("Pseudocolor", "mesh/mixvar")
p = PseudocolorAttributes()
p.min, p.minFlag = 4.0, 1
p.max, p.maxFlag = 13.0, 1
SetPlotOptions(p)
DrawPlots()
# Tell VisIt to always do material interface reconstruction.
m = GetMaterialAttributes()
```

(continues on next page)
2.4.215 SetDefaultMeshManagementAttributes

Synopsis:

SetDefaultMeshManagementAttributes() -> MeshmanagementAttributes object

return type  [MeshmanagementAttributes object] Returns a MeshmanagementAttributes object.

Description:

The GetMeshmanagementAttributes function returns a MeshmanagementAttributes object that contains VisIt's current mesh discretization settings. You can set properties on the MeshManagementAttributes object and then pass it to SetMeshManagementAttributes to make VisIt use the new material attributes that you've specified:

Example:

```c
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/csg.silo")
AddPlot("Mesh", "csgmesh")
DrawPlots()
# Tell VisIt to always do material interface reconstruction.
mma = GetMeshManagementAttributes()
mma.discretizationTolerance = (0.01, 0.025)
SetMeshManagementAttributes(mma)
ClearWindow()
# Redraw the plot forcing VisIt to use the mixed variable information.
DrawPlots()
```

2.4.216 SetDefaultOperatorOptions

Synopsis:

SetDefaultOperatorOptions(atts) -> integer

atts  [operator attributes object] Any type of operator attributes object.

return type  [CLI_return_t] All functions return an integer value of 1 for success and 0 for failure.

Description:

Each operator in VisIt has a group of attributes that controls the operator. To set the attributes for an operator, first create an operator attributes object. This is done by calling a function which is the name of the operator plus the word “Attributes”. For example, a Slice operator’s operator attributes object is created and returned by the SliceAttributes function. Assign the new operator attributes object into a variable and set its fields. After setting the desired fields in the operator attributes object, pass the object to the SetOperatorOptions function. The SetOperatorOptions function determines the type of operator to which the operator attributes object applies and sets the attributes for that operator type. To set the default plot attributes, use the SetDefaultOperatorOptions function. Setting the default attributes ensures
that all future instances of a certain operator are initialized with the new default values. Note that there is no SetOperatorOptions(atts, all) variant of this call. To set operator options for all plots that have a an instance of the associated operator, you must first make all plots active with SetActivePlots() and then use the SetOperatorOptions(atts) variant.

Example:

```python
# visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
AddPlot("Mesh", "mesh1")
AddOperator("Slice", 1) # Add the operator to both plots
a = SliceAttributes()
a.normal, a.upAxis = (0,0,1), (0,1,0)
# Only set the attributes for the active plot.
SetOperatorOptions(a)
DrawPlots()
```

### 2.4.217 SetDefaultPickAttributes

**Synopsis:**

```
SetDefaultPickAttributes(atts) -> integer
```

**atts** [PickAttributes object] A PickAttributes object containing the new pick settings.

**return type** [CLI_return_t] All functions return 1 on success and 0 on failure.

**Description:**

The SetPickAttributes function changes the pick attributes that are used when VisIt picks on plots. The pick attributes allow you to format your pick output in various ways and also allows you to select auxiliary pick variables.

**Example:**

```python
OpenDatabase("/usr/gapps/visit/data/noise.silo")
AddPlot("Pseudocolor", "hgslice")
DrawPlots()
ZonePick(coord=(-5,5,0))
p = GetPickAttributes()
p.showTimeStep = 0
p.showMeshName = 0
p.showZoneId = 0
SetPickAttributes(p)
ZonePick(coord=(0,5,0))
```

### 2.4.218 SetDefaultPlotOptions

**Synopsis:**

```
SetDefaultPlotOptions(atts) -> integer
```

**atts** [plot attributes object] Any type of plot attributes object.

**return type** [CLI_return_t] All functions return an integer value of 1 for success and 0 for failure.
Description:

Each plot in VisIt has a group of attributes that controls the appearance of the plot. To set the attributes for a plot, first create a plot attributes object. This is done by calling a function which is the name of the plot plus the word “Attributes”. For example, a Pseudocolor plot’s plotattributes object is created and returned by the PseudocolorAttributes function. Assign the new plot attributes object into a variable and set its fields. After setting the desired fields in the plot attributes object, pass the object to the SetPlotOptions function. The SetPlotOptions function determines the type of plot to which the plot attributes object applies and sets the attributes for that plot type. To set the default plot attributes, use the SetDefaultPlotOptions function. Setting the default attributes ensures that all future instances of a certain plot are initialized with the new default values.

Example:

```c
#!/usr/bin/env python

OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
p = PseudocolorAttributes()
p.colorTableName = "calewhite"
p.minFlag,p.maxFlag = 1,1
p.min,p.max = -5.0, 8.0
SetPlotOptions(p)
DrawPlots()
```

2.4.219 SetGlobalLineoutAttributes

Synopsis:

`SetGlobalLineoutAttributes(atts) -> integer`


return type  [CLI_return_t] The SetGlobalLineoutAttributes function returns 1 on success and 0 on failure.

Description:

The SetGlobalLineoutAttributes function allows you to set global lineout options that are used in the creation of all lineouts. You can, for example, specify the destination window and the number of sample points for lineouts.

Example:

```c
#!/usr/bin/env python

SetWindowLayout(4)
OpenDatabase("/usr/gapps/visit/data/noise.silo")
AddPlot("Pseudocolor", "hgslice")
DrawPlots()
gla = GetGlobalLineoutAttributes()
gla.createWindow = 0
gla.windowId = 4
gla.samplingOn = 1
gla.numSamples = 150
SetGlobalLineoutAttributes(gla)
Lineout((-5,-8), (-3.5, 8))
```
### SetInteractorAttributes

**Synopsis:**

\[
\text{SetInteractorAttributes}(\text{atts}) \rightarrow \text{integer}
\]

**atts** [InteractorAttributes object] An InteractorAttributes object that contains the new interactor attributes that you want to use.

**return type** [CLI_return_t] SetInteractorAttributes returns 1 on success and 0 on failure.

**Description:**

The SetInteractorAttributes function is used to set certain interactor properties. Interactors, can be thought of as how mouse clicks and movements are translated into actions in the vis window. To set the interactor attributes, first get the interactor attributes using the GetInteractorAttributes function. Once you’ve set the object’s properties, call the SetInteractorAttributes function to make VisIt use the new interactor attributes. The SetDefaultInteractorAttributes function sets the default interactor attributes, which are used for new visualization windows. The default interactor attributes can also be saved to the VisIt configuration file to ensure that future VisIt sessions have the right default interactor attributes.

**Example:**

```python
#% visit -cli
ia = GetInteractorAttributes()
print(ia)
ia.showGuidelines = 0
SetInteractorAttributes(ia)
```

### SetKeyframeAttributes

**Synopsis:**

\[
\text{SetKeyframeAttributes}(\text{kfAtts}) \rightarrow \text{integer}
\]

**kfAtts** [KeyframeAttributes object] A KeyframeAttributes object that contains the new keyframing attributes to use.

**return type** [CLI_return_t] SetKeyframeAttributes returns 1 on success and 0 on failure.

**Description:**

Use the SetKeyframeAttributes function when you want to change VisIt’s keyframing settings. You must pass a KeyframeAttributes object, which you can create using the GetKeyframeAttributes function. The KeyframeAttributes object must contain the keyframing settings that you want VisIt to use. For example, you would use the SetKeyframeAttributes function if you wanted to turn on keyframing mode and set the number of animation frames.

**Example:**

```python
#% visit -cli
k = GetKeyframeAttributes()
print(k)
k.enabled,k.nFrames,k.nFramesWasUserSet = 1, 100, 1
SetKeyframeAttributes(k)
```
2.4.222 SetLight

Synopsis:

SetLight(index, light) -> integer

index [integer] A zero-based integer index into the light list. Index can be in the range [0,7].
light [LightAttributes object] A LightAttributes object containing the properties to use for the specified light.

return type [CLI_return_t] SetLight returns 1 on success and 0 on failure.

Description:
The SetLight function sets the attributes for a specific light.

Example:

```python
# visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "w")
p = PseudocolorAttributes()
p.colorTableName = "xray"
SetPlotOptions(p)
DrawPlots()
light = GetLight(0)
print(light)
light.enabledFlag = 1
light.direction = (0,-1,0)
light.color = (255,0,0,255)
SetLight(0, light)
light.color,light.direction = (0,255,0,255), (-1,0,0)
SetLight(1, light)
```

2.4.223 SetMachineProfile

Synopsis:

SetMachineProfile(MachineProfile) -> integer


Description:
Sets the input machine profile in the HostProfileList, replaces if one already exists Otherwise adds to the list

2.4.224 SetMaterialAttributes

Synopsis:

SetMaterialAttributes(atts) -> integer

atts [MaterialAttributes object] A MaterialAttributes object containing the new settings.

return type [CLI_return_t] Both functions return 1 on success and 0 on failure.

Description:
The `SetMaterialAttributes` function takes a `MaterialAttributes` object and makes VisIt use the material settings that it contains. You use the `SetMaterialAttributes` function when you want to change how VisIt performs material interface reconstruction. The `SetDefaultMaterialAttributes` function sets the default material attributes, which are saved to the config file and are also used by new visualization windows.

**Example:**

```python
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/allinone00.pdb")
AddPlot("Pseudocolor", "mesh/mixvar")
p = PseudocolorAttributes()
p.min,p.minFlag = 4.0, 1
p.max,p.maxFlag = 13.0, 1
SetPlotOptions(p)
DrawPlots()
# Tell VisIt to always do material interface reconstruction.
m = GetMaterialAttributes()
m.forceMIR = 1
SetMaterialAttributes(m)
ClearWindow()
# Redraw the plot forcing VisIt to use the mixed variable information.
DrawPlots()
```

### 2.4.225 SetMeshManagementAttributes

**Synopsis:**

```python
GetMeshManagementAttributes() -> MeshmanagementAttributes object
```

**return type** [MeshmanagementAttributes object] Returns a MeshmanagementAttributes object.

**Description:**

The `GetMeshmanagementAttributes` function returns a MeshmanagementAttributes object that contains VisIt’s current mesh discretization settings. You can set properties on the MeshManagementAttributes object and then pass it to `SetMeshManagementAttributes` to make VisIt use the new material attributes that you’ve specified.

**Example:**

```python
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/csg.silo")
AddPlot("Mesh", "csgmesh")
DrawPlots()
# Tell VisIt to always do material interface reconstruction.
mma = GetMeshManagementAttributes()
mma.discretizationTolerance = (0.01, 0.025)
SetMeshManagementAttributes(mma)
ClearWindow()
# Redraw the plot forcing VisIt to use the mixed variable information.
DrawPlots()
```

### 2.4.226 SetNamedSelectionAutoApply

**Synopsis:**

```python
```
SetNamedSelectionAutoApply(flag) -> integer

**flag** [integer] An integer flag. Non-zero values turn on selection auto apply mode.

**return type** [CLI_return_t] The `SetNamedSelectionAutoApply` function returns 1 on success and 0 on failure.

**Description:**

Named selections are often associated with plots for their data source. When those plots update, their named selections can be updated, which in turn will update any plots that use the named selection. When this mode is enabled, changes to a named selection’s originating plot will cause the selection to be updated automatically.

**Example:**

```
SetNamedSelectionAutoApply(1)
```

### 2.4.227 SetOperatorOptions

**Synopsis:**

```
SetOperatorOptions(atts) -> integer
SetOperatorOptions(atts, operatorIndex) -> integer
SetOperatorOptions(atts, operatorIndex, all) -> integer
```

**atts** [operator attributes object] Any type of operator attributes object.

**operatorIndex** [integer] An optional zero-based integer that serves as an index into the active plot’s operator list. Use this argument if you want to set the operator attributes for a plot that has multiple instances of the same type of operator. For example, if the active plot had a Transform operator followed by a Slice operator followed by another Transform operator and you wanted to adjust the attributes of the second Transform operator, you would pass an operatorIndex value of 2.

**all** [integer] An optional integer argument that tells the function to apply the operator attributes to all plots containing the specified operator if the value of the argument is non-zero.

**return type** [CLI_return_t] All functions return an integer value of 1 for success and 0 for failure.

**Description:**

Each operator in VisIt has a group of attributes that controls the operator. To set the attributes for an operator, first create an operator attributes object. This is done by calling a function which is the name of the operator plus the word “Attributes”. For example, a Slice operator’s operator attributes object is created and returned by the `SliceAttributes` function. Assign the new operator attributes object into a variable and set its fields. After setting the desired fields in the operator attributes object, pass the object to the `SetOperatorOptions` function. The `SetOperatorOptions` function determines the type of operator to which the operator attributes object applies and sets the attributes for that operator type. To set the default plot attributes, use the `SetDefaultOperatorOptions` function. Setting the default attributes ensures that all future instances of a certain operator are initialized with the new default values. Note that there is no `SetOperatorOptions(atts, all)` variant of this call. To set operator options for all plots that have a an instance of the associated operator, you must first make all plots active with `SetActivePlots()` and then use the `SetOperatorOptions(atts)` variant.

**Example:**

```
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
```
AddPlot("Pseudocolor", "u")
AddPlot("Mesh", "mesh1")
AddOperator("Slice", 1) # Add the operator to both plots
a = SliceAttributes()
a.normal, a.upAxis = (0,0,1), (0,1,0)
# Only set the attributes for the active plot.
SetOperatorOptions(a)
DrawPlots()

2.4.228 SetPickAttributes

Synopsis:
SetPickAttributes(atts) -> integer

atts  [PickAttributes object] A PickAttributes object containing the new pick settings.
return type  [CLI_return_t] All functions return 1 on success and 0 on failure.

Description:
The SetPickAttributes function changes the pick attributes that are used when VisIt picks on plots. The
pick attributes allow you to format your pick output in various ways and also allows you to select auxiliary
pick variables.

Example:
OpenDatabase("/usr/gapps/visit/data/noise.silo")
AddPlot("Pseudocolor", "hgslice")
DrawPlots()
ZonePick(coord=(-5,5,0))
p = GetPickAttributes()
p.showTimeStep = 0
p.showMeshName = 0
p.showZoneId = 0
SetPickAttributes(p)
ZonePick(coord=(0,5,0))

2.4.229 SetPipelineCachingMode

Synopsis:
SetPipelineCachingMode(mode) -> integer

return type  [CLI_return_t] The SetPipelineCachingMode function returns 1 for success and 0 for failure.

Description:
The SetPipelineCachingMode function turns pipeline caching on or off in the viewer. When pipeline
caching is enabled, animation timesteps are cached for fast playback. This can be a disadvantage for large
databases or for plots with many timesteps because it increases memory consumption. In those cases, it
is often useful to disable pipeline caching so the viewer does not use as much memory. When the viewer
does not cache pipelines, each plot for a timestep must be recalculated each time the timestep is visited.

Example:
2.4.230 SetPlotDatabaseState

Synopsis:

SetPlotDatabaseState(index, frame, state)

index  [integer] A zero-based integer index that is the plot’s location in the plot list.
frame  [integer] A zero-based integer index representing the animation frame for which we’re going to add a database keyframe.
state  [integer] A zero-based integer index representing the database time state that we’re going to use at the specified animation frame.

Description:

The SetPlotDatabaseState function is used when VisIt is in keyframing mode to add a database keyframe for a specific plot. VisIt uses database keyframes to determine which database state is to be used for a given animation frame. Database keyframes can be used to stop “database time” while “animation time” continues forward and they can also be used to make “database time” go in reverse, etc.

Example:

```python
## visit -cli
SetPipelineCachingMode(0) # Disable caching
OpenDatabase("/usr/gapps/visit/data/wave.visit")
AddPlot("Pseudocolor", "pressure")
AddPlot("Mesh", "quadmesh")
DrawPlots()
for state in range(TimeSliderGetNStates()):
    SetTimeSliderState(state)
```

2.4.231 SetPlotDescription

Synopsis:

SetPlotDescription(index, description) -> integer

index  [integer] The integer index of the plot within the plot list.
**description**  [list] A new description string that will be shown in the plot list so the plot can be identified readily.

**return type**  [CLI_return_t] The SetPlotDescription function returns 1 on success and 0 on failure.

**Description:**
Managing many related plots can be a complex task. This function lets users provide meaningful descriptions for each plot so they can more easily be identified in the plot list.

**Example:**
```
SetPlotDescription(0, 'Mesh for reflected pressure plot')
```

### 2.4.232 SetPlotFollowsTime

**Synopsis:**
```
SetPlotFollowsTime(val) -> integer
```

**val**  [integer] An optional integer flag indicating whether the plot should follow the time slider. The default behavior is for the plot to follow the time slider.

**return type**  [CLI_return_t] The function returns 1 on success and 0 on failure.

**Description:**
SetPlotFollowsTime can let you set whether the active plot follows the time slider.

**Example:**
```
SetPlotFollowsTime()
```

### 2.4.233 SetPlotFrameRange

**Synopsis:**
```
SetPlotFrameRange(index, start, end)
```

**index**  [integer] A zero-based integer representing an index into the plot list.

**start**  [integer] A zero-based integer representing the animation frame where the plot first appears in the visualization.

**end**  [integer] A zero-based integer representing the animation frame where the plot disappears from the visualization.

**Description:**
The SetPlotFrameRange function sets the start and end frames for a plot when VisIt is in keyframing mode. Outside of this frame range, the plot does not appear in the visualization. By default, plots are valid over the entire range of animation frames when they are first created. Frame ranges allow you to construct complex animations where plots appear and disappear dynamically.

**Example:**
```
# visit -cli
OpenDatabase("/usr/gapps/visit/data/wave.visit")
k = GetKeyframeAttributes()
nFrames = 20
k.enabled, k.nFrames, k.nFramesWasUserSet = 1, nFrames, 1
SetKeyframeAttributes(k)
```

(continues on next page)
AddPlot("Pseudocolor", "pressure")
AddPlot("Mesh", "quadmesh")
DrawPlots()

# Make the Pseudocolor plot take up the first half of the animation frames
# before it disappears.
SetPlotFrameRange(0, 0, nFrames/2-1)

# Make the Mesh plot take up the second half of the animation frames.
SetPlotFrameRange(1, nFrames/2, nFrames-1)

for state in range(TimeSliderGetNStates()):
    SetTimeSliderState(state)
    SaveWindow()

2.4.234 SetPlotOptions

Synopsis:

SetPlotOptions(atts) -> integer

atts [plot attributes object] Any type of plot attributes object.

return type [CLI_return_t] All functions return an integer value of 1 for success and 0 for failure.

Description:

Each plot in VisIt has a group of attributes that controls the appearance of the plot. To set the attributes for a plot, first create a plot attributes object. This is done by calling a function which is the name of the plot plus the word “Attributes”. For example, a Pseudocolor plot’s plotattributes object is created and returned by the PseudocolorAttributes function. Assign the new plot attributes object into a variable and set its fields. After setting the desired fields in the plot attributes object, pass the object to the SetPlotOptions function. The SetPlotOptions function determines the type of plot to which the plot attributes object applies and sets the attributes for that plot type. To set the default plot attributes, use the SetDefaultPlotOptions function. Setting the default attributes ensures that all future instances of a certain plot are initialized with the new default values.

Example:

```c
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
p = PseudocolorAttributes()
p.colorTableName = "calewhite"
p.minFlag,p.maxFlag = 1,1
p.min,p.max = -5.0, 8.0
SetPlotOptions(p)
DrawPlots()
```

2.4.235 SetPlotOrderToFirst

Synopsis:

SetPlotOrderToFirst(index) -> integer

index [integer] The integer index of the plot within the plot list.

return type [CLI_return_t] The SetPlotOrderToFirst function returns 1 on success and 0 on failure.
Description:

Move the i’th plot in the plot list to the start of the plot list.

Example:

```
AddPlot('Mesh', 'mesh')
AddPlot('Pseudocolor', 'pressure')  
# Make the Pseudocolor plot first in the plot list
SetPlotOrderToFirst(1)
```

### 2.4.236 SetPlotOrderToLast

**Synopsis:**

```
SetPlotOrderToLast(index) -> integer
```

*index* [integer] The integer index of the plot within the plot list.

**return type** [CLI_return_t] The SetPlotOrderToLast function returns 1 on success and 0 on failure.

**Description:**

Move the i’th plot in the plot list to the end of the plot list.

**Example:**

```
AddPlot('Mesh', 'mesh')
AddPlot('Pseudocolor', 'pressure')  
# Make the Mesh plot last in the plot list
SetPlotOrderToLast(0)
```

### 2.4.237 SetPlotSILRestriction

**Synopsis:**

```
SetPlotSILRestriction(silr) -> integer
SetPlotSILRestriction(silr, all) -> integer
```

*silr* [SIL restriction object] A SIL restriction object.

*all* An optional argument that tells the function if the SIL restriction should be applied to all plots in the plot list.

**return type** [CLI_return_t] The SetPlotSILRestriction function returns an integer value of 1 for success and 0 for failure.

**Description:**

VisIt allows the user to select subsets of databases. The description of the subset is called a Subset Inclusion Lattice Restriction, or SIL restriction. The SIL restriction allows databases to be subselected in several different ways. The VisIt Python Interface provides the SetPlotSILRestriction function to allow Python scripts to turn off portions of the plotted database. The SetPlotSILRestriction function accepts a SILRestriction object that contains the SIL restriction for the active plots. The optional all argument is an integer that tells the function to apply the SIL restriction to all plots when the value of the argument is non-zero. If the all argument is not supplied, then the SIL restriction is only applied to the active plots.

**Example:**
### 2.4.238 SetPrecisionType

**Synopsis:**

SetPrecisionType(typeAsInt)
SetPrecisionType(typeAsString)

**typeAsInt** [double] Precision type specified as an integer. 0 = float 1 = native 2 = double

**typeAsString** [string] Precision type specified as a string. Options are ‘float’, ‘native’, and ‘double’.

**Description:**

The `SetPrecisionType` function sets the floating point precision used by VisIt’s pipeline. The function accepts a single argument either an integer or string representing the precision desired. 0 = “float”, 1 = “native”, 2 = “double”

**Example:**

SetPrecisionType("double")
SetPrecisionType(2)

### 2.4.239 SetPreferredFileFormats

**Synopsis:**

SetPreferredFileFormats(pluginIDs) -> integer

**pluginIDs** [tuple] A tuple of plugin IDs to be attempted first when opening files.

**return type** [CLI_return_t] The `SetPreferredFileFormats` method does not return a value.

**Description:**

The `SetPreferredFileFormats` method is a way to set the list of file format reader plugins which are tried before any others. These IDs must be full IDs, not just names, and are tried in order.

**Example:**

SetPreferredFileFormats('Silo_1.0')
SetPreferredFileFormats(('Silo_1.0','PDB_1.0'))

### 2.4.240 SetPrinterAttributes

**Synopsis:**

```bash
$ visit -cli
OpenDatabase("/usr/gapps/visit/data/multi_curv2d.silo")
AddPlot("Subset", "mat1")
silr = SILRestriction()
silr.TurnOffSet(silr.SetsInCategory('mat1')[1])
SetPlotSILRestriction(silr)
DrawPlots()
```
SetPrinterAttributes(atts)

atts  [PrinterAttributes object] A PrinterAttributes object.

Description:
The SetPrinterAttributes function sets the printer attributes. VisIt uses the printer attributes to determine how the active visualization window should be printed. The function accepts a single argument which is a PrinterAttributes object containing the printer attributes to use for future printing. VisIt allows images to be printed to a network printer or to a PostScript file that can be printed later by other applications.

Example:

```
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/curv2d.silo")
AddPlot("Surface", "v")
DrawPlots()
# Make it print to a file.
p = PrinterAttributes()
p.outputToFile = 1
p.outputToFileName = "printfile"
SetPrinterAttributes(p)
PrintWindow()
```

### 2.4.241 SetQueryFloatFormat

Synopsis:

SetQueryFloatFormat(format_string)

format_string  [string] A string object that provides a printf style floating point format.

Description:
The SetQueryFloatFormat method sets a printf style format string that is used by VisIt’s queries to produce textual output.

Example:

```
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/rect2d.silo")
AddPlot("Pseudocolor", "d")
DrawPlots()
# Set floating point format string.
SetQueryFloatFormat("%.1f")
Query("MinMax")
# Set format back to default "%g".
SetQueryFloatFormat("%g")
Query("MinMax")
```

### 2.4.242 SetQueryOutputToObject

Synopsis:

SetQueryOutputToObject()
Description:

SetQueryOutputToObject changes the return type of future Queries to the ‘object’ or Python dictionary form. This is the same object that would be returned by calling ‘GetQueryOutputObject()’ after a Query call. All other output modes are still available after the Query call (eg GetQueryOutputValue(), GetQueryOutputObject(), GetQueryOutputString()).

Example:

```python
# % visit -cli
OpenDatabase("/usr/gapps/visit/data/rect2d.silo")
AddPlot("Pseudocolor", "d")
DrawPlots()
# Set query output type.
SetQueryOutputToObject()
query_output = Query("MinMax")
print(query_output)
```

2.4.243 SetQueryOutputToString

Synopsis:

SetQueryOutputToString()

Description:

SetQueryOutputToString changes the return type of future Queries to the ‘string’ form. This is the same as what would be returned by calling ‘GetQueryOutputString’ after a Query call. All other output modes are still available after the Query call (eg GetQueryOutputValue(), GetQueryOutputObject(), GetQueryOutputString()).

Example:

```python
# % visit -cli
OpenDatabase("/usr/gapps/visit/data/rect2d.silo")
AddPlot("Pseudocolor", "d")
DrawPlots()
# Set query output type.
SetQueryOutputToString()
query_output = Query("MinMax")
print(query_output)
'```

2.4.244 SetQueryOutputToValue

Synopsis:

SetQueryOutputToValue()

Description:

SetQueryOutputToValue changes the return type of future Queries to the ‘value’ form. This is the same as what would be returned by calling ‘GetQueryOutputValue()’ after a Query call. All other output modes
are still available after the Query call (eg GetQueryOutputValue(), GetQueryOutputObject(), GetQueryOutputString()).

Example:

```python
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/rect2d.silo")
AddPlot("Pseudocolor", "d")
DrawPlots()
# Set query output type.
SetQueryOutputToValue()
query_output = Query("MinMax")
print(query_output)
(0.02357020415365696, 0.9489759802818298)
```

### 2.4.245 SetQueryOverTimeAttributes

**Synopsis:**

SetQueryOverTimeAttributes(atts) -> integer

**atts** [QueryOverTimeAttributes object] A QueryOverTimeAttributes object containing the new settings to use for queries over time.

**return type** [CLI_return_t] All functions return 1 on success and 0 on failure.

**Description:**

The SetQueryOverTimeAttributes function changes the settings that VisIt uses for query over time. The SetDefaultQueryOverTimeAttributes function changes the settings that new visualization windows inherit for doing query over time. Finally, the ResetQueryOverTimeAttributes function forces VisIt to use the stored default query over time attributes instead of the previous settings.

Example:

```python
#% visit -cli
SetWindowLayout(4)
OpenDatabase("/usr/gapps/visit/data/allinone00.pdb")
AddPlot("Pseudocolor", "mesh/mixvar")
DrawPlots()
qot = GetQueryOverTimeAttributes()
# Make queries over time go to window 4.
qot.createWindow,q.windowId = 0, 4
SetQueryOverTimeAttributes(qot)
QueryOverTime("Min")
# Make queries over time only use half of the number of time states.
qot.endTimeFlag,qot.endTime = 1, GetDatabaseNStates() / 2
SetQueryOverTimeAttributes(qot)
QueryOverTime("Min")
ResetView()
```

### 2.4.246 SetRemoveDuplicateNodes

**Synopsis:**

SetRemoveDuplicateNodes(val) -> integer
**val** [integer] Either a zero (false) or non-zero (true) integer value to indicate if duplicate nodes in fully disconnected unstructured grids should be automatically removed by visit.

**return type** [CLI_return_t] The SetRemoveDuplicateNodes function returns 1 on success and 0 on failure.

**Description:**

The SetRemoveDuplicateNodes function sets a boolean in the global attributes indicating whether or not duplicate nodes in fully disconnected unstructured grids should be automatically removed. The default behavior is for the original grid to be left as read, which may slow down VisIt’s performance for extraordinary large meshes. Turning this feature off tells VisIt to remove the duplicate nodes after the mesh is read, but before further processing in VisIt.

**Example:**

```bash
#% visit -cli
SetRemoveDuplicateNodes(1) # turn this feature on
SetRemoveDuplicateNodes(0) # turn this feature off
```

### 2.4.247 SetRenderingAttributes

**Synopsis:**

SetRenderingAttributes(atts) -> integer

**atts** [RenderingAttributes object] A RenderingAttributes object that contains the rendering attributes that we want to make VisIt use.

**return type** [CLI_return_t] The SetRenderingAttributes function returns 1 on success and 0 on failure.

**Description:**

The SetRenderingAttributes makes VisIt use the rendering attributes stored in the specified RenderingAttributes object. The RenderingAttributes object stores rendering attributes such as: scalable rendering options, shadows, specular highlights, display lists, etc.

**Example:**

```bash
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/noise.silo")
AddPlot("Surface", "hgslice")
DrawPlots()
v = GetView2D()
v.viewNormal = (-0.215934, -0.454611, 0.864119)
v.viewUp = (0.973938, -0.163188, 0.157523)
v.imageZoom = 1.64765
SetView3D(v)
light = GetLight(0)
light.direction = (0,1,-1)
SetLight(0, light)
r = GetRenderingAttributes()
print(r)
r.scalableActivationMode = r.Always
r.doShadowing = 1
SetRenderingAttributes(r)
```
2.4.248 SetSaveWindowAttributes

Synopsis:

SetSaveWindowAttributes(atts)

atts [SaveWindowAttributes object] A SaveWindowAttributes object.

Description:

The SetSaveWindowAttributes function sets the format and filename that are used to save windows when the SaveWindow function is called. The contents of the active visualization window can be saved as TIFF, JPEG, RGB, PPM, PNG images or they can be saved as curve, Alias Wavefront Obj, or VTK geometry files. To set the SaveWindowAttributes, create a SaveWindowAttributes object using the SaveWindowAttributes function and assign it into a variable. Set the fields in the object and pass it to the SetSaveWindowAttributes function.

Example:

```python
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/curv3d.silo")
AddPlot("Pseudocolor", "d")
DrawPlots()
# Set the save window attributes
s = SaveWindowAttributes()
s.fileName = "test"
s.format = s.JPG
s.progressive = 1
s.fileName = "test"
SetSaveWindowAttributes(s)
# Save the window
SaveWindow()
```

2.4.249 SetTimeSliderState

Synopsis:

SetTimeSliderState(state) -> integer

state [integer] A zero-based integer containing the time state that we want to make active.

return type [CLI_return_t] The SetTimeSliderState function returns 1 on success and 0 on failure.

Description:

The SetTimeSliderState function sets the time state for the active time slider. This is the function to use if you want to animate through time or change the current keyframe frame.

Example:

```python
#% visit -cli
path = "/usr/gapps/visit/data/

dbs = (path + "dbA00.pdb", path + "dbB00.pdb", path + "dbC00.pdb")
for db in dbs:
    OpenDatabase(db)
    AddPlot("FilledBoundary", "material(mesh)"
    DrawPlots()
CreateDatabaseCorrelation("common", dbs, 1)
```
tsNames = GetWindowInformation().timeSliders
for ts in tsNames:
    SetActiveTimeSlider(ts)
for state in list(range(TimeSliderGetNStates())) + [0]:
    SetTimeSliderState(state)

### 2.4.250 SetTreatAllDBsAsTimeVarying

**Synopsis:**

SetTreatAllDBsAsTimeVarying(val) -> integer

**val** [integer] Either a zero (false) or non-zero (true) integer value to indicate if all databases should be treated as time varying (true) or not (false).

**return type** [CLI_return_t] The SetTreatAllDBsAsTimeVarying function returns 1 on success and 0 on failure.

**Description:**

The SetTreatAllDBsAsTimeVarying function sets a boolean in the global attributes indicating if all databases should be treated as time varying or not. Ordinarily, VisIt tries to minimize file I/O and database interaction by avoiding re-reading metadata that is ‘time-invariant’ and, therefore, assumed to be the same in a database from one time step to the next. However, sometimes, portions of the metadata, such as the list of variable names and/or number of domains, does in fact vary. In this case, VisIt can actually fail to acknowledge the existence of new variables in the file. Turning this feature on forces VisIt to re-read metadata each time the time-state is changed.

**Example:**

```python
#% visit -cli
SetTreatAllDBsAsTimeVarying(1) # turn this feature on
SetTreatAllDBsAsTimeVarying(0) # turn this feature off
```

### 2.4.251 SetTryHarderCyclesTimes

**Synopsis:**

SetTryHarderCyclesTimes(val) -> integer

**val** [integer] Either a zero (false) or non-zero (true) integer value to indicate if VisIt read cycle/time information for all timesteps when opening a database.

**return type** [CLI_return_t] The SetTryHarderCyclesTimes function returns 1 on success and 0 on failure.

**Description:**

For certain classes of databases, obtaining cycle/time information for all time states in the database is an expensive operation, requiring each file to be opened and queried. The cost of the operation gets worse the more time states there are in the database. Ordinarily, VisIt does not bother to query each time state for precise cycle/time information. In fact, often VisIt can guess this information from the filename(s) comprising the database. However, turning this feature on will force VisIt to obtain accurate cycle/time information for all time states by opening and querying all file(s) in the database.

**Example:**

```python
#% visit -cli
SetTryHarderCyclesTimes(1) # turn this feature on
SetTryHarderCyclesTimes(0) # turn this feature off
```
#% visit -cli
SetTryHarderCyclesTimes(1) # Turn this feature on
SetTryHarderCyclesTimes(0) # Turn this feature off

## 2.4.252 SetUltraScript

**Synopsis:**

SetUltraScript(filename) -> integer

**filename** [string] The name of the file to be used as the ultra script when LoadUltra is called.

**return type** [CLI_return_t] The SetUltraScript function returns 1.

**Description:**

Set the path to the script to be used by the LoadUltra command. Normal users do not need to use this function.

## 2.4.253 SetView2D

**Synopsis:**

SetView2D(View2DAttributes) -> integer

**view** [ViewAttributes object] A ViewAttributes object containing the view.

**return type** [CLI_return_t] All functions returns 1 on success and 0 on failure.

**Description:**

The view is a crucial part of a visualization since it determines which parts of the database are examined. The VisIt Python Interface provides four functions for setting the view: SetView2D, SetView3D, SetViewCurve, and SetViewAxisArray. If the visualization window contains 2D plots, use the SetView2D function. To set the view, first create the appropriate ViewAttributes object and set the object’s fields to set a new view. After setting the fields, pass the object to the matching SetView function. A common use of the SetView functions is to animate the view to produce simple animations where the camera appears to fly around the plots in the visualization window.

**Example:**

```python
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "v")
DrawPlots()
va = GetView3D()
va.RotateAxis(1, 30.0)  # rotate around the y axis 30 degrees.
SetView3D(va)
v0 = GetView3D()
v1 = GetView3D()
v1.camera, v1.viewUp = (1, 1, 1), (-1, 1, -1)
v1.parallelScale = 10.
for i in range(0, 20):
    t = float(i) / 19.
    v2 = (1. - t) * v0 + t * v1
    SetView3D(v2)  # Animate the view.
```
2.4.254 SetView3D

Synopsis:

SetView3D(View3DAttributes) -> integer

view [ViewAttributes object] A ViewAttributes object containing the view.

return type [CLI_return_t] All functions returns 1 on success and 0 on failure.

Description:

The view is a crucial part of a visualization since it determines which parts of the database are examined. The VisIt Python Interface provides four functions for setting the view: SetView2D, SetView3D, SetViewCurve, and SetViewAxisArray. Use the SetView3D function when the visualization window contains 3D plots. To set the view, first create the appropriate ViewAttributes object and set the object’s fields to set a new view. After setting the fields, pass the object to the matching SetView function. A common use of the SetView functions is to animate the view to produce simple animations where the camera appears to fly around the plots in the visualization window. A View3D object also supports the RotateAxis(int axis, double deg) method which mimics the ‘rotx’, ‘roty’ and ‘rotz’ view commands in the GUI.

Example:

```python
# visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "v")
DrawPlots()
va = GetView3D()
va.RotateAxis(1,30.0) # rotate around the y axis 30 degrees.
SetView3D(va)
v0 = GetView3D()
v1 = GetView3D()
v1.camera,v1.viewUp = (1,1,1),(-1,1,-1)
v1.parallelScale = 10.
for i in range(0,20):
    t = float(i) / 19.
    v2 = (1. - t) * v0 + t * v1
    SetView3D(v2) # Animate the view.
```

2.4.255 SetViewAxisArray

Synopsis:

SetViewAxisArray(ViewAxisArrayAttributes) -> integer

view [ViewAttributes object] A ViewAttributes object containing the view.

return type [CLI_return_t] All functions returns 1 on success and 0 on failure.

Description:

The view is a crucial part of a visualization since it determines which parts of the database are examined. The VisIt Python Interface provides four functions for setting the view: SetView2D, SetView3D, SetViewCurve, and SetViewAxisArray. To set the view, first create the appropriate ViewAttributes object and set the object’s fields to set a new view. After setting the fields, pass the object to the matching SetView function. A common use of the SetView functions is to animate the view to produce simple animations where the camera appears to fly around the plots in the visualization window.
Example:

```python
## visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "v")
DrawPlots()
va = GetView3D()
va.RotateAxis(1,30.0) # rotate around the y axis 30 degrees.
SetView3D(va)
v0 = GetView3D()
v1 = GetView3D()
v1.camera,v1.viewUp = (1,1,1),(-1,1,-1)
v1.parallelScale = 10.
for i in range(0,20):
    t = float(i) / 19.
    v2 = (1. - t) * v0 + t * v1
    SetView3D(v2) # Animate the view.
```

### 2.4.256 SetViewCurve

**Synopsis:**

SetViewCurve(ViewCurveAttributes) -> integer

**view** [ViewAttributes object] A ViewAttributes object containing the view.

**return type** [CLI_return_t] All functions returns 1 on success and 0 on failure.

**Description:**

The view is a crucial part of a visualization since it determines which parts of the database are examined. The VisIt Python Interface provides four functions for setting the view: SetView2D, SetView3D, SetViewCurve, and SetViewAxisArray. To set the view, first create the appropriate ViewAttributes object and set the object’s fields to set a new view. After setting the fields, pass the object to the matching SetView function. A common use of the SetView functions is to animate the view to produce simple animations where the camera appears to fly around the plots in the visualization window.

Example:

```python
## visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "v")
DrawPlots()
va = GetView3D()
va.RotateAxis(1,30.0) # rotate around the y axis 30 degrees.
SetView3D(va)
v0 = GetView3D()
v1 = GetView3D()
v1.camera,v1.viewUp = (1,1,1),(-1,1,-1)
v1.parallelScale = 10.
for i in range(0,20):
    t = float(i) / 19.
    v2 = (1. - t) * v0 + t * v1
    SetView3D(v2) # Animate the view.
```
2.4.257 SetViewExtentsType

Synopsis:

```
SetViewExtentsType(type) -> integer
```

- **type** [integer] An integer or a string. Options are 0, 1 and ‘original’, ‘actual’, respectively.
- **return type** [CLI_return_t] SetViewExtentsType returns 1 on success and 0 on failure.

Description:

VisIt can use a plot’s spatial extents in two ways when computing the view. The first way of using the extents is to use the “original” extents, which are the spatial extents before any modifications, such as subset selection, have been made to the plot. This ensures that the view will remain relatively constant for a plot. Alternatively, you can use the “actual” extents, which are the spatial extents of the pieces of the plot that remain after operations such as subset selection.

Example:

```
# visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
SetViewExtentsType("actual")
AddPlot("FilledBoundary", "mat1")
DrawPlots()
v = GetView3D()
v.viewNormal = (-0.618945, 0.450655, 0.643286)
v.viewUp = (0.276106, 0.891586, -0.358943)
SetView3D(v)
mats = GetMaterials()
nmats = len(mats)
# Turn off all but the last material in sequence and watch # the view update each time.
for i in range(nmats-1):
    index = nmats-1-i
    TurnMaterialsOff(mats[index])
    SaveWindow()
SetViewExtentsType("original")
```

2.4.258 SetViewKeyframe

Synopsis:

```
SetViewKeyframe() -> integer
```

- **return type** [CLI_return_t] The SetViewKeyframe function returns 1 on success and 0 on failure.

Description:

The SetViewKeyframe function adds a view keyframe when VisIt is in keyframing mode. View keyframes are used to set the view at crucial points during an animation. Frames that lie between view keyframes have an interpolated view that is based on the view keyframes. You can use the SetViewKeyframe function to create complex camera animations that allow you to fly around (or through) your visualization.

Example:

```
```
2.4.259 SetWindowArea

Synopsis:

```
SetWindowArea(x, y, width, height) -> integer
```

x  [integer] An integer that is the left X coordinate in screen pixels.
y  [integer] An integer that is the top Y coordinate in screen pixels.
width  [integer] An integer that is the width of the window area in pixels.
height  [integer] An integer that is the height of the window area in pixels.

return type  [CLI_return_t] The SetWindowArea function returns 1 on success and 0 on failure.

Description:

The SetWindowArea method sets the area of the screen that can be used by VisIt’s visualization windows. This is useful for making sure windows are a certain size when running a Python script.

Example:

```
import visit
visit.Launch()
visit.SetWindowArea(0, 0, 600, 600)
visit.SetWindowLayout(4)
```

2.4.260 SetWindowLayout

Synopsis:
SetWindowLayout(layout) -> integer

**layout** [integer] An integer that specifies the window layout. (1,2,4,8,9,16 are valid)

**return type** [CLI_return_t] The SetWindowLayout function returns an integer value of 1 for success and 0 for failure.

**Description:**
VisIt’s visualization windows can be arranged in various tiled patterns that allow VisIt to make good use of the screen while displaying several visualization windows. The window layout determines how windows are shown on the screen. The SetWindowLayout function sets the window layout. The layout argument is an integer value equal to 1,2,4,8,9, or 16.

**Example:**
```bash
# visit -cli
SetWindowLayout(2)  # switch to 1x2 layout
SetWindowLayout(4)  # switch to 2x2 layout
SetWindowLayout(8)  # switch to 2x4 layout
```

### 2.4.261 SetWindowMode

**Synopsis:**
SetWindowMode(mode) -> integer


**return type** [CLI_return_t] The SetWindowMode function returns 1 on success and 0 on failure.

**Description:**
VisIt’s visualization windows have various window modes that alter their behavior. Most of the time a visualization window is in “navigate” mode which changes the view when the mouse is moved in the window. The “zoom” mode allows a zoom rectangle to be drawn in the window for changing the view. The “pick” mode retrieves information about the plots when the mouse is clicked in the window. The “lineout” mode allows the user to draw lines which produce curve plots.

**Example:**
```bash
# visit -cli
OpenDatabase("/usr/gapps/visit/data/curv2d.silo")
AddPlot("Pseudocolor", "d")
DrawPlots()
SetWindowMode("zoom")
SetWindowMode("navigate")
SetWindowMode("zoom")
# Draw a rectangle in the visualization window to zoom the plots
```

### 2.4.262 ShowAllWindows

**Synopsis:**
ShowAllWindows() -> integer

**return type** [CLI_return_t] The ShowAllWindows function returns 1 on success and 0 on failure.

**Description:**

The `ShowAllWindows` function tells VisIt's viewer to show all of its visualization windows. The command line interface calls `ShowAllWindows` before giving control to any user-supplied script to ensure that the visualization windows appear as expected. Call the `ShowAllWindows` function when using the VisIt module inside another Python interpreter so the visualization windows are made visible.

**Example:**

```python
## python
import visit
visit.Launch()
visit.ShowAllWindows()
```

### 2.4.263 ShowToolbars

**Synopsis:**

```
ShowToolbars() -> integer
ShowToolbars(allWindows) -> integer
```

- **allWindows** [integer] An integer value that tells VisIt to show the toolbars for all windows when it is non-zero.
- **return type** [CLI_return_t] The `ShowToolbars` function returns 1 on success and 0 on failure.

**Description:**

The `ShowToolbars` function tells VisIt to show the toolbars for the active visualization window or for all visualization windows when the optional `allWindows` argument is provided and is set to a non-zero value.

**Example:**

```python
## visit -cli
SetWindowLayout(4)
HideToolbars(1)
ShowToolbars()
# Show the toolbars for all windows.
ShowToolbars(1)
```

### 2.4.264 Source

**Synopsis:**

```
Source(filename)
```

**Description:**

The `Source` function reads in the contents of a text file and interprets it with the Python interpreter. This is a simple mechanism that allows simple scripts to be included in larger scripts. The `Source` function takes a single string argument that contains the name of the script to execute.

**Example:**

```python
## visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
DrawPlots()
# include another script that does some animation.
Source("Animate.py")
```
### 2.4.265 SuppressMessages

**Synopsis:**

```cpp
SuppressMessages(level) -> integer
```

**level** [integer] An integer value of 1, 2, 3 or 4

**return type** [CLI_return_t] The SuppressMessages function returns the previous suppression level on success and 0 on failure.

**Description:**

The SuppressMessage function sets the suppression level for status messages generated by VisIt. A value of 1 suppresses all types of messages. A value of 2 suppresses Warnings and Messages but does NOT suppress Errors. A value of 3 suppresses Messages but does not suppress Warnings or Errors. A value of 4 does not suppress any messages. The default setting is 4.

**Example:**

```python
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/rect2d.silo")
AddPlot("Pseudocolor", "d")
DrawPlots()
# Turn off Warning and Message messages.
SuppressMessages(2)
SaveWindow()
```

### 2.4.266 SuppressQueryOutputOff

**Synopsis:**

```cpp
SuppressQueryOutputOff() -> integer
```

**return type** [CLI_return_t] The SuppressQueryOutput function returns 1 on success and 0 on failure.

**Description:**

The SuppressQueryOutput function tells VisIt to turn on/off the automatic printing of query output. Query output will still be available via GetQueryOutputString and GetQueryOutputValue.

**Example:**

```python
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/rect2d.silo")
AddPlot("Pseudocolor", "d")
DrawPlots()
# Turn off automatic printing of Query output.
SuppressQueryOutputOn()
Query("MinMax")
print("The min is: %g and the max is: %g" % GetQueryOutputValue())
# Turn on automatic printing of Query output.
SuppressQueryOutputOff()
Query("MinMax")
```
2.4.267 SuppressQueryOutputOn

Synopsis:

SuppressQueryOutputOn() -> integer

return type  [CLI_return_t] The SuppressQueryOutput function returns 1 on success and 0 on failure.

Description:

The SuppressQueryOutput function tells VisIt to turn on/off the automatic printing of query output. Query output will still be available via GetQueryOutputString and GetQueryOutputValue.

Example:

```python
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/rect2d.silo")
AddPlot("Pseudocolor", "d")
DrawPlots()
# Turn off automatic printing of Query output.
SuppressQueryOutputOn()
Query("MinMax")
print("The min is: \%g and the max is: \%g \nGetQueryOutputValue())
# Turn on automatic printing of Query output.
SuppressQueryOutputOff()
Query("MinMax")
```

2.4.268 TimeSliderGetNStates

Synopsis:

TimeSliderGetNStates() -> integer

return type  [CLI_return_t] Returns an integer containing the number of time states for the current time slider.

Description:

The TimeSliderGetNStates function returns the number of time states for the active time slider. Remember that the length of the time slider does not have to be equal to the number of time states in a time-varying database because of database correlations and keyframing. If you want to iterate through time, use this function to determine the number of iterations that are required to reach the end of the active time slider.

Example:

```python
OpenDatabase("/usr/gapps/visit/data/wave.visit")
AddPlot("Pseudocolor", "pressure")
DrawPlots()
for state in range(TimeSliderGetNStates()):
    SetTimeSliderState(state)
    SaveWindow()
```

2.4.269 TimeSliderNextState

Synopsis:

TimeSliderNextState() -> integer
return type  [CLI_return_t] The TimeSliderNextState function returns 1 on success and 0 on failure.

Description:

The TimeSliderNextState function advances the active time slider to the next time slider state.

Example:

```plaintext
# Assume that files are being written to the disk.
#% visit -cli
OpenDatabase("dynamic*.silo database")
AddPlot("Pseudocolor", "var")
AddPlot("Mesh", "mesh")
DrawPlots()
SetTimeSliderState(TimeSliderGetNStates() - 1)
while 1:
    SaveWindow()
    TimeSliderPreviousState()
```

2.4.270 TimeSliderPreviousState

Synopsis:

TimeSliderPreviousState() -> integer

return type  [CLI_return_t] The TimeSliderPreviousState function returns 1 on success and 0 on failure.

Description:

The TimeSliderPreviousState function moves the active time slider to the previous time slider state.

Example:

```plaintext
# Assume that files are being written to the disk.
#% visit -cli
OpenDatabase("dynamic*.silo database")
AddPlot("Pseudocolor", "var")
AddPlot("Mesh", "mesh")
DrawPlots()
while 1:
    TimeSliderNextState()
    SaveWindow()
```

2.4.271 TimeSliderSetState

Synopsis:

TimeSliderSetState(state) -> integer

state  [integer] A zero-based integer containing the time state that we want to make active.

return type  [CLI_return_t] The TimeSliderSetState function returns 1 on success and 0 on failure.

Description:

The TimeSliderSetState function sets the time state for the active time slider. This is the function to use if you want to animate through time or change the current keyframe frame.

Example:
```python
#$ visit -cli
path = "\(/usr/gapps/visit/data/"
dbs = (path + "dbA00.pdb", path + "dbB00.pdb", path + "dbC00.pdb")
for db in dbs:
  OpenDatabase(db)
  AddPlot("FilledBoundary", "material(mesh")
  DrawPlots()
CreateDatabaseCorrelation("common", dbs, 1)
tsNames = GetWindowInformation().timeSliders
for ts in tsNames:
  SetActiveTimeSlider(ts)
for state in list(range(TimeSliderGetNStates())) + [0]:
  TimeSliderSetState(state)
```

### 2.4.272 ToggleBoundingBoxMode

**Synopsis:**

`ToggleBoundingBoxMode()` -> integer

**Return type**  [CLI_return_t] All functions return 1 on success and 0 on failure.

**Description:**

The visualization window has various modes that affect its behavior and the VisIt Python Interface provides a few functions to toggle some of those modes. The ToggleBoundingBoxMode function toggles bounding box mode on and off. When the visualization window is in bounding box mode, any plots it contains are hidden while the view is being changed so the window redraws faster.

**Example:**

```python
#$ visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
DrawPlots()
# Turn on spin mode.
ToggleSpinMode()
# Rotate the plot interactively using the mouse and watch it keep spinning
# after the mouse release.
# Turn off spin mode.
ToggleSpinMode()
```

### 2.4.273 ToggleCameraViewMode

**Synopsis:**

`ToggleCameraViewMode()` -> integer

**Return type**  [CLI_return_t] All functions return 1 on success and 0 on failure.

**Description:**

The visualization window has various modes that affect its behavior and the VisIt Python Interface provides a few functions to toggle some of those modes. The ToggleCameraViewMode function toggles camera view mode on and off. When the visualization window is in camera view mode, the view is updated using any view keyframes that have been defined when VisIt is in keyframing mode.
Example:

```c
% visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
DrawPlots()
# Turn on spin mode.
ToggleSpinMode()
# Rotate the plot interactively using the mouse and watch it keep spinning after the mouse release.
# Turn off spin mode.
ToggleSpinMode()
```

### 2.4.274 ToggleFullScreenMode

**Synopsis:**

```c
ToggleFullScreenMode() -> integer
```

**return type**  [CLI_return_t] All functions return 1 on success and 0 on failure.

**Description:**

The visualization window has various modes that affect its behavior and the VisIt Python Interface provides a few functions to toggle some of those modes. The ToggleFullScreenMode function toggles full-frame mode on and off. When the visualization window is in fullframe mode, the viewport is stretched non-uniformly so that it covers most of the visualization window. While not maintaining a 1:1 aspect ratio, it does make better use of the visualization window.

Example:

```c
% visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
DrawPlots()
# Turn on spin mode.
ToggleSpinMode()
# Rotate the plot interactively using the mouse and watch it keep spinning after the mouse release.
# Turn off spin mode.
ToggleSpinMode()
```

### 2.4.275 ToggleLockTime

**Synopsis:**

```c
ToggleLockTime() -> integer
```

**return type**  [CLI_return_t] All functions return 1 on success and 0 on failure.

**Description:**

The visualization window has various modes that affect its behavior and the VisIt Python Interface provides a few functions to toggle some of those modes. The ToggleLockTime function turns time locking on and off in a visualization window. When time locking is on in a visualization window, VisIt creates a database correlation that works for the databases in all visualization windows that are time-locked. When
you change the time state using the time slider for the the afore-mentioned database correlation, it has the effect of updating time in all time-locked visualization windows.

**Example:**

```python
# visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
DrawPlots()
# Turn on spin mode.
ToggleSpinMode()
# Rotate the plot interactively using the mouse and watch it keep spinning
# after the mouse release.
# Turn off spin mode.
ToggleSpinMode()
```

### 2.4.276 ToggleLockTools

**Synopsis:**

```python
ToggleBoundingBoxMode() -> integer
ToggleCameraViewMode() -> integer
ToggleFullFrameMode() -> integer
ToggleLockTime() -> integer
ToggleLockViewMode() -> integer
ToggleMaintainViewMode() -> integer
ToggleSpinMode() -> integer
```

**return type** [CLI_return_t] All functions return 1 on success and 0 on failure.

**Description:**

The visualization window has various modes that affect its behavior and the VisIt Python Interface provides a few functions to toggle some of those modes. The ToggleBoundingBoxMode function toggles bounding box mode on and off. When the visualization window is in bounding box mode, any plots it contains are hidden while the view is being changed so the window redraws faster. The ToggleCameraViewMode function toggles camera view mode on and off. When the visualization window is in camera view mode, the view is updated using any view keyframes that have been defined when VisIt is in keyframing mode. The ToggleFullFrameMode function toggles fullframe mode on and off. When the visualization window is in fullframe mode, the viewport is stretched non-uniformly so that it covers most of the visualization window. While not maintaining a 1:1 aspect ratio, it does make better use of the visualization window. The ToggleLockTime function turns time locking on and off in a visualization window. When time locking is on in a visualization window, VisIt creates a database correlation that works for the databases in all visualization windows that are time-locked. When you change the time state using the time slider for the the afore-mentioned database correlation, it has the effect of updating time in all time-locked visualization windows. The ToggleLockViewMode function turns lock view mode on and off. When windows are in lock view mode, each view change is broadcast to other windows that are also in lock view mode. This allows windows containing similar plots to be compared easily. The ToggleMaintainViewMode function forces the view, that was in effect when the mode was toggled to be used for all subsequent time states. The ToggleSpinMode function turns spin mode on and off. When the visualization window is in spin mode, it continues to spin along the axis of rotation when the view is changed interactively.

**Example:**
2.4.277 ToggleLockViewMode

Synopsis:

ToggleLockViewMode() -> integer

return type [CLI_return_t] All functions return 1 on success and 0 on failure.

Description:

The visualization window has various modes that affect its behavior and the VisIt Python Interface provides a few functions to toggle some of those modes. The ToggleLockViewMode function turns lock view mode on and off. When windows are in lock view mode, each view change is broadcast to other windows that are also in lock view mode. This allows windows containing similar plots to be compared easily.

Example:

```plaintext
# visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
DrawPlots()
# Turn on spin mode.
ToggleSpinMode()
# Rotate the plot interactively using the mouse and watch it keep spinning
# after the mouse release.
# Turn off spin mode.
ToggleSpinMode()
```

2.4.278 ToggleMaintainViewMode

Synopsis:

ToggleMaintainViewMode() -> integer

return type [CLI_return_t] All functions return 1 on success and 0 on failure.

Description:

The visualization window has various modes that affect its behavior and the VisIt Python Interface provides a few functions to toggle some of those modes. The ToggleMaintainViewMode function forces the view that was in effect when the mode was toggled to be used for all subsequent time states.

Example:

```plaintext
# visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
DrawPlots()
# Turn on spin mode.
ToggleSpinMode()
# Rotate the plot interactively using the mouse and watch it keep spinning
# after the mouse release.
# Turn off spin mode.
ToggleSpinMode()
```
2.4.279 ToggleSpinMode

Synopsis:

```python
ToggleSpinMode() -> integer
```

**return type**  [CLI_return_t] All functions return 1 on success and 0 on failure.

**Description:**

The visualization window has various modes that affect its behavior and the VisIt Python Interface provides a few functions to toggle some of those modes. The ToggleSpinMode function turns spin mode on and off. When the visualization window is in spin mode, it continues to spin along the axis of rotation when the view is changed interactively.

**Example:**

```python
# visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
DrawPlots()
    # Turn on spin mode.
    ToggleSpinMode()
    # Rotate the plot interactively using the mouse and watch it keep spinning
    # after the mouse release.
    # Turn off spin mode.
    ToggleSpinMode()
```

2.4.280 TurnDomainsOff

Synopsis:

```python
TurnDomainsOff() -> integer
TurnDomainsOff(set_name) -> integer
TurnDomainsOff(tuple_set_name) -> integer
```

**set_name**  [string] The name of the set to modify.

**tuple_set_name**  [tuple of strings] A tuple of strings for the sets to modify.

**return type**  [CLI_return_t] The Turn functions return an integer with a value of 1 for success or 0 for failure.

**Description:**

The Turn functions are provided to simplify the removal of material or domain subsets. Instead of creating a SILRestriction object, you can use the Turn functions to turn materials or domains on or off. The
TurnDomainsOff function turns domains off. All of the Turn functions have three possible argument lists. When you do not provide any arguments, the function applies to all subsets in the SIL so if you called the TurnDomainsOff function with no arguments, all domains would be turned off. All functions can also take a string argument, which is the name of the set to modify. For example, you could turn off domain 0 by calling the TurnDomainsOff with a single argument of “domain0” (or the appropriate set name). All of the Turn functions can also be used to modify more than one set if you provide a tuple of set names. After you use the Turn functions to change the SIL restriction, you might want to call the ListMaterials or ListDomains functions to make sure that the SIL restriction was actually modified.

Example:

```python
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
DrawPlots()
TurnMaterialsOff("4") # Turn off material 4
TurnMaterialsOff(("1", "2")) # Turn off materials 1 and 2
TurnMaterialsOn() # Turn on all materials
```

2.4.281 TurnDomainsOn

Synopsis:

- `TurnDomainsOn()` -> integer
- `TurnDomainsOn(set_name)` -> integer
- `TurnDomainsOn(tuple_set_name)` -> integer

- **set_name** [string] The name of the set to modify.
- **tuple_set_name** [tuple of strings] A tuple of strings for the sets to modify.
- **return type** [CLI_return_t] The Turn functions return an integer with a value of 1 for success or 0 for failure.

Description:

The Turn functions are provided to simplify the removal of material or domain subsets. Instead of creating a SILRestriction object, you can use the Turn functions to turn materials or domains on or off. The TurnDomainsOn function turns domains on. All of the Turn functions have three possible argument lists. When you do not provide any arguments, the function applies to all subsets in the SIL so if you called the TurnDomainsOn function with no arguments, all domains would be turned on. All functions can also take a string argument, which is the name of the set to modify. For example, you could turn on domain 0 by calling the TurnDomainsOn with a single argument of “domain0” (or the appropriate set name). All of the Turn functions can also be used to modify more than one set if you provide a tuple of set names. After you use the Turn functions to change the SIL restriction, you might want to call the ListMaterials or ListDomains functions to make sure that the SIL restriction was actually modified.

Example:

```python
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
DrawPlots()
TurnMaterialsOff("4") # Turn off material 4
TurnMaterialsOff(("1", "2")) # Turn off materials 1 and 2
TurnMaterialsOn() # Turn on all materials
```
2.4.282 TurnMaterialsOff

Synopsis:

<table>
<thead>
<tr>
<th>Function</th>
<th>Return Type</th>
</tr>
</thead>
<tbody>
<tr>
<td>TurnMaterialsOff() -&gt; integer</td>
<td></td>
</tr>
<tr>
<td>TurnMaterialsOff(set_name) -&gt; integer</td>
<td></td>
</tr>
<tr>
<td>TurnMaterialsOff(tuple_set_name) -&gt; integer</td>
<td></td>
</tr>
</tbody>
</table>

set_name  [string] The name of the set to modify.

tuple_set_name  [tuple of strings] A tuple of strings for the sets to modify.

return type  [CLI_return_t] The Turn functions return an integer with a value of 1 for success or 0 for failure.

Description:

The Turn functions are provided to simplify the removal of material or domain subsets. Instead of creating a SILRestriction object, you can use the Turn functions to turn materials or domains on or off. The TurnMaterialsOff function turns materials off. All of the Turn functions have three possible argument lists. When you do not provide any arguments, the function applies to all subsets in the SIL so if you called the TurnMaterialsOff function with no arguments, all materials would be turned off. All functions can also take a string argument, which is the name of the set to modify. For example, you could turn off material 0 by calling TurnMaterialsOff with a single argument of “material0” (or the appropriate set name). All of the Turn functions can also be used to modify more than one set if you provide a tuple of set names. After you use the Turn functions to change the SIL restriction, you might want to call the ListMaterials or ListDomains functions to make sure that the SIL restriction was actually modified.

Example:

```bash
# visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
DrawPlots()
TurnMaterialsOff("4")  # Turn off material 4
TurnMaterialsOff("(1", "2")")  # Turn off materials 1 and 2
TurnMaterialsOn()  # Turn on all materials
```

2.4.283 TurnMaterialsOn

Synopsis:

<table>
<thead>
<tr>
<th>Function</th>
<th>Return Type</th>
</tr>
</thead>
<tbody>
<tr>
<td>TurnMaterialsOn() -&gt; integer</td>
<td></td>
</tr>
<tr>
<td>TurnMaterialsOn(string) -&gt; integer</td>
<td></td>
</tr>
<tr>
<td>TurnMaterialsOn(tuple of strings) -&gt; integer</td>
<td></td>
</tr>
</tbody>
</table>

set_name  [string] The name of the set to modify.

tuple_set_name  [tuple of strings] A tuple of strings for the sets to modify.

return type  [CLI_return_t] The Turn functions return an integer with a value of 1 for success or 0 for failure.

Description:

The Turn functions are provided to simplify the removal of material or domain subsets. Instead of creating a SILRestriction object, you can use the Turn functions to turn materials or domains on or off. The TurnMaterialsOn function turns materials on. All of the Turn functions have three possible argument lists. When you do not provide any arguments, the function applies to all subsets in the SIL so if you called the TurnMaterialsOn function with no arguments, all materials would be turned off. All functions can also take a string argument, which is the name of the set to modify. For example, you could turn on
material 0 by calling the TurnMaterialsOn with a single argument of “material0” (or the appropriate set name). All of the Turn functions can also be used to modify more than one set if you provide a tuple of set names. After you use the Turn functions to change the SIL restriction, you might want to call the ListMaterials or ListDomains functions to make sure that the SIL restriction was actually modified.

Example:

```
# visit -cli
OpenDatabase("/usr/gapps/visit/data/globe.silo")
AddPlot("Pseudocolor", "u")
DrawPlots()
TurnMaterialsOff("4")  # Turn off material 4
TurnMaterialsOff(("1", "2"))  # Turn off materials 1 and 2
TurnMaterialsOn()  # Turn on all materials
```

### 2.4.284 UndoView

**Synopsis:**

```
UndoView()
```

**Description:**

When the view changes in the visualization window, it puts the old view on a stack of views. The UndoView function restores the view on top of the stack and removes it. This allows the user to undo up to ten view changes.

Example:

```
# visit -cli
OpenDatabase("/usr/gapps/visit/data/curv2d.silo")
AddPlot("Subset", "mat1")
DrawPlots()
v = GetView2D()
v.windowCoords = (-2.3,2.4,0.2,4.9)
SetView2D(v)
UndoView()
```

### 2.4.285 UpdateNamedSelection

**Synopsis:**

```
UpdateNamedSelection(name) -> integer
UpdateNamedSelection(name, properties) -> integer
```

**name** [string] The name of the selection to update.

**properties** [SelectionProperties object] An optional SelectionProperties object that contains the selection properties to use when reevaluating the selection.

**return type** [CLI_return_t] The UpdateNamedSelection function returns 1 on success and 0 on failure.

**Description:**

This function causes VisIt to reevaluate a named selection using new selection properties. If no selection properties are provided then the selection will be reevaluated using data for the plot that was associated
with the selection when it was created. This is useful if you want to change a plot in several ways before causing its associated named selection to update using the changes.

Example:

```python
s = GetSelection('selection1')
s.selectionType = s.CumulativeQuerySelection
s.histogramType = s.HistogramMatches
s.combineRule = s.CombineOr
s.variables = ('temperature',)
s.variableMins = (2.9,)
s.variableMaxs = (3.1,)
UpdateNamedSelection('selection1', s)
```

### 2.4.286 Version

**Synopsis:**

`Version() -> string`

**return type** [string] The `Version` function return a string that represents VisIt’s version.

**Description:**

The `Version` function returns a string that represents VisIt’s version. The version string can be used in Python scripts to make sure that the VisIt module is a certain version before processing the rest of the Python script.

**Example:**

```python
#% visit -cli
print("We are running VisIt version \$s" % Version())
```

### 2.4.287 WriteConfigFile

**Synopsis:**

`WriteConfigFile()`

**Description:**

The viewer maintains internal settings which determine the default values for objects like plots and operators. The viewer can save out the default values so they can be used in future VisIt sessions. The `WriteConfig` function tells the viewer to write out the settings to the VisIt configuration file.

**Example:**

```python
#% visit -cli
p = PseudocolorAttributes()
p.minFlag, p.min = 1, 5.0
p.maxFlag, p.max = 1, 20.0
SetDefaultPlotOptions(p)
# Save the new default Pseudocolor settings to the config file.
WriteConfig()
```
2.4.288 WriteScript

Example:

```python
f = open('script.py', 'wt')
WriteScript(f)
f.close()
```

2.4.289 ZonePick

Synopsis:

```
ZonePick(namedarg1=arg1, namedarg2=arg2, ...) -> dictionary
```

coord [tuple] A tuple of doubles containing the spatial coordinate (x, y, z).
x [integer] An integer containing the screen X location (in pixels) offset from the left side of the visualization window.
y [integer] An integer containing the screen Y location (in pixels) offset from the bottom of the visualization window.
vars (optional) [tuple] A tuple of strings with the variable names for which to return results. Default is the currently plotted variable.
do_time (optional) [integer] An integer indicating whether to do a time pick. 1 -> do a time pick, 0 (default) -> do not do a time pick.
start_time (optional) [integer] An integer with the starting frame index. Default is 0.
end_time (optional) [integer] An integer with the ending frame index. Default is num_timestates-1.
stride (optional) [integer] An integer with the stride for advancing in time. Default is 1.
preserve_coord (optional) [integer] An integer indicating whether to pick an element or a coordinate. 0 -> used picked element (default), 1-> used picked coordinate.
curve_plot_type (optional) [integer] An integer indicating whether the output should be on a single axis or with multiple axes. 0 -> single Y axis (default), 1 -> multiple Y Axes.
return type [dictionary] ZonePick returns a python dictionary of the pick results, unless do_time is specified, then a time curve is created in a new window. If the picked variable is node centered, the variable values are grouped according to incident node ids.

Description:

The ZonePick function prints pick information for the cell (a.k.a zone) that contains the specified point. The point can be specified as a 2D or 3D point in world space or it can be specified as a pixel location in screen space. If the point is specified as a pixel location then VisIt finds the zone that contains the intersection of a cell and a ray that is projected into the mesh. Once the zonal pick has been calculated, you can use the GetPickOutput function to retrieve the printed pick output as a string which can be used for other purposes.

Example:

```python
#% visit -cli
OpenDatabase("/usr/gapps/visit/data/noise.silo")
AddPlot("Pseudocolor", "hgslice")
DrawPlots()
# Perform zone pick in screen space
pick_out = ZonePick(x=200,y=200)
```

(continues on next page)
2.5 Attribute Reference

This chapter shows all the attributes that can be set to control the behavior of VisIt. The attributes themselves are not documented, but their names are usually quite explanatory. When a member of an attribute can take values from a given list of options, the default option is printed first in italic followed by a comma separated list of the other available options.

The listing is ordered in alphabetical ordering of the name of the attribute set. For each set the function that will provide you with these attributes is printed in italic.

Many functions return an integer where 1 means success and 0 means failure. This behavior is represented by the type CLI_return_t in an attempt to distinguish it from functions that may utilize the full range of integers. |

2.5.1 AMRStitchCell: AMRStitchCellAttributes()

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>CreateCellsOfType</td>
<td>DualGridAndStitchCells, DualGrid, StitchCells</td>
</tr>
</tbody>
</table>

2.5.2 Animation: AnimationAttributes()

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>animationMode</td>
<td>StopMode, ReversePlayMode, PlayMode</td>
</tr>
<tr>
<td>pipelineCachingMode</td>
<td>0</td>
</tr>
<tr>
<td>frameIncrement</td>
<td>1</td>
</tr>
<tr>
<td>timeout</td>
<td>1</td>
</tr>
<tr>
<td>playbackMode</td>
<td>PlayOnce, Looping, Swing</td>
</tr>
</tbody>
</table>
### 2.5.3 Annotation: AnnotateAttributes()

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>axes2D.visible</td>
<td>1</td>
</tr>
<tr>
<td>axes2D.autoSetTicks</td>
<td>1</td>
</tr>
<tr>
<td>axes2D.autoSetScaling</td>
<td>1</td>
</tr>
<tr>
<td>axes2D.lineWidth</td>
<td>0</td>
</tr>
<tr>
<td>axes2D.tickLocation</td>
<td>Outside, Inside, Both</td>
</tr>
<tr>
<td>axes2D.tickAxes</td>
<td>BottomLeft, Off, Bottom, Left, All</td>
</tr>
<tr>
<td>axes2D xAxis.title.visible</td>
<td>1</td>
</tr>
<tr>
<td>axes2D xAxis.title.font.font</td>
<td>Courier, Arial, Times</td>
</tr>
<tr>
<td>axes2D xAxis.title.font.scale</td>
<td>1</td>
</tr>
<tr>
<td>axes2D xAxis.title.font.useForegroundColor</td>
<td>1</td>
</tr>
<tr>
<td>axes2D xAxis.title.title</td>
<td>“X-Axis”</td>
</tr>
<tr>
<td>axes2D xAxis.title.units</td>
<td>“”</td>
</tr>
<tr>
<td>axes2D xAxis.label.visible</td>
<td>1</td>
</tr>
<tr>
<td>axes2D xAxis.label.font.font</td>
<td>Courier, Arial, Times</td>
</tr>
<tr>
<td>axes2D xAxis.label.font.scale</td>
<td>1</td>
</tr>
<tr>
<td>axes2D xAxis.label.font.useForegroundColor</td>
<td>1</td>
</tr>
<tr>
<td>axes2D xAxis.label.font.color</td>
<td>(0, 0, 0, 255)</td>
</tr>
<tr>
<td>axes2D xAxis.label.font.bold</td>
<td>1</td>
</tr>
<tr>
<td>axes2D xAxis.label.font.italic</td>
<td>1</td>
</tr>
<tr>
<td>axes2D xAxis.label.scaling</td>
<td>0</td>
</tr>
<tr>
<td>axes2D xAxis.tickMarks.visible</td>
<td>1</td>
</tr>
<tr>
<td>axes2D xAxis.tickMarks.majorMinimum</td>
<td>0</td>
</tr>
<tr>
<td>axes2D xAxis.tickMarks.majorMaximum</td>
<td>1</td>
</tr>
<tr>
<td>axes2D xAxis.tickMarks.minorSpacing</td>
<td>0.02</td>
</tr>
<tr>
<td>axes2D xAxis.tickMarks.majorSpacing</td>
<td>0.2</td>
</tr>
<tr>
<td>axes2D xAxis.grid</td>
<td>0</td>
</tr>
<tr>
<td>axes2D yAxis.title.visible</td>
<td>1</td>
</tr>
<tr>
<td>axes2D yAxis.title.font.font</td>
<td>Courier, Arial, Times</td>
</tr>
<tr>
<td>axes2D yAxis.title.font.scale</td>
<td>1</td>
</tr>
<tr>
<td>axes2D yAxis.title.font.useForegroundColor</td>
<td>1</td>
</tr>
<tr>
<td>axes2D yAxis.title.font.color</td>
<td>(0, 0, 0, 255)</td>
</tr>
<tr>
<td>axes2D yAxis.title.font.bold</td>
<td>1</td>
</tr>
<tr>
<td>axes2D yAxis.title.font.italic</td>
<td>1</td>
</tr>
<tr>
<td>axes2D yAxis.title.userTitle</td>
<td>0</td>
</tr>
<tr>
<td>axes2D yAxis.title.userUnits</td>
<td>0</td>
</tr>
<tr>
<td>axes2D yAxis.title.title</td>
<td>“Y-Axis”</td>
</tr>
<tr>
<td>axes2D yAxis.title.units</td>
<td>“”</td>
</tr>
</tbody>
</table>

Continued on next page
<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>axes2D.yAxis.label.visible</td>
<td>1</td>
</tr>
<tr>
<td>axes2D.yAxis.label.font.font</td>
<td>Courier, Arial, Times</td>
</tr>
<tr>
<td>axes2D.yAxis.label.font.scale</td>
<td>1</td>
</tr>
<tr>
<td>axes2D.yAxis.label.font.useForegroundColor</td>
<td>1</td>
</tr>
<tr>
<td>axes2D.yAxis.label.font.color</td>
<td>(0, 0, 0, 255)</td>
</tr>
<tr>
<td>axes2D.yAxis.label.font.bold</td>
<td>1</td>
</tr>
<tr>
<td>axes2D.yAxis.label.font.italic</td>
<td>1</td>
</tr>
<tr>
<td>axes2D.yAxis.label.scaling</td>
<td>0</td>
</tr>
<tr>
<td>axes2D.yAxis.tickMarks.visible</td>
<td>1</td>
</tr>
<tr>
<td>axes2D.yAxis.tickMarks.majorMinimum</td>
<td>0</td>
</tr>
<tr>
<td>axes2D.yAxis.tickMarks.majorMaximum</td>
<td>1</td>
</tr>
<tr>
<td>axes2D.yAxis.tickMarks.minorSpacing</td>
<td>0.02</td>
</tr>
<tr>
<td>axes2D.yAxis.tickMarks.majorSpacing</td>
<td>0.2</td>
</tr>
<tr>
<td>axes2D.yAxis.grid</td>
<td>0</td>
</tr>
<tr>
<td>axes3D.visible</td>
<td>1</td>
</tr>
<tr>
<td>axes3D.autoSetTicks</td>
<td>1</td>
</tr>
<tr>
<td>axes3D.autoSetScaling</td>
<td>1</td>
</tr>
<tr>
<td>axes3D.lineWidth</td>
<td>0</td>
</tr>
<tr>
<td>axes3D.tickLocation</td>
<td>Inside, Outside, Both</td>
</tr>
<tr>
<td>axes3D.axesType</td>
<td>ClosestTriad, FurthestTriad, OutsideEdges, StaticTriad, StaticEdges</td>
</tr>
<tr>
<td>axes3D.triadFlag</td>
<td>1</td>
</tr>
<tr>
<td>axes3D.bboxFlag</td>
<td>1</td>
</tr>
<tr>
<td>axes3D.xAxis.title.visible</td>
<td>1</td>
</tr>
<tr>
<td>axes3D.xAxis.title.font.font</td>
<td>Arial, Courier, Times</td>
</tr>
<tr>
<td>axes3D.xAxis.title.font.scale</td>
<td>1</td>
</tr>
<tr>
<td>axes3D.xAxis.title.font.useForegroundColor</td>
<td>1</td>
</tr>
<tr>
<td>axes3D.xAxis.title.font.color</td>
<td>(0, 0, 0, 255)</td>
</tr>
<tr>
<td>axes3D.xAxis.title.font.bold</td>
<td>0</td>
</tr>
<tr>
<td>axes3D.xAxis.title.font.italic</td>
<td>0</td>
</tr>
<tr>
<td>axes3D.xAxis.title.userTitle</td>
<td>0</td>
</tr>
<tr>
<td>axes3D.xAxis.title.userUnits</td>
<td>0</td>
</tr>
<tr>
<td>axes3D.xAxis.title.title</td>
<td>&quot;X-Axis&quot;</td>
</tr>
<tr>
<td>axes3D.xAxis.title.units</td>
<td>&quot;&quot;</td>
</tr>
<tr>
<td>axes3D.xAxis.label.visible</td>
<td>1</td>
</tr>
<tr>
<td>axes3D.xAxis.label.font.font</td>
<td>Arial, Courier, Times</td>
</tr>
<tr>
<td>axes3D.xAxis.label.font.scale</td>
<td>1</td>
</tr>
<tr>
<td>axes3D.xAxis.label.font.useForegroundColor</td>
<td>1</td>
</tr>
<tr>
<td>axes3D.xAxis.label.font.color</td>
<td>(0, 0, 0, 255)</td>
</tr>
<tr>
<td>axes3D.xAxis.label.font.bold</td>
<td>0</td>
</tr>
<tr>
<td>axes3D.xAxis.label.font.italic</td>
<td>0</td>
</tr>
<tr>
<td>axes3D.xAxis.label.scaling</td>
<td>0</td>
</tr>
<tr>
<td>axes3D.xAxis.tickMarks.visible</td>
<td>1</td>
</tr>
<tr>
<td>axes3D.xAxis.tickMarks.majorMinimum</td>
<td>0</td>
</tr>
<tr>
<td>axes3D.xAxis.tickMarks.majorMaximum</td>
<td>1</td>
</tr>
<tr>
<td>axes3D.xAxis.tickMarks.minorSpacing</td>
<td>0.02</td>
</tr>
<tr>
<td>axes3D.xAxis.tickMarks.majorSpacing</td>
<td>0.2</td>
</tr>
<tr>
<td>axes3D.xAxis.grid</td>
<td>0</td>
</tr>
<tr>
<td>axes3D.yAxis.title.visible</td>
<td>1</td>
</tr>
<tr>
<td>axes3D.yAxis.title.font.font</td>
<td>Arial, Courier, Times</td>
</tr>
<tr>
<td>axes3D.yAxis.title.font.scale</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td>------------------</td>
<td>---------</td>
</tr>
<tr>
<td>axes3D.yAxis.title.font.useForegroundColor</td>
<td>1</td>
</tr>
<tr>
<td>axes3D.yAxis.title.userTitle</td>
<td>0</td>
</tr>
<tr>
<td>axes3D.yAxis.label.visible</td>
<td>1</td>
</tr>
<tr>
<td>axes3D.yAxis.label.font.color</td>
<td>(0, 0, 0, 255)</td>
</tr>
<tr>
<td>axes3D.yAxis.tickMarks.visible</td>
<td>1</td>
</tr>
<tr>
<td>axes3D.yAxis.tickMarks.majorSpacing</td>
<td>0.2</td>
</tr>
<tr>
<td>axes3D.zAxis.title.font.scale</td>
<td>1</td>
</tr>
<tr>
<td>axes3D.zAxis.title.font.italic</td>
<td>0</td>
</tr>
<tr>
<td>axes3D.zAxis.title.units</td>
<td>“”</td>
</tr>
<tr>
<td>axes3D.zAxis.label.font.useForegroundColor</td>
<td>1</td>
</tr>
<tr>
<td>axes3D.zAxis.label.scaling</td>
<td>0</td>
</tr>
<tr>
<td>axes3D.zAxis.tickMarks.minorSpacing</td>
<td>0.02</td>
</tr>
<tr>
<td>axes3D.bboxLocation</td>
<td>(0, 1, 0, 1, 0, 1)</td>
</tr>
</tbody>
</table>
Table 2.1 – continued from previous page

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>axes3D.triadLineWidth</td>
<td>1</td>
</tr>
<tr>
<td>axes3D.triadFont</td>
<td>0</td>
</tr>
<tr>
<td>axes3D.triadBold</td>
<td>1</td>
</tr>
<tr>
<td>axes3D.triadItalic</td>
<td>1</td>
</tr>
<tr>
<td>axes3D.triadSetManually</td>
<td>0</td>
</tr>
<tr>
<td>userInfoFlag</td>
<td>1</td>
</tr>
<tr>
<td>userInfoFont.font</td>
<td>Arial, Courier, Times</td>
</tr>
<tr>
<td>userInfoFont.scale</td>
<td>1</td>
</tr>
<tr>
<td>userInfoFont.useForegroundColor</td>
<td>1</td>
</tr>
<tr>
<td>userInfoFont.color</td>
<td>(0, 0, 0, 255)</td>
</tr>
<tr>
<td>userInfoFont.bold</td>
<td>0</td>
</tr>
<tr>
<td>userInfoFont.italic</td>
<td>0</td>
</tr>
<tr>
<td>databaseInfoFlag</td>
<td>1</td>
</tr>
<tr>
<td>databaseInfoFont.font</td>
<td>Arial, Courier, Times</td>
</tr>
<tr>
<td>databaseInfoFont.scale</td>
<td>1</td>
</tr>
<tr>
<td>databaseInfoFont.useForegroundColor</td>
<td>1</td>
</tr>
<tr>
<td>databaseInfoFont.color</td>
<td>(0, 0, 0, 255)</td>
</tr>
<tr>
<td>databaseInfoFont.bold</td>
<td>0</td>
</tr>
<tr>
<td>databaseInfoFont.italic</td>
<td>0</td>
</tr>
<tr>
<td>databaseInfoExpansionMode</td>
<td>File, Directory, Full, Smart, SmartDirectory</td>
</tr>
<tr>
<td>databaseInfoTimeScale</td>
<td>1</td>
</tr>
<tr>
<td>databaseInfoTimeOffset</td>
<td>0</td>
</tr>
<tr>
<td>legendInfoFlag</td>
<td>1</td>
</tr>
<tr>
<td>backgroundColor</td>
<td>(255, 255, 255, 255)</td>
</tr>
<tr>
<td>foregroundColor</td>
<td>(0, 0, 0, 255)</td>
</tr>
<tr>
<td>gradientBackgroundStyle</td>
<td>Radial, TopToBottom, BottomToTop, LeftToRight, RightToLeft</td>
</tr>
<tr>
<td>gradientColor1</td>
<td>(0, 0, 255, 255)</td>
</tr>
<tr>
<td>gradientColor2</td>
<td>(0, 0, 255, 255)</td>
</tr>
<tr>
<td>backgroundMode</td>
<td>Solid, Gradient, Image, ImageSphere</td>
</tr>
<tr>
<td>backgroundImage</td>
<td></td>
</tr>
<tr>
<td>imageRepeatX</td>
<td>1</td>
</tr>
<tr>
<td>imageRepeatY</td>
<td>1</td>
</tr>
<tr>
<td>axesArray.visible</td>
<td>1</td>
</tr>
<tr>
<td>axesArray.ticksVisible</td>
<td>1</td>
</tr>
<tr>
<td>axesArray.autoSetTicks</td>
<td>1</td>
</tr>
<tr>
<td>axesArray.autoSetScaling</td>
<td>1</td>
</tr>
<tr>
<td>axesArray.lineWidth</td>
<td>0</td>
</tr>
<tr>
<td>axesArray.axes.title.visible</td>
<td>1</td>
</tr>
<tr>
<td>axesArray.axes.title.font.font</td>
<td>Arial, Courier, Times</td>
</tr>
<tr>
<td>axesArray.axes.title.font.scale</td>
<td>1</td>
</tr>
<tr>
<td>axesArray.axes.title.font.useForegroundColor</td>
<td>1</td>
</tr>
<tr>
<td>axesArray.axes.title.font.color</td>
<td>(0, 0, 0, 255)</td>
</tr>
<tr>
<td>axesArray.axes.title.font.bold</td>
<td>0</td>
</tr>
<tr>
<td>axesArray.axes.title.font.italic</td>
<td>0</td>
</tr>
<tr>
<td>axesArray.axes.title.userTitle</td>
<td>0</td>
</tr>
<tr>
<td>axesArray.axes.title.userUnits</td>
<td>0</td>
</tr>
<tr>
<td>axesArray.axes.title.title</td>
<td></td>
</tr>
<tr>
<td>axesArray.axes.title.units</td>
<td></td>
</tr>
<tr>
<td>axesArray.axes.label.visible</td>
<td>1</td>
</tr>
</tbody>
</table>

Continued on next page
### Table 2.1 – continued from previous page

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td><code>axesArray.axes.label.font.font</code></td>
<td>Arial, Courier, Times</td>
</tr>
<tr>
<td><code>axesArray.axes.label.font.scale</code></td>
<td>1</td>
</tr>
<tr>
<td><code>axesArray.axes.label.font.useForegroundColor</code></td>
<td>1</td>
</tr>
<tr>
<td><code>axesArray.axes.label.font.color</code></td>
<td>(0, 0, 255)</td>
</tr>
<tr>
<td><code>axesArray.axes.label.font.bold</code></td>
<td>0</td>
</tr>
<tr>
<td><code>axesArray.axes.label.font.italic</code></td>
<td>0</td>
</tr>
<tr>
<td><code>axesArray.axes.label.scaling</code></td>
<td>0</td>
</tr>
<tr>
<td><code>axesArray.axes.tickMarks.visible</code></td>
<td>1</td>
</tr>
<tr>
<td><code>axesArray.axes.tickMarks.majorMinimum</code></td>
<td>0</td>
</tr>
<tr>
<td><code>axesArray.axes.tickMarks.majorMaximum</code></td>
<td>1</td>
</tr>
<tr>
<td><code>axesArray.axes.tickMarks.minorSpacing</code></td>
<td>0.02</td>
</tr>
<tr>
<td><code>axesArray.axes.tickMarks.majorSpacing</code></td>
<td>0.2</td>
</tr>
<tr>
<td><code>axesArray.axes.grid</code></td>
<td>0</td>
</tr>
</tbody>
</table>

2.5.4 Axis:  

AxisAttributes()
<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>title.visible</td>
<td>1</td>
</tr>
<tr>
<td>title.font.font</td>
<td>Arial, Courier, Times</td>
</tr>
<tr>
<td>title.font.scale</td>
<td>1</td>
</tr>
<tr>
<td>title.font.useForegroundColor</td>
<td>1</td>
</tr>
<tr>
<td>title.font.color</td>
<td>(0, 0, 0, 255)</td>
</tr>
<tr>
<td>title.font.bold</td>
<td>0</td>
</tr>
<tr>
<td>title.font.italic</td>
<td>0</td>
</tr>
<tr>
<td>title.userTitle</td>
<td>0</td>
</tr>
<tr>
<td>title.userUnits</td>
<td>0</td>
</tr>
<tr>
<td>title.title</td>
<td>&quot;&quot;</td>
</tr>
<tr>
<td>title.units</td>
<td>&quot;&quot;</td>
</tr>
<tr>
<td>label.visible</td>
<td>1</td>
</tr>
<tr>
<td>label.font.font</td>
<td>Arial, Courier, Times</td>
</tr>
<tr>
<td>label.font.scale</td>
<td>1</td>
</tr>
<tr>
<td>label.font.useForegroundColor</td>
<td>1</td>
</tr>
<tr>
<td>label.font.color</td>
<td>(0, 0, 0, 255)</td>
</tr>
<tr>
<td>label.font.bold</td>
<td>0</td>
</tr>
<tr>
<td>label.font.italic</td>
<td>0</td>
</tr>
<tr>
<td>label.scaling</td>
<td>0</td>
</tr>
<tr>
<td>tickMarks.visible</td>
<td>1</td>
</tr>
<tr>
<td>tickMarks.majorMinimum</td>
<td>0</td>
</tr>
<tr>
<td>tickMarks.majorMaximum</td>
<td>1</td>
</tr>
<tr>
<td>tickMarks.minorSpacing</td>
<td>0.02</td>
</tr>
<tr>
<td>tickMarks.majorSpacing</td>
<td>0.2</td>
</tr>
<tr>
<td>grid</td>
<td>0</td>
</tr>
</tbody>
</table>

2.5.5 AxisAlignedSlice4D:  

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>I</td>
<td>()</td>
</tr>
<tr>
<td>J</td>
<td>()</td>
</tr>
<tr>
<td>K</td>
<td>()</td>
</tr>
<tr>
<td>L</td>
<td>()</td>
</tr>
</tbody>
</table>
2.5.6 Boundary:

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>colorType</td>
<td>ColorByMultipleColors, ColorBySingleColor, ColorByColorTable</td>
</tr>
<tr>
<td>colorTableName</td>
<td>“Default”</td>
</tr>
<tr>
<td>invertColorTable</td>
<td>0</td>
</tr>
<tr>
<td>legendFlag</td>
<td>1</td>
</tr>
<tr>
<td>lineWidth</td>
<td>0</td>
</tr>
<tr>
<td>singleColor</td>
<td>(0, 0, 0, 255)</td>
</tr>
<tr>
<td>boundaryNames</td>
<td>()</td>
</tr>
<tr>
<td>opacity</td>
<td>1</td>
</tr>
<tr>
<td>wireframe</td>
<td>0</td>
</tr>
<tr>
<td>smoothingLevel</td>
<td>0</td>
</tr>
</tbody>
</table>

2.5.7 BoundaryOp:

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>smoothingLevel</td>
<td>0</td>
</tr>
</tbody>
</table>

2.5.8 Box:

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>smoothingLevel</td>
<td>0</td>
</tr>
<tr>
<td>Attribute</td>
<td>Default/Allowed Values</td>
</tr>
<tr>
<td>-----------</td>
<td>-----------------------</td>
</tr>
<tr>
<td>amount</td>
<td>Some, All</td>
</tr>
<tr>
<td>minx</td>
<td>0</td>
</tr>
<tr>
<td>maxx</td>
<td>1</td>
</tr>
<tr>
<td>miny</td>
<td>0</td>
</tr>
<tr>
<td>maxy</td>
<td>1</td>
</tr>
<tr>
<td>minz</td>
<td>0</td>
</tr>
<tr>
<td>maxz</td>
<td>1</td>
</tr>
<tr>
<td>inverse</td>
<td>0</td>
</tr>
</tbody>
</table>

2.5.9 CartographicProjection: `CartographicProjectionAttributes()`

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>projectionID</td>
<td>aitoff, eck4, eqdc, hammer, laea, lcc, merc, mill, moll, ortho, wink2</td>
</tr>
<tr>
<td>centralMeridian</td>
<td>0</td>
</tr>
</tbody>
</table>

2.5.10 Clip: `ClipAttributes()`
### Attribute Reference

#### 2.5.11 Cone:  

[ConeAttributes()](#)

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>angle</td>
<td>45</td>
</tr>
<tr>
<td>origin</td>
<td>(0, 0, 0)</td>
</tr>
<tr>
<td>normal</td>
<td>(0, 0, 1)</td>
</tr>
<tr>
<td>representation</td>
<td>Flattened, ThreeD, R_Theta</td>
</tr>
<tr>
<td>upAxis</td>
<td>(0, 1, 0)</td>
</tr>
<tr>
<td>cutByLength</td>
<td>0</td>
</tr>
<tr>
<td>length</td>
<td>1</td>
</tr>
</tbody>
</table>

#### 2.5.12 ConnectedComponents:  

[ConnectedComponentsAttributes()](#)

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>plane1Status</td>
<td>1</td>
</tr>
<tr>
<td>plane2Status</td>
<td>0</td>
</tr>
<tr>
<td>plane3Status</td>
<td>0</td>
</tr>
<tr>
<td>plane1Origin</td>
<td>(0, 0, 0)</td>
</tr>
<tr>
<td>plane2Origin</td>
<td>(0, 0, 0)</td>
</tr>
<tr>
<td>plane3Origin</td>
<td>(0, 0, 0)</td>
</tr>
<tr>
<td>plane1Normal</td>
<td>(1, 0, 0)</td>
</tr>
<tr>
<td>plane2Normal</td>
<td>(0, 1, 0)</td>
</tr>
<tr>
<td>plane3Normal</td>
<td>(0, 0, 1)</td>
</tr>
<tr>
<td>planeInverse</td>
<td>0</td>
</tr>
<tr>
<td>planeToolControlledClipPlane</td>
<td>Plane1, None, Plane2, Plane3</td>
</tr>
<tr>
<td>center</td>
<td>(0, 0, 0)</td>
</tr>
<tr>
<td>radius</td>
<td>1</td>
</tr>
<tr>
<td>sphereInverse</td>
<td>0</td>
</tr>
</tbody>
</table>
2.5.13 ConstructDataBinning:

```
<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>name</td>
<td>&quot;&quot;</td>
</tr>
<tr>
<td>varnames</td>
<td>()</td>
</tr>
<tr>
<td>binType</td>
<td>()</td>
</tr>
<tr>
<td>binBoundaries</td>
<td>()</td>
</tr>
<tr>
<td>reductionOperator</td>
<td>Average, Minimum, Maximum, StandardDeviation, Variance, Sum, Count, RMS, PDF</td>
</tr>
<tr>
<td>varForReductionOperator</td>
<td>&quot;&quot;</td>
</tr>
<tr>
<td>undefinedValue</td>
<td>0</td>
</tr>
<tr>
<td>binningScheme</td>
<td>Uniform, Unknown</td>
</tr>
<tr>
<td>numBins</td>
<td>()</td>
</tr>
<tr>
<td>overTime</td>
<td>0</td>
</tr>
<tr>
<td>timeStart</td>
<td>0</td>
</tr>
<tr>
<td>timeEnd</td>
<td>1</td>
</tr>
<tr>
<td>timeStride</td>
<td>1</td>
</tr>
<tr>
<td>outOfBoundsBehavior</td>
<td>Clamp, Discard</td>
</tr>
</tbody>
</table>
```

2.5.14 Contour:

```
<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>defaultPalette.GetControlPoints(0).colors</td>
<td>(255, 0, 0, 255)</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(0).position</td>
<td>0</td>
</tr>
</tbody>
</table>
```

Continued on next page
Table 2.2 – continued from previous page

<table>
<thead>
<tr>
<th>index</th>
<th>colors</th>
<th>position</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>(0, 255, 0, 255)</td>
<td>0.034</td>
</tr>
<tr>
<td>2</td>
<td>(0, 0, 255, 255)</td>
<td>0.069</td>
</tr>
<tr>
<td>3</td>
<td>(0, 255, 255, 255)</td>
<td>0.103</td>
</tr>
<tr>
<td>4</td>
<td>(255, 0, 255, 255)</td>
<td>0.138</td>
</tr>
<tr>
<td>5</td>
<td>(255, 255, 0, 255)</td>
<td>0.172</td>
</tr>
<tr>
<td>6</td>
<td>(255, 135, 0, 255)</td>
<td>0.207</td>
</tr>
<tr>
<td>7</td>
<td>(255, 0, 135, 255)</td>
<td>0.241</td>
</tr>
<tr>
<td>8</td>
<td>(168, 168, 168, 255)</td>
<td>0.276</td>
</tr>
<tr>
<td>9</td>
<td>(255, 68, 68, 255)</td>
<td>0.31</td>
</tr>
<tr>
<td>10</td>
<td>(99, 255, 99, 255)</td>
<td>0.345</td>
</tr>
<tr>
<td>11</td>
<td>(99, 99, 255, 255)</td>
<td>0.379</td>
</tr>
<tr>
<td>12</td>
<td>(40, 165, 165, 255)</td>
<td>0.414</td>
</tr>
<tr>
<td>13</td>
<td>(255, 99, 255, 255)</td>
<td>0.448</td>
</tr>
<tr>
<td>14</td>
<td>(99, 99, 255, 255)</td>
<td>0.483</td>
</tr>
<tr>
<td>15</td>
<td>(40, 165, 165, 255)</td>
<td>0.517</td>
</tr>
<tr>
<td>16</td>
<td>(170, 79, 255, 255)</td>
<td>0.552</td>
</tr>
<tr>
<td>17</td>
<td>(150, 0, 0, 255)</td>
<td>0.586</td>
</tr>
<tr>
<td>18</td>
<td>(0, 150, 0, 255)</td>
<td>0.621</td>
</tr>
<tr>
<td>19</td>
<td>(0, 0, 150, 255)</td>
<td>0.655</td>
</tr>
<tr>
<td>20</td>
<td>(0, 109, 109, 255)</td>
<td>0.69</td>
</tr>
<tr>
<td>21</td>
<td>(150, 0, 150, 255)</td>
<td>0.724</td>
</tr>
<tr>
<td>22</td>
<td>(150, 0, 150, 255)</td>
<td>0.759</td>
</tr>
<tr>
<td>23</td>
<td>(150, 84, 0, 255)</td>
<td>0.793</td>
</tr>
<tr>
<td>24</td>
<td>(160, 0, 79, 255)</td>
<td>0.828</td>
</tr>
<tr>
<td>25</td>
<td>(255, 104, 28, 255)</td>
<td>0.862</td>
</tr>
</tbody>
</table>

Continued on next page
Table 2.2 – continued from previous page

<table>
<thead>
<tr>
<th>Method</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>defaultPalette.GetControlPoints(26).colors</td>
<td>(0, 170, 81, 255)</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(26).position</td>
<td>0.897</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(27).colors</td>
<td>(68, 255, 124, 255)</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(27).position</td>
<td>0.931</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(28).colors</td>
<td>(0, 130, 255, 255)</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(28).position</td>
<td>0.966</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(29).colors</td>
<td>(130, 0, 255, 255)</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(29).position</td>
<td>1</td>
</tr>
<tr>
<td>defaultPalette.smoothing</td>
<td>None, Linear, CubicSpline</td>
</tr>
<tr>
<td>defaultPalette.equalSpacingFlag</td>
<td>1</td>
</tr>
<tr>
<td>defaultPalette.discreteFlag</td>
<td>1</td>
</tr>
<tr>
<td>defaultPalette.categoryName</td>
<td>“Standard”</td>
</tr>
<tr>
<td>changedColors</td>
<td>()</td>
</tr>
<tr>
<td>colorType</td>
<td><strong>ColorByMultipleColors, ColorBySingleColor, ColorByColorTable</strong></td>
</tr>
<tr>
<td>colorTableName</td>
<td>“Default”</td>
</tr>
<tr>
<td>invertColorTable</td>
<td>0</td>
</tr>
<tr>
<td>legendFlag</td>
<td>1</td>
</tr>
<tr>
<td>lineWidth</td>
<td>0</td>
</tr>
<tr>
<td>singleColor</td>
<td>(255, 0, 0, 255)</td>
</tr>
<tr>
<td></td>
<td>SetMultiColor(0, (255, 0, 0, 255))</td>
</tr>
<tr>
<td></td>
<td>SetMultiColor(1, (0, 255, 0, 255))</td>
</tr>
<tr>
<td></td>
<td>SetMultiColor(2, (0, 0, 255, 255))</td>
</tr>
<tr>
<td></td>
<td>SetMultiColor(3, (0, 255, 255, 255))</td>
</tr>
<tr>
<td></td>
<td>SetMultiColor(4, (255, 0, 255, 255))</td>
</tr>
<tr>
<td></td>
<td>SetMultiColor(5, (255, 255, 0, 255))</td>
</tr>
<tr>
<td></td>
<td>SetMultiColor(6, (255, 135, 0, 255))</td>
</tr>
<tr>
<td></td>
<td>SetMultiColor(7, (255, 0, 135, 255))</td>
</tr>
<tr>
<td></td>
<td>SetMultiColor(8, (168, 168, 168, 255))</td>
</tr>
<tr>
<td></td>
<td>SetMultiColor(9, (255, 68, 68, 255))</td>
</tr>
<tr>
<td>contourNLevels</td>
<td>10</td>
</tr>
<tr>
<td>contourValue</td>
<td>()</td>
</tr>
<tr>
<td>contourPercent</td>
<td>()</td>
</tr>
<tr>
<td>contourMethod</td>
<td><strong>Level, Value, Percent</strong></td>
</tr>
<tr>
<td>minFlag</td>
<td>0</td>
</tr>
<tr>
<td>maxFlag</td>
<td>0</td>
</tr>
<tr>
<td>min</td>
<td>0</td>
</tr>
<tr>
<td>max</td>
<td>1</td>
</tr>
<tr>
<td>scaling</td>
<td><strong>Linear, Log</strong></td>
</tr>
<tr>
<td>wireframe</td>
<td>0</td>
</tr>
</tbody>
</table>
2.5.15 CoordSwap:  
*CoordSwapAttributes()*

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>newCoord1</td>
<td>Coord1, Coord2, Coord3</td>
</tr>
<tr>
<td>newCoord2</td>
<td>Coord2, Coord1, Coord3</td>
</tr>
<tr>
<td>newCoord3</td>
<td>Coord3, Coord1, Coord2</td>
</tr>
</tbody>
</table>

2.5.16 CreateBonds:  
*CreateBondsAttributes()*

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>elementVariable</td>
<td>“element”</td>
</tr>
<tr>
<td>atomicNumber1</td>
<td>(1, -1)</td>
</tr>
<tr>
<td>atomicNumber2</td>
<td>(-1, -1)</td>
</tr>
<tr>
<td>minDist</td>
<td>(0.4, 0.4)</td>
</tr>
<tr>
<td>maxDist</td>
<td>(1.2, 1.9)</td>
</tr>
<tr>
<td>maxBondsClamp</td>
<td>10</td>
</tr>
<tr>
<td>addPeriodicBonds</td>
<td>0</td>
</tr>
<tr>
<td>useUnitCellVectors</td>
<td>1</td>
</tr>
<tr>
<td>periodicInX</td>
<td>1</td>
</tr>
<tr>
<td>periodicInY</td>
<td>1</td>
</tr>
<tr>
<td>periodicInZ</td>
<td>1</td>
</tr>
<tr>
<td>xVector</td>
<td>(1, 0, 0)</td>
</tr>
<tr>
<td>yVector</td>
<td>(0, 1, 0)</td>
</tr>
<tr>
<td>zVector</td>
<td>(0, 0, 1)</td>
</tr>
</tbody>
</table>

2.5.17 Curve:  
*CurveAttributes()*

2.5. Attribute Reference 605
<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>showLines</td>
<td>1</td>
</tr>
<tr>
<td>lineWidth</td>
<td>0</td>
</tr>
<tr>
<td>showPoints</td>
<td>0</td>
</tr>
<tr>
<td>symbol</td>
<td>Point, TriangleUp, TriangleDown, Square, Circle, Plus, X</td>
</tr>
<tr>
<td>pointSize</td>
<td>5</td>
</tr>
<tr>
<td>pointFillMode</td>
<td>Static, Dynamic</td>
</tr>
<tr>
<td>pointStride</td>
<td>1</td>
</tr>
<tr>
<td>symbolDensity</td>
<td>50</td>
</tr>
<tr>
<td>curveColorSource</td>
<td>Cycle, Custom</td>
</tr>
<tr>
<td>curveColor</td>
<td>(0, 0, 0, 255)</td>
</tr>
<tr>
<td>showLegend</td>
<td>1</td>
</tr>
<tr>
<td>showLabels</td>
<td>1</td>
</tr>
<tr>
<td>designator</td>
<td>...</td>
</tr>
<tr>
<td>doBallTimeCue</td>
<td>0</td>
</tr>
<tr>
<td>ballTimeCueColor</td>
<td>(0, 0, 0, 255)</td>
</tr>
<tr>
<td>timeCueBallSize</td>
<td>0.01</td>
</tr>
<tr>
<td>doLineTimeCue</td>
<td>0</td>
</tr>
<tr>
<td>lineTimeCueColor</td>
<td>(0, 0, 0, 255)</td>
</tr>
<tr>
<td>lineTimeCueWidth</td>
<td>0</td>
</tr>
<tr>
<td>doCropTimeCue</td>
<td>0</td>
</tr>
<tr>
<td>timeForTimeCue</td>
<td>0</td>
</tr>
<tr>
<td>fillMode</td>
<td>NoFill, Solid, HorizontalGradient, VerticalGradient</td>
</tr>
<tr>
<td>fillColor1</td>
<td>(255, 0, 0, 255)</td>
</tr>
<tr>
<td>fillColor2</td>
<td>(255, 100, 100, 255)</td>
</tr>
<tr>
<td>polarToCartesian</td>
<td>0</td>
</tr>
<tr>
<td>polarCoordinateOrder</td>
<td>R_Theta, Theta_R</td>
</tr>
<tr>
<td>angleUnits</td>
<td>Radians, Degrees</td>
</tr>
</tbody>
</table>

2.5.18 Cylinder: `CylinderAttributes()`

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>point1</td>
<td>(0, 0, 0)</td>
</tr>
<tr>
<td>point2</td>
<td>(1, 0, 0)</td>
</tr>
<tr>
<td>radius</td>
<td>1</td>
</tr>
<tr>
<td>inverse</td>
<td>0</td>
</tr>
</tbody>
</table>
2.5.19 DataBinning:  

DataBinningAttributes()

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>numDimensions</td>
<td>One, Two, Three</td>
</tr>
<tr>
<td>dim1BinBasedOn</td>
<td>Variable, X, Y, Z</td>
</tr>
<tr>
<td>dim1Var</td>
<td>“default”</td>
</tr>
<tr>
<td>dim1SpecifyRange</td>
<td>0</td>
</tr>
<tr>
<td>dim1MinRange</td>
<td>0</td>
</tr>
<tr>
<td>dim1MaxRange</td>
<td>1</td>
</tr>
<tr>
<td>dim1NumBins</td>
<td>50</td>
</tr>
<tr>
<td>dim2BinBasedOn</td>
<td>Variable, X, Y, Z</td>
</tr>
<tr>
<td>dim2Var</td>
<td>“default”</td>
</tr>
<tr>
<td>dim2SpecifyRange</td>
<td>0</td>
</tr>
<tr>
<td>dim2MinRange</td>
<td>0</td>
</tr>
<tr>
<td>dim2MaxRange</td>
<td>1</td>
</tr>
<tr>
<td>dim2NumBins</td>
<td>50</td>
</tr>
<tr>
<td>dim3BinBasedOn</td>
<td>Variable, X, Y, Z</td>
</tr>
<tr>
<td>dim3Var</td>
<td>“default”</td>
</tr>
<tr>
<td>dim3SpecifyRange</td>
<td>0</td>
</tr>
<tr>
<td>dim3MinRange</td>
<td>0</td>
</tr>
<tr>
<td>dim3MaxRange</td>
<td>1</td>
</tr>
<tr>
<td>dim3NumBins</td>
<td>50</td>
</tr>
<tr>
<td>outOfBoundsBehavior</td>
<td>Clamp, Discard</td>
</tr>
<tr>
<td>reductionOperator</td>
<td>Average, Minimum, Maximum, StandardDeviation, Variance, Sum, Count, RMS, PDF</td>
</tr>
<tr>
<td>varForReduction</td>
<td>“default”</td>
</tr>
<tr>
<td>emptyVal</td>
<td>0</td>
</tr>
<tr>
<td>outputType</td>
<td>OutputOnBins, OutputOnInputMesh</td>
</tr>
<tr>
<td>removeEmptyValFromCurve</td>
<td>1</td>
</tr>
</tbody>
</table>

2.5.20 DeferExpression:  

DeferExpressionAttributes()

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>exprs</td>
<td>()</td>
</tr>
</tbody>
</table>

2.5. Attribute Reference
2.5.21 Displace: \[\text{DisplaceAttributes()}\]

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>factor</td>
<td>1</td>
</tr>
<tr>
<td>variable</td>
<td>“default”</td>
</tr>
</tbody>
</table>

2.5.22 DualMesh: \[\text{DualMeshAttributes()}\]

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>mode</td>
<td>Auto, NodesToZones, ZonesToNodes</td>
</tr>
</tbody>
</table>

2.5.23 Edge: \[\text{EdgeAttributes()}\]

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>dummy</td>
<td>1</td>
</tr>
</tbody>
</table>
2.5.24 Elevate:  

```
Attribute          Default/Allowed Values
useXYLimits        Auto, Never, Always
limitsMode         OriginalData, CurrentPlot
scaling            Linear, Log, Skew
skewFactor         1
minFlag            0
min                0
maxFlag            0
max                1
zeroFlag           0
variable           “default”
```

2.5.25 EllipsoidSlice:  

```
Attribute          Default/Allowed Values
origin             (0, 0, 0)
radii              (1, 1, 1)
rotationAngle      (0, 0, 0)
```

2.5.26 Explode:  

```
ExplodeAttributes()
```
### 2.5.27 ExportDB: ExportDBAttributes()

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>allTimes</td>
<td>0</td>
</tr>
<tr>
<td>dirname</td>
<td>..&quot;</td>
</tr>
<tr>
<td>filename</td>
<td>&quot;visit_ex_db&quot;</td>
</tr>
<tr>
<td>timeStateFormat</td>
<td>&quot;_%04d&quot;</td>
</tr>
<tr>
<td>db_type</td>
<td>..&quot;</td>
</tr>
<tr>
<td>db_type_fullname</td>
<td>..&quot;</td>
</tr>
<tr>
<td>variables</td>
<td>()</td>
</tr>
<tr>
<td>writeUsingGroups</td>
<td>0</td>
</tr>
<tr>
<td>groupSize</td>
<td>48</td>
</tr>
<tr>
<td>opts.types</td>
<td>()</td>
</tr>
<tr>
<td>opts.help</td>
<td>..&quot;</td>
</tr>
</tbody>
</table>

Explosions does not contain any ExplodeAttributes objects.
### 2.5.28 ExternalSurface:

**ExternalSurfaceAttributes()**

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>removeGhosts</td>
<td>0</td>
</tr>
<tr>
<td>edgesIn2D</td>
<td>1</td>
</tr>
</tbody>
</table>

### 2.5.29 Extrude:

**ExtrudeAttributes()**

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>axis</td>
<td>(0, 0, 1)</td>
</tr>
<tr>
<td>byVariable</td>
<td>0</td>
</tr>
<tr>
<td>variable</td>
<td>“default”</td>
</tr>
<tr>
<td>length</td>
<td>1</td>
</tr>
<tr>
<td>steps</td>
<td>1</td>
</tr>
<tr>
<td>preserveOriginalCellNumbers</td>
<td>1</td>
</tr>
</tbody>
</table>

### 2.5.30 FFT:

**FFTAttributes()**

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>dummy</td>
<td>0</td>
</tr>
</tbody>
</table>

---

2.5. Attribute Reference 611
### 2.5.31 FilledBoundary:

#### FilledBoundaryAttributes()

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>colorType</td>
<td>ColorByMultipleColors, ColorBySingleColor, ColorByColorTable</td>
</tr>
<tr>
<td>colorTableName</td>
<td>“Default”</td>
</tr>
<tr>
<td>invertColorTable</td>
<td>0</td>
</tr>
<tr>
<td>legendFlag</td>
<td>1</td>
</tr>
<tr>
<td>lineWidth</td>
<td>0</td>
</tr>
<tr>
<td>singleColor</td>
<td>(0, 0, 0, 255)</td>
</tr>
<tr>
<td>boundaryNames</td>
<td>()</td>
</tr>
<tr>
<td>opacity</td>
<td>1</td>
</tr>
<tr>
<td>wireframe</td>
<td>0</td>
</tr>
<tr>
<td>drawInternal</td>
<td>0</td>
</tr>
<tr>
<td>smoothingLevel</td>
<td>0</td>
</tr>
<tr>
<td>cleanZonesOnly</td>
<td>0</td>
</tr>
<tr>
<td>mixedColor</td>
<td>(255, 255, 255, 255)</td>
</tr>
<tr>
<td>pointSize</td>
<td>0.05</td>
</tr>
<tr>
<td>pointType</td>
<td>Point, Box, Axis, Icosahedron, Octahedron, Tetrahedron, SphereGeometry, Sphere</td>
</tr>
<tr>
<td>pointSizeVarEnabled</td>
<td>0</td>
</tr>
<tr>
<td>pointSizeVar</td>
<td>“default”</td>
</tr>
<tr>
<td>pointSizePixels</td>
<td>2</td>
</tr>
</tbody>
</table>

### 2.5.32 Flux:

#### FluxAttributes()

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>flowField</td>
<td>“default”</td>
</tr>
<tr>
<td>weight</td>
<td>0</td>
</tr>
<tr>
<td>weightField</td>
<td>“default”</td>
</tr>
</tbody>
</table>
2.5.33 Font: \textit{FontAttributes()}

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>font</td>
<td>Arial, Courier, Times</td>
</tr>
<tr>
<td>scale</td>
<td>1</td>
</tr>
<tr>
<td>useForegroundColor</td>
<td>1</td>
</tr>
<tr>
<td>color</td>
<td>(0, 0, 0, 255)</td>
</tr>
<tr>
<td>bold</td>
<td>0</td>
</tr>
<tr>
<td>italic</td>
<td>0</td>
</tr>
</tbody>
</table>

2.5.34 Global: \textit{GlobalAttributes()}
<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>sources</td>
<td>()</td>
</tr>
<tr>
<td>windows</td>
<td>(1)</td>
</tr>
<tr>
<td>activeWindow</td>
<td>0</td>
</tr>
<tr>
<td>iconifiedFlag</td>
<td>0</td>
</tr>
<tr>
<td>autoUpdateFlag</td>
<td>0</td>
</tr>
<tr>
<td>replacePlots</td>
<td>0</td>
</tr>
<tr>
<td>applyOperator</td>
<td>1</td>
</tr>
<tr>
<td>applySelection</td>
<td>1</td>
</tr>
<tr>
<td>applyWindow</td>
<td>0</td>
</tr>
<tr>
<td>executing</td>
<td>0</td>
</tr>
<tr>
<td>windowLayout</td>
<td>1</td>
</tr>
<tr>
<td>makeDefaultConfirm</td>
<td>1</td>
</tr>
<tr>
<td>cloneWindowOnFirstRef</td>
<td>0</td>
</tr>
<tr>
<td>automaticallyAddOperator</td>
<td>0</td>
</tr>
<tr>
<td>tryHarderCyclesTimes</td>
<td>0</td>
</tr>
<tr>
<td>treatAllDBsAsTimeVarying</td>
<td>0</td>
</tr>
<tr>
<td>createMeshQualityExpressions</td>
<td>1</td>
</tr>
<tr>
<td>createTimeDerivativeExpressions</td>
<td>1</td>
</tr>
<tr>
<td>createVectorMagnitudeExpressions</td>
<td>1</td>
</tr>
<tr>
<td>newPlotsInheritSILRestriction</td>
<td>1</td>
</tr>
<tr>
<td>userDirForSessionFiles</td>
<td>0</td>
</tr>
<tr>
<td>save Crash Recovery File</td>
<td>1</td>
</tr>
<tr>
<td>ignoreExtentsFromDbs</td>
<td>0</td>
</tr>
<tr>
<td>expandNewPlots</td>
<td>0</td>
</tr>
<tr>
<td>userRestoreSessionFile</td>
<td>0</td>
</tr>
<tr>
<td>precisionType</td>
<td>Native, Float, Double</td>
</tr>
<tr>
<td>backendType</td>
<td>VTK, VTKM</td>
</tr>
<tr>
<td>removeDuplicateNodes</td>
<td>0</td>
</tr>
</tbody>
</table>

2.5.35 Histogram:

```
HistogramAttributes()
```
<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>basedOn</td>
<td>ManyZonesForSingleVar, ManyVarsForSingleZone</td>
</tr>
<tr>
<td>histogramType</td>
<td>Frequency, Weighted, Variable</td>
</tr>
<tr>
<td>weightVariable</td>
<td>“default”</td>
</tr>
<tr>
<td>limitsMode</td>
<td>OriginalData, CurrentPlot</td>
</tr>
<tr>
<td>minFlag</td>
<td>0</td>
</tr>
<tr>
<td>maxFlag</td>
<td>0</td>
</tr>
<tr>
<td>min</td>
<td>0</td>
</tr>
<tr>
<td>max</td>
<td>1</td>
</tr>
<tr>
<td>numBins</td>
<td>32</td>
</tr>
<tr>
<td>domain</td>
<td>0</td>
</tr>
<tr>
<td>zone</td>
<td>0</td>
</tr>
<tr>
<td>useBinWidths</td>
<td>1</td>
</tr>
<tr>
<td>outputType</td>
<td>Block, Curve</td>
</tr>
<tr>
<td>lineWidth</td>
<td>0</td>
</tr>
<tr>
<td>color</td>
<td>(200, 80, 40, 255)</td>
</tr>
<tr>
<td>dataScale</td>
<td>Linear, Log, SquareRoot</td>
</tr>
<tr>
<td>binScale</td>
<td>Linear, Log, SquareRoot</td>
</tr>
<tr>
<td>normalizeHistogram</td>
<td>0</td>
</tr>
<tr>
<td>computeAsCDF</td>
<td>0</td>
</tr>
</tbody>
</table>

### 2.5.36 IndexSelect:

*IndexSelectAttributes()*
2.5.37 IntegralCurve:  

IntegralCurveAttributes()

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>attributeName</td>
<td>“Whole”</td>
</tr>
<tr>
<td>subsetName</td>
<td>“Whole”</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>maxDim</td>
<td>ThreeD, OneD, TwoD</td>
</tr>
<tr>
<td>dim</td>
<td>TwoD, OneD, ThreeD</td>
</tr>
<tr>
<td>xAbsMax</td>
<td>-1</td>
</tr>
<tr>
<td>xMin</td>
<td>0</td>
</tr>
<tr>
<td>xMax</td>
<td>-1</td>
</tr>
<tr>
<td>xIncr</td>
<td>1</td>
</tr>
<tr>
<td>xWrap</td>
<td>0</td>
</tr>
<tr>
<td>yAbsMax</td>
<td>-1</td>
</tr>
<tr>
<td>yMin</td>
<td>0</td>
</tr>
<tr>
<td>yMax</td>
<td>-1</td>
</tr>
<tr>
<td>yIncr</td>
<td>1</td>
</tr>
<tr>
<td>yWrap</td>
<td>0</td>
</tr>
<tr>
<td>zAbsMax</td>
<td>-1</td>
</tr>
<tr>
<td>zMin</td>
<td>0</td>
</tr>
<tr>
<td>zMax</td>
<td>-1</td>
</tr>
<tr>
<td>zIncr</td>
<td>1</td>
</tr>
<tr>
<td>zWrap</td>
<td>0</td>
</tr>
<tr>
<td>useWholeCollection</td>
<td>1</td>
</tr>
<tr>
<td>sourceType</td>
<td>SpecifiedPoint, PointList, SpecifiedLine, Circle, SpecifiedPlane, SpecifiedSphere, SpecifiedBox, Selection, FieldData</td>
</tr>
<tr>
<td>pointSource</td>
<td>(0, 0, 0)</td>
</tr>
<tr>
<td>lineStart</td>
<td>(0, 0, 0)</td>
</tr>
<tr>
<td>lineEnd</td>
<td>(1, 0, 0)</td>
</tr>
<tr>
<td>planeOrigin</td>
<td>(0, 0, 0)</td>
</tr>
<tr>
<td>planeNormal</td>
<td>(0, 0, 1)</td>
</tr>
<tr>
<td>planeUpAxis</td>
<td>(0, 1, 0)</td>
</tr>
<tr>
<td>radius</td>
<td>1</td>
</tr>
<tr>
<td>sphereOrigin</td>
<td>(0, 0, 0)</td>
</tr>
<tr>
<td>boxExtents</td>
<td>(0, 1, 0, 1, 0, 1)</td>
</tr>
<tr>
<td>useWholeBox</td>
<td>1</td>
</tr>
<tr>
<td>pointList</td>
<td>(0, 0, 1, 0, 0, 1, 0, 0, 1, 0)</td>
</tr>
<tr>
<td>fieldData</td>
<td>()</td>
</tr>
<tr>
<td>sampleDensity0</td>
<td>2</td>
</tr>
<tr>
<td>sampleDensity1</td>
<td>2</td>
</tr>
<tr>
<td>Sample Density2</td>
<td>2</td>
</tr>
<tr>
<td>----------------</td>
<td>---</td>
</tr>
<tr>
<td>DataValue</td>
<td>TimeAbsolute, Solid, SeedPointID, Speed, Vorticity, ArcLength, TimeRelative, AverageDistanceFromSeed, CorrelationDistance, Difference, Variable</td>
</tr>
<tr>
<td>Integration Direction</td>
<td>&quot;...&quot;</td>
</tr>
<tr>
<td>MaxSteps</td>
<td>1000</td>
</tr>
<tr>
<td>Terminate By Distance</td>
<td>0</td>
</tr>
<tr>
<td>Term Distance</td>
<td>10</td>
</tr>
<tr>
<td>Terminate By Time</td>
<td>0</td>
</tr>
<tr>
<td>Term Time</td>
<td>10</td>
</tr>
<tr>
<td>Max Step Length</td>
<td>0.1</td>
</tr>
<tr>
<td>Limit Maximum Timestep</td>
<td>0</td>
</tr>
<tr>
<td>Max Time Step</td>
<td>0.1</td>
</tr>
<tr>
<td>RelTol</td>
<td>0.0001</td>
</tr>
<tr>
<td>AbsTol Size Type</td>
<td>FractionOfBBox, Absolute</td>
</tr>
<tr>
<td>AbsTol Absolute</td>
<td>1e-06</td>
</tr>
<tr>
<td>AbsTol BBox</td>
<td>1e-06</td>
</tr>
<tr>
<td>Field Type</td>
<td>Default, FlashField, M3DC12DField, M3DC13DField, Nek5000Field, NektarPPField, NIMRODField</td>
</tr>
<tr>
<td>Field Constant</td>
<td>1</td>
</tr>
<tr>
<td>Velocity Source</td>
<td>(0, 0, 0)</td>
</tr>
<tr>
<td>Integration Type</td>
<td>DormandPrince, Euler, Leapfrog, Adams Bashforth, RK4, M3DC12DIntegrator</td>
</tr>
<tr>
<td>Parallelization Algorithm Type</td>
<td>VisitSelects, LoadOnDemand, Parallel Static Domains, MasterSlave</td>
</tr>
<tr>
<td>Max Process Count</td>
<td>10</td>
</tr>
<tr>
<td>Max Domain Cache Size</td>
<td>3</td>
</tr>
<tr>
<td>Work Group Size</td>
<td>32</td>
</tr>
<tr>
<td>Pathlines</td>
<td>0</td>
</tr>
<tr>
<td>Pathlines Override Starting Time Flag</td>
<td>0</td>
</tr>
<tr>
<td>Pathlines Override Starting Time</td>
<td>0</td>
</tr>
<tr>
<td>Pathlines Period</td>
<td>0</td>
</tr>
<tr>
<td>Pathlines CMFE</td>
<td>POS_CMFE, CONN_CMFE</td>
</tr>
<tr>
<td>Display Geometry</td>
<td>Lines, Tubes, Ribbons</td>
</tr>
<tr>
<td>Cleanup Method</td>
<td>NoCleanup, Merge, Before, After</td>
</tr>
<tr>
<td>Cleanup Threshold</td>
<td>1e-08</td>
</tr>
<tr>
<td>Crop Begin Flag</td>
<td>0</td>
</tr>
<tr>
<td>Crop Begin</td>
<td>0</td>
</tr>
<tr>
<td>Crop End Flag</td>
<td>0</td>
</tr>
<tr>
<td>Crop End</td>
<td>0</td>
</tr>
<tr>
<td>Crop Value</td>
<td>Time, Distance, StepNumber</td>
</tr>
<tr>
<td>Sample Distance0</td>
<td>10</td>
</tr>
<tr>
<td>Sample Distance1</td>
<td>10</td>
</tr>
<tr>
<td>Sample Distance2</td>
<td>10</td>
</tr>
<tr>
<td>Fill Interior</td>
<td>1</td>
</tr>
<tr>
<td>Random Samples</td>
<td>0</td>
</tr>
<tr>
<td>Random Seed</td>
<td>0</td>
</tr>
<tr>
<td>Number Of Random Samples</td>
<td>1</td>
</tr>
<tr>
<td>Issue Advection Warnings</td>
<td>1</td>
</tr>
<tr>
<td>Issue Boundary Warnings</td>
<td>1</td>
</tr>
<tr>
<td>Issue Termination Warnings</td>
<td>1</td>
</tr>
<tr>
<td>Issue Stepsize Warnings</td>
<td>1</td>
</tr>
<tr>
<td>Issue Stiffness Warnings</td>
<td>1</td>
</tr>
<tr>
<td>Issue Critical Points Warnings</td>
<td>1</td>
</tr>
</tbody>
</table>
### 2.5.38 InverseGhostZone:

**InverseGhostZoneAttributes()**

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>requestGhostZones</td>
<td>1</td>
</tr>
<tr>
<td>showDuplicated</td>
<td>1</td>
</tr>
<tr>
<td>showEnhancedConnectivity</td>
<td>1</td>
</tr>
<tr>
<td>showReducedConnectivity</td>
<td>1</td>
</tr>
<tr>
<td>showAMRRefined</td>
<td>1</td>
</tr>
<tr>
<td>showExterior</td>
<td>1</td>
</tr>
<tr>
<td>showNotApplicable</td>
<td>1</td>
</tr>
</tbody>
</table>

### 2.5.39 Isosurface:

**IsosurfaceAttributes()**

Table 2.3 – continued from previous page

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>criticalPointThreshold</td>
<td>0.001</td>
</tr>
<tr>
<td>correlationDistanceAngTol</td>
<td>5</td>
</tr>
<tr>
<td>correlationDistanceMinDistAbsolute</td>
<td>1</td>
</tr>
<tr>
<td>correlationDistanceMinDistBBox</td>
<td>0.005</td>
</tr>
<tr>
<td>correlationDistanceMinDistType</td>
<td>FractionOfBBox, Absolute</td>
</tr>
<tr>
<td>selection</td>
<td>&quot;&quot;</td>
</tr>
</tbody>
</table>
### 2.5.40 Isovolume:

**IsovolumeAttributes()**

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>lbound</td>
<td>-1e+37</td>
</tr>
<tr>
<td>ubound</td>
<td>1e+37</td>
</tr>
<tr>
<td>variable</td>
<td>“default”</td>
</tr>
</tbody>
</table>

### 2.5.41 Keyframe:

**KeyframeAttributes()**

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>enabled</td>
<td>0</td>
</tr>
<tr>
<td>nFrames</td>
<td>1</td>
</tr>
<tr>
<td>nFrames Was UserSet</td>
<td>0</td>
</tr>
</tbody>
</table>
2.5.42 LCS: \[ \text{LCSAttributes}(\) \]

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>sourceType</td>
<td>NativeMesh, RegularGrid</td>
</tr>
<tr>
<td>Resolution</td>
<td>(10, 10, 10)</td>
</tr>
<tr>
<td>UseDataSetStart</td>
<td>Full, Subset</td>
</tr>
<tr>
<td>StartPosition</td>
<td>(0, 0, 0)</td>
</tr>
<tr>
<td>UseDataSetEnd</td>
<td>Full, Subset</td>
</tr>
<tr>
<td>EndPosition</td>
<td>(1, 1, 1)</td>
</tr>
<tr>
<td>integrationDirection</td>
<td>Forward, Backward, Both</td>
</tr>
<tr>
<td>auxiliaryGrid</td>
<td>None, TwoDim, ThreeDim</td>
</tr>
<tr>
<td>auxiliaryGridSpacing</td>
<td>0.0001</td>
</tr>
<tr>
<td>maxSteps</td>
<td>1000</td>
</tr>
<tr>
<td>operationType</td>
<td>Lyapunov, IntegrationTime, ArcLength, AverageDistanceFromSeed, EigenValue, EigenVector</td>
</tr>
<tr>
<td>cauchyGreenTensor</td>
<td>Right, Left</td>
</tr>
<tr>
<td>eigenComponent</td>
<td>Largest, Smallest, Intermediate, PosShearVector, NegShearVector, PosLambdaShearVector,</td>
</tr>
<tr>
<td>eigenWeight</td>
<td>1</td>
</tr>
<tr>
<td>operatorType</td>
<td>BaseValue, Gradient</td>
</tr>
<tr>
<td>terminateType</td>
<td>Time, Distance, Size</td>
</tr>
<tr>
<td>terminateBySize</td>
<td>0</td>
</tr>
<tr>
<td>termSize</td>
<td>10</td>
</tr>
<tr>
<td>terminateByDistance</td>
<td>0</td>
</tr>
<tr>
<td>termDistance</td>
<td>10</td>
</tr>
<tr>
<td>terminateByTime</td>
<td>0</td>
</tr>
<tr>
<td>termTime</td>
<td>10</td>
</tr>
<tr>
<td>maxStepLength</td>
<td>0.1</td>
</tr>
<tr>
<td>limitMaximumTimestep</td>
<td>0</td>
</tr>
<tr>
<td>maxTimeStep</td>
<td>1000</td>
</tr>
<tr>
<td>relTol</td>
<td>0.0001</td>
</tr>
<tr>
<td>absTolSizeType</td>
<td>FractionOfBBox, Absolute</td>
</tr>
<tr>
<td>absTolAbsolute</td>
<td>1e-06</td>
</tr>
<tr>
<td>absTolBBox</td>
<td>1e-06</td>
</tr>
<tr>
<td>fieldType</td>
<td>Default, FlashField, M3DC12DField, M3DC13DField, Nek5000Field, NektarPPField, NIMROD</td>
</tr>
<tr>
<td>fieldConstant</td>
<td>1</td>
</tr>
<tr>
<td>velocitySource</td>
<td>(0, 0, 0)</td>
</tr>
<tr>
<td>integrationType</td>
<td>DormandPrince, Euler, Leapfrog, AdamsBashforth, RK4, M3DC12DIntegrator</td>
</tr>
<tr>
<td>clampLogValues</td>
<td>0</td>
</tr>
<tr>
<td>parallelizationAlgorithmType</td>
<td>VisItSelects, LoadOnDemand, ParallelStaticDomains, MasterSlave</td>
</tr>
<tr>
<td>maxProcessCount</td>
<td>10</td>
</tr>
<tr>
<td>maxDomainCacheSize</td>
<td>3</td>
</tr>
<tr>
<td>workGroupSize</td>
<td>32</td>
</tr>
<tr>
<td>pathlines</td>
<td>0</td>
</tr>
<tr>
<td>pathlinesOverrideStartingTimeFlag</td>
<td>0</td>
</tr>
<tr>
<td>pathlinesOverrideStartingTime</td>
<td>0</td>
</tr>
<tr>
<td>pathlinesPeriod</td>
<td>0</td>
</tr>
<tr>
<td>pathlinesCMFE</td>
<td>POS_CMFE, CONN_CMFE</td>
</tr>
<tr>
<td>thresholdLimit</td>
<td>0.1</td>
</tr>
<tr>
<td>radialLimit</td>
<td>0.1</td>
</tr>
</tbody>
</table>

Continued on next page
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>boundaryLimit</td>
<td>0.1</td>
</tr>
<tr>
<td>seedLimit</td>
<td>10</td>
</tr>
<tr>
<td>issueAdvectionWarnings</td>
<td>1</td>
</tr>
<tr>
<td>issueBoundaryWarnings</td>
<td>1</td>
</tr>
<tr>
<td>issueTerminationWarnings</td>
<td>1</td>
</tr>
<tr>
<td>issueStepsSizeWarnings</td>
<td>1</td>
</tr>
<tr>
<td>issueStiffnessWarnings</td>
<td>1</td>
</tr>
<tr>
<td>issueCriticalPointsWarnings</td>
<td>1</td>
</tr>
<tr>
<td>criticalPointThreshold</td>
<td>0.001</td>
</tr>
</tbody>
</table>

2.5.43 Label: $\text{LabelAttributes()}$
### Attribute

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>legendFlag</td>
<td>1</td>
</tr>
<tr>
<td>showNodes</td>
<td>0</td>
</tr>
<tr>
<td>showCells</td>
<td>1</td>
</tr>
<tr>
<td>restrictNumberOfLabels</td>
<td>1</td>
</tr>
<tr>
<td>drawLabelsFacing</td>
<td>Front, Back, FrontAndBack</td>
</tr>
<tr>
<td>labelDisplayFormat</td>
<td>Natural, LogicalIndex, Index</td>
</tr>
<tr>
<td>numberOfLabels</td>
<td>200</td>
</tr>
<tr>
<td>textFont1.font</td>
<td>Arial, Courier, Times</td>
</tr>
<tr>
<td>textFont1.scale</td>
<td>4</td>
</tr>
<tr>
<td>textFont1.useForegroundColor</td>
<td>1</td>
</tr>
<tr>
<td>textFont1.color</td>
<td>(255, 0, 0, 255)</td>
</tr>
<tr>
<td>textFont1.bold</td>
<td>0</td>
</tr>
<tr>
<td>textFont1.italic</td>
<td>0</td>
</tr>
<tr>
<td>textFont2.font</td>
<td>Arial, Courier, Times</td>
</tr>
<tr>
<td>textFont2.scale</td>
<td>4</td>
</tr>
<tr>
<td>textFont2.useForegroundColor</td>
<td>1</td>
</tr>
<tr>
<td>textFont2.color</td>
<td>(0, 0, 255, 255)</td>
</tr>
<tr>
<td>textFont2.bold</td>
<td>0</td>
</tr>
<tr>
<td>textFont2.italic</td>
<td>0</td>
</tr>
<tr>
<td>horizontalJustification</td>
<td>HCenter, Left, Right</td>
</tr>
<tr>
<td>verticalJustification</td>
<td>VCenter, Top, Bottom</td>
</tr>
<tr>
<td>depthTestMode</td>
<td>LABEL_DT_AUTO, LABEL_DT_ALWAYS, LABEL_DT_NEVER</td>
</tr>
<tr>
<td>formatTemplate</td>
<td>“%g”</td>
</tr>
</tbody>
</table>

### 2.5.44 Lagrangian:

**LagrangianAttributes()**

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>seedPoint</td>
<td>(0, 0, 0)</td>
</tr>
<tr>
<td>numSteps</td>
<td>1000</td>
</tr>
<tr>
<td>XAxisSample</td>
<td>Step, Time, ArcLength, Speed, Vorticity, Variable</td>
</tr>
<tr>
<td>YAxisSample</td>
<td>Step, Time, ArcLength, Speed, Vorticity, Variable</td>
</tr>
<tr>
<td>variable</td>
<td>&quot;default&quot;</td>
</tr>
</tbody>
</table>
2.5. Attribute Reference

### 2.5.45 Light:

**LightAttributes()**

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>enabledFlag</td>
<td>1</td>
</tr>
<tr>
<td>type</td>
<td><strong>Camera</strong>, Ambient, Object</td>
</tr>
<tr>
<td>direction</td>
<td>(0, 0, -1)</td>
</tr>
<tr>
<td>color</td>
<td>(255, 255, 255, 255)</td>
</tr>
<tr>
<td>brightness</td>
<td>1</td>
</tr>
</tbody>
</table>

### 2.5.46 LimitCycle:

**LimitCycleAttributes()**

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>sourceType</td>
<td><strong>SpecifiedLine</strong>, SpecifiedPlane</td>
</tr>
<tr>
<td>lineStart</td>
<td>(0, 0, 0)</td>
</tr>
<tr>
<td>lineEnd</td>
<td>(1, 0, 0)</td>
</tr>
<tr>
<td>planeOrigin</td>
<td>(0, 0, 0)</td>
</tr>
<tr>
<td>planeNormal</td>
<td>(0, 0, 1)</td>
</tr>
<tr>
<td>planeUpAxis</td>
<td>(0, 1, 0)</td>
</tr>
<tr>
<td>sampleDensity0</td>
<td>2</td>
</tr>
<tr>
<td>sampleDensity1</td>
<td>2</td>
</tr>
<tr>
<td>dataValue</td>
<td><strong>TimeAbsolute</strong>, Solid, SeedPointID, Speed, Vorticity, ArcLength, TimeRelative, AverageDistanceFromSeed, CorrelationDistance, Difference, Variable</td>
</tr>
<tr>
<td>dataVariable</td>
<td>&quot;&quot;</td>
</tr>
<tr>
<td>integrationDirection</td>
<td><strong>Forward</strong>, Backward, Both, ForwardDirectionless, BackwardDirectionless, BothDirectionless</td>
</tr>
<tr>
<td>maxSteps</td>
<td>1000</td>
</tr>
<tr>
<td>terminateByDistance</td>
<td>0</td>
</tr>
<tr>
<td>termDistance</td>
<td>10</td>
</tr>
<tr>
<td>terminateByTime</td>
<td>0</td>
</tr>
<tr>
<td>termTime</td>
<td>10</td>
</tr>
<tr>
<td>maxStepLength</td>
<td>0.1</td>
</tr>
<tr>
<td>limitMaximumTimestep</td>
<td>0</td>
</tr>
<tr>
<td>maxTimeStep</td>
<td>0.1</td>
</tr>
<tr>
<td>relTol</td>
<td>0.0001</td>
</tr>
<tr>
<td>absTolSizeType</td>
<td><strong>FractionOfBBBox</strong>, Absolute</td>
</tr>
<tr>
<td>absTolAbsolute</td>
<td>1e-06</td>
</tr>
<tr>
<td>absTolBBBox</td>
<td>1e-06</td>
</tr>
<tr>
<td>fieldType</td>
<td>Default, FlashField, M3DC12DField, M3DC13DField, Nek5000Field, NektarPPField, NIMRODField</td>
</tr>
<tr>
<td>--------------------</td>
<td>---------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>fieldConstant</td>
<td>1</td>
</tr>
<tr>
<td>velocitySource</td>
<td>(0, 0, 0)</td>
</tr>
<tr>
<td>integrationType</td>
<td>DormandPrince, Euler, Leapfrog, AdamsBashforth, RK4, M3DC12DIntegrator</td>
</tr>
<tr>
<td>parallelizationAlgorithmType</td>
<td>VisItSelects, LoadOnDemand, ParallelStaticDomains, MasterSlave</td>
</tr>
<tr>
<td>maxProcessCount</td>
<td>10</td>
</tr>
<tr>
<td>maxDomainCacheSize</td>
<td>3</td>
</tr>
<tr>
<td>workGroupSize</td>
<td>32</td>
</tr>
<tr>
<td>pathlines</td>
<td>0</td>
</tr>
<tr>
<td>pathlinesOverrideStartingTimeFlag</td>
<td>0</td>
</tr>
<tr>
<td>pathlinesOverrideStartingTime</td>
<td>0</td>
</tr>
<tr>
<td>pathlinesPeriod</td>
<td>0</td>
</tr>
<tr>
<td>pathlinesCMFE</td>
<td>POS_CMFE, CONN_CMFE</td>
</tr>
<tr>
<td>sampleDistance0</td>
<td>10</td>
</tr>
<tr>
<td>sampleDistance1</td>
<td>10</td>
</tr>
<tr>
<td>sampleDistance2</td>
<td>10</td>
</tr>
<tr>
<td>fillInterior</td>
<td>1</td>
</tr>
<tr>
<td>randomSamples</td>
<td>0</td>
</tr>
<tr>
<td>randomSeed</td>
<td>0</td>
</tr>
<tr>
<td>numberOfRandomSamples</td>
<td>1</td>
</tr>
<tr>
<td>forceNodeCenteredData</td>
<td>0</td>
</tr>
<tr>
<td>cycleTolerance</td>
<td>1e-06</td>
</tr>
<tr>
<td>maxIterations</td>
<td>10</td>
</tr>
<tr>
<td>showPartialResults</td>
<td>1</td>
</tr>
<tr>
<td>showReturnDistances</td>
<td>0</td>
</tr>
<tr>
<td>issueAdvectionWarnings</td>
<td>1</td>
</tr>
<tr>
<td>issueBoundaryWarnings</td>
<td>1</td>
</tr>
<tr>
<td>issueTerminationWarnings</td>
<td>1</td>
</tr>
<tr>
<td>issueStepsizeWarnings</td>
<td>1</td>
</tr>
<tr>
<td>issueStiffnessWarnings</td>
<td>1</td>
</tr>
<tr>
<td>issueCriticalPointsWarnings</td>
<td>1</td>
</tr>
<tr>
<td>criticalPointThreshold</td>
<td>0.001</td>
</tr>
<tr>
<td>correlationDistanceAngTol</td>
<td>5</td>
</tr>
<tr>
<td>correlationDistanceMinDistAbsolute</td>
<td>1</td>
</tr>
<tr>
<td>correlationDistanceMinDistBBBox</td>
<td>0.005</td>
</tr>
<tr>
<td>correlationDistanceMinDistType</td>
<td>FractionOfBBox, Absolute</td>
</tr>
</tbody>
</table>

### 2.5.47 Lineout:

**LineoutAttributes()**
### Attribute Reference

#### 2.5.48 Material:

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>smoothing</td>
<td>0</td>
</tr>
<tr>
<td>forceMIR</td>
<td>0</td>
</tr>
<tr>
<td>cleanZonesOnly</td>
<td>0</td>
</tr>
<tr>
<td>needValidConnectivity</td>
<td>0</td>
</tr>
<tr>
<td>algorithm</td>
<td>EquiZ, EquiT, Isovolume, PLIC, Discrete</td>
</tr>
<tr>
<td>iterationEnabled</td>
<td>0</td>
</tr>
<tr>
<td>numIterations</td>
<td>5</td>
</tr>
<tr>
<td>iterationDamping</td>
<td>0.4</td>
</tr>
<tr>
<td>simplifyHeavilyMixedZones</td>
<td>0</td>
</tr>
<tr>
<td>maxMaterialsPerZone</td>
<td>3</td>
</tr>
<tr>
<td>isoVolumeFraction</td>
<td>0.5</td>
</tr>
<tr>
<td>annealingTime</td>
<td>10</td>
</tr>
</tbody>
</table>

#### 2.5.49 Mesh:

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>point1</td>
<td>(0, 0, 0)</td>
</tr>
<tr>
<td>point2</td>
<td>(1, 1, 0)</td>
</tr>
<tr>
<td>interactive</td>
<td>0</td>
</tr>
<tr>
<td>ignoreGlobal</td>
<td>0</td>
</tr>
<tr>
<td>samplingOn</td>
<td>0</td>
</tr>
<tr>
<td>numberOfSamplePoints</td>
<td>50</td>
</tr>
<tr>
<td>refineLabels</td>
<td>0</td>
</tr>
<tr>
<td>Attribute</td>
<td>Default/Allowed Values</td>
</tr>
<tr>
<td>-----------------</td>
<td>--------------------------</td>
</tr>
<tr>
<td>legendFlag</td>
<td>1</td>
</tr>
<tr>
<td>lineWidth</td>
<td>0</td>
</tr>
<tr>
<td>meshColor</td>
<td>(0, 0, 0, 255)</td>
</tr>
<tr>
<td>meshColorSource</td>
<td>Foreground, MeshCustom</td>
</tr>
<tr>
<td>opaqueColorSource</td>
<td>Background, OpaqueCustom</td>
</tr>
<tr>
<td>opaqueMode</td>
<td>Auto, On, Off</td>
</tr>
<tr>
<td>pointSize</td>
<td>0.05</td>
</tr>
<tr>
<td>opaqueColor</td>
<td>(255, 255, 255, 255)</td>
</tr>
<tr>
<td>smoothingLevel</td>
<td>None, Fast, High</td>
</tr>
<tr>
<td>pointSizeVarEnabled</td>
<td>0</td>
</tr>
<tr>
<td>pointSizeVar</td>
<td>&quot;default&quot;</td>
</tr>
<tr>
<td>pointType</td>
<td>Point, Box, Axis, Icosahedron, Octahedron, Tetrahedron, SphereGeometry, Sphere</td>
</tr>
<tr>
<td>showInternal</td>
<td>0</td>
</tr>
<tr>
<td>pointSizePixels</td>
<td>2</td>
</tr>
<tr>
<td>opacity</td>
<td>1</td>
</tr>
</tbody>
</table>

### 2.5.50 MeshManagement: MeshManagementAttributes()

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>discretizationTolerance</td>
<td>(0.02, 0.025, 0.05)</td>
</tr>
<tr>
<td>discretizationToleranceX</td>
<td>()</td>
</tr>
<tr>
<td>discretizationToleranceY</td>
<td>()</td>
</tr>
<tr>
<td>discretizationToleranceZ</td>
<td>()</td>
</tr>
<tr>
<td>discretizationMode</td>
<td>Uniform, Adaptive, MultiPass</td>
</tr>
<tr>
<td>discretizeBoundaryOnly</td>
<td>0</td>
</tr>
<tr>
<td>passNativeCSG</td>
<td>0</td>
</tr>
</tbody>
</table>

### 2.5.51 Molecule: MoleculeAttributes()
<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>drawAtomsAs</td>
<td>SphereAtoms, NoAtoms, ImposterAtoms</td>
</tr>
<tr>
<td>scaleRadiusBy</td>
<td>Fixed, Covalent, Atomic, Variable</td>
</tr>
<tr>
<td>drawBondsAs</td>
<td>CylinderBonds, NoBonds, LineBonds</td>
</tr>
<tr>
<td>colorBonds</td>
<td>ColorByAtom, SingleColor</td>
</tr>
<tr>
<td>bondSingleColor</td>
<td>(128, 128, 128, 255)</td>
</tr>
<tr>
<td>radiusVariable</td>
<td>&quot;default&quot;</td>
</tr>
<tr>
<td>radiusScaleFactor</td>
<td>1</td>
</tr>
<tr>
<td>radiusFixed</td>
<td>0.3</td>
</tr>
<tr>
<td>atomSphereQuality</td>
<td>Medium, Low, High, Super</td>
</tr>
<tr>
<td>bondCylinderQuality</td>
<td>Medium, Low, High, Super</td>
</tr>
<tr>
<td>bondRadius</td>
<td>0.12</td>
</tr>
<tr>
<td>bondLineWidth</td>
<td>0</td>
</tr>
<tr>
<td>elementColorTable</td>
<td>&quot;cpk_jmol&quot;</td>
</tr>
<tr>
<td>residueTypeColorTable</td>
<td>&quot;amino_shapely&quot;</td>
</tr>
<tr>
<td>residueSequenceColorTable</td>
<td>&quot;Default&quot;</td>
</tr>
<tr>
<td>continuousColorTable</td>
<td>&quot;Default&quot;</td>
</tr>
<tr>
<td>legendFlag</td>
<td>1</td>
</tr>
<tr>
<td>minFlag</td>
<td>0</td>
</tr>
<tr>
<td>scalarMin</td>
<td>0</td>
</tr>
<tr>
<td>maxFlag</td>
<td>0</td>
</tr>
<tr>
<td>scalarMax</td>
<td>1</td>
</tr>
</tbody>
</table>

### 2.5.52 MultiCurve: `MultiCurveAttributes()`

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>defaultPalette.GetControlPoints(0).colors</td>
<td>(255, 0, 0, 255)</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(0).position</td>
<td>0</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(1).colors</td>
<td>(0, 255, 0, 255)</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(1).position</td>
<td>0.034</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(2).colors</td>
<td>(0, 0, 255, 255)</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(2).position</td>
<td>0.069</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(3).colors</td>
<td>(0, 255, 255, 255)</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(3).position</td>
<td>0.103</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(4).colors</td>
<td>(255, 0, 255, 255)</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(4).position</td>
<td>0.138</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(5).colors</td>
<td>(255, 255, 0, 255)</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(5).position</td>
<td>0.172</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(6).colors</td>
<td>(255, 135, 0, 255)</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(6).position</td>
<td>0.207</td>
</tr>
</tbody>
</table>

Continued on next page
Table 2.6 – continued from previous page

<table>
<thead>
<tr>
<th>Access Path</th>
<th>Color</th>
</tr>
</thead>
<tbody>
<tr>
<td>defaultPalette.GetControlPoints(7).colors</td>
<td>(255, 0, 135, 255)</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(7).position</td>
<td>0.241</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(8).colors</td>
<td>(168, 168, 255)</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(8).position</td>
<td>0.276</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(9).colors</td>
<td>(255, 68, 255)</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(9).position</td>
<td>0.31</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(10).position</td>
<td>0.345</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(11).position</td>
<td>0.379</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(12).position</td>
<td>0.414</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(13).position</td>
<td>0.448</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(14).position</td>
<td>0.483</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(15).colors</td>
<td>(255, 170, 99, 255)</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(15).position</td>
<td>0.517</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(16).position</td>
<td>0.552</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(17).colors</td>
<td>(150, 0, 0, 255)</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(17).position</td>
<td>0.586</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(18).colors</td>
<td>(0, 150, 0, 255)</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(18).position</td>
<td>0.621</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(19).colors</td>
<td>(0, 0, 150, 255)</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(19).position</td>
<td>0.655</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(20).colors</td>
<td>(0, 109, 109, 255)</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(20).position</td>
<td>0.69</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(21).colors</td>
<td>(150, 0, 150, 255)</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(21).position</td>
<td>0.724</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(22).colors</td>
<td>(150, 150, 0, 255)</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(22).position</td>
<td>0.759</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(23).colors</td>
<td>(150, 84, 0, 255)</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(23).position</td>
<td>0.793</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(24).colors</td>
<td>(160, 0, 79, 255)</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(24).position</td>
<td>0.828</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(25).colors</td>
<td>(255, 104, 28, 255)</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(25).position</td>
<td>0.862</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(26).colors</td>
<td>(0, 170, 81, 255)</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(26).position</td>
<td>0.897</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(27).colors</td>
<td>(68, 255, 124, 255)</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(27).position</td>
<td>0.931</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(28).colors</td>
<td>(0, 130, 255, 255)</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(28).position</td>
<td>0.966</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(29).colors</td>
<td>(130, 0, 255, 255)</td>
</tr>
<tr>
<td>defaultPalette.GetControlPoints(29).position</td>
<td>1</td>
</tr>
<tr>
<td>defaultPalette.smoothing</td>
<td>None, Linear, CubicSpline</td>
</tr>
<tr>
<td>defaultPalette.equalSpacingFlag</td>
<td>1</td>
</tr>
<tr>
<td>defaultPalette.discreteFlag</td>
<td>1</td>
</tr>
<tr>
<td>defaultPalette.categoryName</td>
<td>“Standard”</td>
</tr>
</tbody>
</table>

Continued on next page
Table 2.6 – continued from previous page

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>changedColors</td>
<td>()</td>
</tr>
<tr>
<td>colorType</td>
<td><strong>ColorByMultipleColors</strong>, <strong>ColorBySingleColor</strong></td>
</tr>
<tr>
<td>singleColor</td>
<td>(255, 0, 0, 255)</td>
</tr>
<tr>
<td></td>
<td>SetMultiColor(0, (255, 0, 0, 255))</td>
</tr>
<tr>
<td></td>
<td>SetMultiColor(1, (0, 255, 0, 255))</td>
</tr>
<tr>
<td></td>
<td>SetMultiColor(2, (0, 0, 255, 255))</td>
</tr>
<tr>
<td></td>
<td>SetMultiColor(3, (0, 255, 255, 255))</td>
</tr>
<tr>
<td></td>
<td>SetMultiColor(4, (255, 0, 255, 255))</td>
</tr>
<tr>
<td></td>
<td>SetMultiColor(5, (255, 255, 0, 255))</td>
</tr>
<tr>
<td></td>
<td>SetMultiColor(6, (255, 135, 0, 255))</td>
</tr>
<tr>
<td></td>
<td>SetMultiColor(7, (255, 0, 135, 255))</td>
</tr>
<tr>
<td></td>
<td>SetMultiColor(8, (168, 168, 168, 255))</td>
</tr>
<tr>
<td></td>
<td>SetMultiColor(9, (255, 68, 68, 255))</td>
</tr>
<tr>
<td></td>
<td>SetMultiColor(10, (99, 255, 99, 255))</td>
</tr>
<tr>
<td></td>
<td>SetMultiColor(11, (99, 99, 255, 255))</td>
</tr>
<tr>
<td></td>
<td>SetMultiColor(12, (40, 165, 165, 255))</td>
</tr>
<tr>
<td></td>
<td>SetMultiColor(13, (255, 99, 255, 255))</td>
</tr>
<tr>
<td></td>
<td>SetMultiColor(14, (255, 255, 99, 255))</td>
</tr>
<tr>
<td></td>
<td>SetMultiColor(15, (255, 170, 99, 255))</td>
</tr>
<tr>
<td>lineWidth</td>
<td>0</td>
</tr>
<tr>
<td>yAxisTitleFormat</td>
<td>“%g”</td>
</tr>
<tr>
<td>useYAxisTickSpacing</td>
<td>0</td>
</tr>
<tr>
<td>yAxisTickSpacing</td>
<td>1</td>
</tr>
<tr>
<td>displayMarkers</td>
<td>1</td>
</tr>
<tr>
<td>markerScale</td>
<td>1</td>
</tr>
<tr>
<td>markerLineWidth</td>
<td>0</td>
</tr>
<tr>
<td>markerVariable</td>
<td>“default”</td>
</tr>
<tr>
<td>displayIds</td>
<td>0</td>
</tr>
<tr>
<td>idVariable</td>
<td>“default”</td>
</tr>
<tr>
<td>legendFlag</td>
<td>1</td>
</tr>
</tbody>
</table>

2.5.53 MultiresControl:

MultiresControlAttributes()
### OnionPeel:

#### OnionPeelAttributes()

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>adjacencyType</td>
<td>Node, Face</td>
</tr>
<tr>
<td>useGlobalId</td>
<td>0</td>
</tr>
<tr>
<td>categoryName</td>
<td>“Whole”</td>
</tr>
<tr>
<td>subsetName</td>
<td>“Whole”</td>
</tr>
<tr>
<td>index</td>
<td>(0)</td>
</tr>
<tr>
<td>logical</td>
<td>0</td>
</tr>
<tr>
<td>requestedLayer</td>
<td>0</td>
</tr>
<tr>
<td>seedType</td>
<td>SeedCell, SeedNode</td>
</tr>
<tr>
<td>honorOriginalMesh</td>
<td>1</td>
</tr>
</tbody>
</table>

### ParallelCoordinates:

#### ParallelCoordinatesAttributes()
### Attribute Reference

#### Attribute Default/Allowed Values

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>scalarAxisNames</td>
<td>()</td>
</tr>
<tr>
<td>visualAxisNames</td>
<td>()</td>
</tr>
<tr>
<td>extentMinima</td>
<td>()</td>
</tr>
<tr>
<td>extentMaxima</td>
<td>()</td>
</tr>
<tr>
<td>drawLines</td>
<td>1</td>
</tr>
<tr>
<td>linesColor</td>
<td>(128, 0, 0, 255)</td>
</tr>
<tr>
<td>drawContext</td>
<td>1</td>
</tr>
<tr>
<td>contextGamma</td>
<td>2</td>
</tr>
<tr>
<td>contextNumPartitions</td>
<td>128</td>
</tr>
<tr>
<td>contextColor</td>
<td>(0, 220, 0, 255)</td>
</tr>
<tr>
<td>drawLinesOnlyIfExtentsOn</td>
<td>1</td>
</tr>
<tr>
<td>unifyAxisExtents</td>
<td>0</td>
</tr>
<tr>
<td>linesNumPartitions</td>
<td>512</td>
</tr>
<tr>
<td>focusGamma</td>
<td>4</td>
</tr>
<tr>
<td>drawFocusAs</td>
<td><strong>BinsOfConstantColor</strong>, IndividualLines, BinsColoredByPopulation</td>
</tr>
</tbody>
</table>

#### PersistentParticles Attributes

### 2.5.56 PersistentParticles:

#### PersistentParticlesAttributes

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>startIndex</td>
<td>0</td>
</tr>
<tr>
<td>stopIndex</td>
<td>1</td>
</tr>
<tr>
<td>stride</td>
<td>1</td>
</tr>
<tr>
<td>startPathType</td>
<td><strong>Absolute, Relative</strong></td>
</tr>
<tr>
<td>stopPathType</td>
<td><strong>Absolute, Relative</strong></td>
</tr>
<tr>
<td>traceVariableX</td>
<td>“default”</td>
</tr>
<tr>
<td>traceVariableY</td>
<td>“default”</td>
</tr>
<tr>
<td>traceVariableZ</td>
<td>“default”</td>
</tr>
<tr>
<td>connectParticles</td>
<td>0</td>
</tr>
<tr>
<td>showPoints</td>
<td>0</td>
</tr>
<tr>
<td>indexVariable</td>
<td>“default”</td>
</tr>
</tbody>
</table>
### 2.5.57 Poincare:  

`PoincareAttributes()`

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>opacityType</td>
<td>Explicit, ColorTable</td>
</tr>
<tr>
<td>opacity</td>
<td>1</td>
</tr>
<tr>
<td>minPunctures</td>
<td>50</td>
</tr>
<tr>
<td>maxPunctures</td>
<td>500</td>
</tr>
<tr>
<td>puncturePlotType</td>
<td>Single, Double</td>
</tr>
<tr>
<td>maxSteps</td>
<td>1000</td>
</tr>
<tr>
<td>terminateByTime</td>
<td>0</td>
</tr>
<tr>
<td>termTime</td>
<td>10</td>
</tr>
<tr>
<td>puncturePeriodTolerance</td>
<td>0.01</td>
</tr>
<tr>
<td>puncturePlane</td>
<td>Poloidal, Toroidal, Arbitrary</td>
</tr>
<tr>
<td>sourceType</td>
<td>SpecifiedPoint, PointList, SpecifiedLine</td>
</tr>
<tr>
<td>pointSource</td>
<td>(0, 0, 0)</td>
</tr>
<tr>
<td>pointList</td>
<td>(0, 0, 1, 0, 1, 0, 1, 0)</td>
</tr>
<tr>
<td>lineStart</td>
<td>(0, 0, 0)</td>
</tr>
<tr>
<td>lineEnd</td>
<td>(1, 0, 0)</td>
</tr>
<tr>
<td>pointDensity</td>
<td>1</td>
</tr>
<tr>
<td>fieldType</td>
<td>Default, FlashField, M3DC12DField, M3DC13DField, Nek5000Field, NektarPPField, NIMRODField</td>
</tr>
<tr>
<td>forceNodeCenteredData</td>
<td>0</td>
</tr>
<tr>
<td>fieldConstant</td>
<td>1</td>
</tr>
<tr>
<td>velocitySource</td>
<td>(0, 0, 0)</td>
</tr>
<tr>
<td>integrationType</td>
<td>AdamsBashforth, Euler, Leapfrog, DornandPrince, RK4, M3DC12DIntegrator</td>
</tr>
<tr>
<td>coordinateSystem</td>
<td>Cartesian, Cylindrical</td>
</tr>
<tr>
<td>maxStepLength</td>
<td>0.1</td>
</tr>
<tr>
<td>limitMaximumTimestep</td>
<td>0</td>
</tr>
<tr>
<td>maxTimeStep</td>
<td>0.1</td>
</tr>
<tr>
<td>relTol</td>
<td>0.0001</td>
</tr>
<tr>
<td>absTolSizeType</td>
<td>FractionOfBBox, Absolute</td>
</tr>
<tr>
<td>absTolAbsolute</td>
<td>1e-05</td>
</tr>
<tr>
<td>absTolBBox</td>
<td>1e-06</td>
</tr>
<tr>
<td>analysis</td>
<td>Normal, None</td>
</tr>
<tr>
<td>maximumToroidalWinding</td>
<td>0</td>
</tr>
<tr>
<td>overrideToroidalWinding</td>
<td>0</td>
</tr>
<tr>
<td>overridePoloidalWinding</td>
<td>0</td>
</tr>
<tr>
<td>windingPairConfidence</td>
<td>0.9</td>
</tr>
<tr>
<td>rationalSurfaceFactor</td>
<td>0.1</td>
</tr>
<tr>
<td>overlaps</td>
<td>Remove, Raw, Merge, Smooth</td>
</tr>
<tr>
<td>meshType</td>
<td>Curves, Surfaces</td>
</tr>
<tr>
<td>numberPlanes</td>
<td>1</td>
</tr>
<tr>
<td>singlePlane</td>
<td>0</td>
</tr>
<tr>
<td>minFlag</td>
<td>0</td>
</tr>
<tr>
<td>max</td>
<td>0</td>
</tr>
<tr>
<td>minFlag</td>
<td>0</td>
</tr>
<tr>
<td>maxFlag</td>
<td>0</td>
</tr>
<tr>
<td>colorType</td>
<td>ColorByColorTable, ColorBySingleColor</td>
</tr>
<tr>
<td>singleColor</td>
<td>(0, 0, 0, 255)</td>
</tr>
<tr>
<td>Parameter</td>
<td>Value</td>
</tr>
<tr>
<td>---------------------------</td>
<td>------------------------------</td>
</tr>
<tr>
<td>colorTableName</td>
<td>“Default”</td>
</tr>
<tr>
<td>showRationalSurfaces</td>
<td>0</td>
</tr>
<tr>
<td>RationalSurfaceMaxIterations</td>
<td>2</td>
</tr>
<tr>
<td>showOPoints</td>
<td>0</td>
</tr>
<tr>
<td>OPointMaxIterations</td>
<td>2</td>
</tr>
<tr>
<td>showXPoints</td>
<td>0</td>
</tr>
<tr>
<td>XPointMaxIterations</td>
<td>2</td>
</tr>
<tr>
<td>performOLineAnalysis</td>
<td>0</td>
</tr>
<tr>
<td>OLineToroidalWinding</td>
<td>1</td>
</tr>
<tr>
<td>OLineAxisFileName</td>
<td>“”</td>
</tr>
<tr>
<td>showChaotic</td>
<td>0</td>
</tr>
<tr>
<td>showIslands</td>
<td>0</td>
</tr>
<tr>
<td>SummaryFlag</td>
<td>1</td>
</tr>
<tr>
<td>verboseFlag</td>
<td>0</td>
</tr>
<tr>
<td>show1DPlots</td>
<td>0</td>
</tr>
<tr>
<td>showLines</td>
<td>1</td>
</tr>
<tr>
<td>showPoints</td>
<td>0</td>
</tr>
<tr>
<td>parallelizationAlgorithmType</td>
<td>VisItSelects, LoadOnDemand, ParallelStaticDomains, MasterSlave</td>
</tr>
<tr>
<td>maxProcessCount</td>
<td>10</td>
</tr>
<tr>
<td>maxDomainCacheSize</td>
<td>3</td>
</tr>
<tr>
<td>workGroupSize</td>
<td>32</td>
</tr>
<tr>
<td>pathlines</td>
<td>0</td>
</tr>
<tr>
<td>pathlinesOverrideStartingTimeFlag</td>
<td>0</td>
</tr>
<tr>
<td>pathlinesOverrideStartingTime</td>
<td>0</td>
</tr>
<tr>
<td>pathlinesPeriod</td>
<td>0</td>
</tr>
<tr>
<td>pathlinesCMFE</td>
<td>POS_CMFE, CONN_CMFE</td>
</tr>
<tr>
<td>issueTerminationWarnings</td>
<td>1</td>
</tr>
<tr>
<td>issueStepsizeWarnings</td>
<td>1</td>
</tr>
<tr>
<td>issueStiffnessWarnings</td>
<td>1</td>
</tr>
<tr>
<td>issueCriticalPointsWarnings</td>
<td>1</td>
</tr>
<tr>
<td>criticalPointThreshold</td>
<td>0.001</td>
</tr>
<tr>
<td>Attribute</td>
<td>Default/Allowed Values</td>
</tr>
<tr>
<td>---------------</td>
<td>------------------------</td>
</tr>
<tr>
<td>printerName</td>
<td>&quot;&quot;</td>
</tr>
<tr>
<td>printProgram</td>
<td>&quot;lpr&quot;</td>
</tr>
<tr>
<td>documentName</td>
<td>&quot;untitled&quot;</td>
</tr>
<tr>
<td>creator</td>
<td>&quot;&quot;</td>
</tr>
<tr>
<td>numCopies</td>
<td>1</td>
</tr>
<tr>
<td>portrait</td>
<td>1</td>
</tr>
<tr>
<td>printColor</td>
<td>1</td>
</tr>
<tr>
<td>outputToFile</td>
<td>0</td>
</tr>
<tr>
<td>outputToFileName</td>
<td>&quot;untitled&quot;</td>
</tr>
<tr>
<td>pageSize</td>
<td>2</td>
</tr>
</tbody>
</table>

### 2.5.59 Process: \[ProcessAttributes()\]

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>pids</td>
<td>()</td>
</tr>
<tr>
<td>ppids</td>
<td>()</td>
</tr>
<tr>
<td>hosts</td>
<td>()</td>
</tr>
<tr>
<td>isParallel</td>
<td>0</td>
</tr>
<tr>
<td>memory</td>
<td>()</td>
</tr>
<tr>
<td>times</td>
<td>()</td>
</tr>
</tbody>
</table>

### 2.5.60 Project: \[ProjectAttributes()\]

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>projectionType</td>
<td>XYCartesian, ZYCartesian, XZCartesian, XRCylindrical, YRCylindrical, ZRCylindrical</td>
</tr>
<tr>
<td>vectorTransformMethod</td>
<td>AsDirection, None, AsPoint, AsDisplacement</td>
</tr>
</tbody>
</table>
2.5.61 Pseudocolor: \textit{PseudocolorAttributes()}

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>scaling</td>
<td>Linear, Log, Skew</td>
</tr>
<tr>
<td>skewFactor</td>
<td>1</td>
</tr>
<tr>
<td>limitsMode</td>
<td>OriginalData, CurrentPlot</td>
</tr>
<tr>
<td>minFlag</td>
<td>0</td>
</tr>
<tr>
<td>min</td>
<td>0</td>
</tr>
<tr>
<td>useBelowMinColor</td>
<td>0</td>
</tr>
<tr>
<td>belowMinColor</td>
<td>(0, 0, 0, 255)</td>
</tr>
<tr>
<td>maxFlag</td>
<td>0</td>
</tr>
<tr>
<td>max</td>
<td>1</td>
</tr>
<tr>
<td>useAboveMaxColor</td>
<td>0</td>
</tr>
<tr>
<td>aboveMaxColor</td>
<td>(0, 0, 0, 255)</td>
</tr>
<tr>
<td>centering</td>
<td>Natural, Nodal, Zonal</td>
</tr>
<tr>
<td>colorTableName</td>
<td>“hot”</td>
</tr>
<tr>
<td>invertColorTable</td>
<td>0</td>
</tr>
<tr>
<td>opacityType</td>
<td>FullyOpaque, ColorTable, Constant, Ramp, VariableRange</td>
</tr>
<tr>
<td>opacityVariable</td>
<td>“”</td>
</tr>
<tr>
<td>opacity</td>
<td>1</td>
</tr>
<tr>
<td>opacityVarMin</td>
<td>0</td>
</tr>
<tr>
<td>opacityVarMax</td>
<td>1</td>
</tr>
<tr>
<td>opacityVarMinFlag</td>
<td>0</td>
</tr>
<tr>
<td>opacityVarMaxFlag</td>
<td>0</td>
</tr>
<tr>
<td>pointSize</td>
<td>0.05</td>
</tr>
<tr>
<td>pointType</td>
<td>Point, Box, Axis, Icosahedron, Octahedron, Tetrahedron, SphereGeometry, Sphere</td>
</tr>
<tr>
<td>pointSizeVarEnabled</td>
<td>0</td>
</tr>
<tr>
<td>pointSizeVar</td>
<td>“default”</td>
</tr>
<tr>
<td>pointSizePixels</td>
<td>2</td>
</tr>
<tr>
<td>lineType</td>
<td>Line, Tube, Ribbon</td>
</tr>
<tr>
<td>lineWidth</td>
<td>0</td>
</tr>
<tr>
<td>tubeResolution</td>
<td>10</td>
</tr>
<tr>
<td>tubeRadiusSizeType</td>
<td>FractionOfBBox, Absolute</td>
</tr>
<tr>
<td>tubeRadiusAbsolute</td>
<td>0.125</td>
</tr>
<tr>
<td>tubeRadiusBBox</td>
<td>0.005</td>
</tr>
<tr>
<td>tubeRadiusVarEnabled</td>
<td>0</td>
</tr>
<tr>
<td>tubeRadiusVar</td>
<td>“”</td>
</tr>
<tr>
<td>tubeRadiusVarRatio</td>
<td>10</td>
</tr>
<tr>
<td>tailStyle</td>
<td>None, Spheres, Cones</td>
</tr>
<tr>
<td>headStyle</td>
<td>None, Spheres, Cones</td>
</tr>
<tr>
<td>endPointRadiusSizeType</td>
<td>FractionOfBBox, Absolute</td>
</tr>
<tr>
<td>endPointRadiusAbsolute</td>
<td>0.125</td>
</tr>
</tbody>
</table>
Table 2.8 – continued from previous page

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>endPointRadiusBBox</td>
<td>0.05</td>
</tr>
<tr>
<td>endPointResolution</td>
<td>10</td>
</tr>
<tr>
<td>endPointRatio</td>
<td>5</td>
</tr>
<tr>
<td>endPointRadiusVarEnabled</td>
<td>0</td>
</tr>
<tr>
<td>endPointRadiusVar</td>
<td>&quot;&quot;</td>
</tr>
<tr>
<td>endPointRadiusVarRatio</td>
<td>10</td>
</tr>
<tr>
<td>renderSurfaces</td>
<td>1</td>
</tr>
<tr>
<td>renderWireframe</td>
<td>0</td>
</tr>
<tr>
<td>renderPoints</td>
<td>0</td>
</tr>
<tr>
<td>smoothingLevel</td>
<td>0</td>
</tr>
<tr>
<td>legendFlag</td>
<td>1</td>
</tr>
<tr>
<td>lightingFlag</td>
<td>1</td>
</tr>
<tr>
<td>wireframeColor</td>
<td>(0, 0, 0, 0)</td>
</tr>
<tr>
<td>pointColor</td>
<td>(0, 0, 0, 0)</td>
</tr>
</tbody>
</table>

2.5.62 RadialResample:  

`RadialResampleAttributes()`

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>isFast</td>
<td>0</td>
</tr>
<tr>
<td>minTheta</td>
<td>0</td>
</tr>
<tr>
<td>maxTheta</td>
<td>90</td>
</tr>
<tr>
<td>deltaTheta</td>
<td>5</td>
</tr>
<tr>
<td>radius</td>
<td>0.5</td>
</tr>
<tr>
<td>deltaRadius</td>
<td>0.05</td>
</tr>
<tr>
<td>center</td>
<td>(0.5, 0.5, 0.5)</td>
</tr>
<tr>
<td>is3D</td>
<td>1</td>
</tr>
<tr>
<td>minAzimuth</td>
<td>0</td>
</tr>
<tr>
<td>maxAzimuth</td>
<td>180</td>
</tr>
<tr>
<td>deltaAzimuth</td>
<td>5</td>
</tr>
</tbody>
</table>
2.5.63 Reflect:  

ReflectAttributes()

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>octant</td>
<td>PXPYPZ, NXYPZ, PXNYPZ, NXNYPZ, PXPYNZ, NXPNZ, PXNYPNZ, NXPYNZ, NXNYPNZ, NXPNZYNZ</td>
</tr>
<tr>
<td>useXBoundary</td>
<td>1</td>
</tr>
<tr>
<td>specifiedX</td>
<td>0</td>
</tr>
<tr>
<td>useYBoundary</td>
<td>1</td>
</tr>
<tr>
<td>specifiedY</td>
<td>0</td>
</tr>
<tr>
<td>useZBoundary</td>
<td>1</td>
</tr>
<tr>
<td>specifiedZ</td>
<td>0</td>
</tr>
<tr>
<td>reflections</td>
<td>(1, 0, 1, 0, 0, 0, 0, 0)</td>
</tr>
<tr>
<td>planePoint</td>
<td>(0, 0, 0)</td>
</tr>
<tr>
<td>planeNormal</td>
<td>(0, 0, 0)</td>
</tr>
<tr>
<td>reflectType</td>
<td>Axis, Plane</td>
</tr>
</tbody>
</table>

2.5.64 Remap:  

RemapAttributes()

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>useExtents</td>
<td>1</td>
</tr>
<tr>
<td>startX</td>
<td>0</td>
</tr>
<tr>
<td>endX</td>
<td>1</td>
</tr>
<tr>
<td>cellsX</td>
<td>10</td>
</tr>
<tr>
<td>startY</td>
<td>0</td>
</tr>
<tr>
<td>endY</td>
<td>1</td>
</tr>
<tr>
<td>cellsY</td>
<td>10</td>
</tr>
<tr>
<td>is3D</td>
<td>1</td>
</tr>
<tr>
<td>startZ</td>
<td>0</td>
</tr>
<tr>
<td>endZ</td>
<td>1</td>
</tr>
<tr>
<td>cellsZ</td>
<td>10</td>
</tr>
<tr>
<td>variableType</td>
<td>intrinsic, extrinsic</td>
</tr>
</tbody>
</table>
2.5.65 Rendering:  

RenderingAttributes()

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>antialiasing</td>
<td>0</td>
</tr>
<tr>
<td>orderComposite</td>
<td>1</td>
</tr>
<tr>
<td>depthCompositeThreads</td>
<td>2</td>
</tr>
<tr>
<td>depthCompositeBlocking</td>
<td>65536</td>
</tr>
<tr>
<td>alphaCompositeThreads</td>
<td>2</td>
</tr>
<tr>
<td>alphaCompositeBlocking</td>
<td>65536</td>
</tr>
<tr>
<td>depthPeeling</td>
<td>0</td>
</tr>
<tr>
<td>occlusionRatio</td>
<td>0</td>
</tr>
<tr>
<td>numberOfPeels</td>
<td>16</td>
</tr>
<tr>
<td>multiresolutionMode</td>
<td>0</td>
</tr>
<tr>
<td>multiresolutionCellSize</td>
<td>0.002</td>
</tr>
<tr>
<td>geometryRepresentation</td>
<td>Surfaces, Wireframe, Points</td>
</tr>
<tr>
<td>stereoRendering</td>
<td>0</td>
</tr>
<tr>
<td>stereoType</td>
<td>CrystalEyes, RedBlue, Interlaced, RedGreen</td>
</tr>
<tr>
<td>notifyForEachRender</td>
<td>0</td>
</tr>
<tr>
<td>scalableActivationMode</td>
<td>Auto, Never, Always</td>
</tr>
<tr>
<td>scalableAutoThreshold</td>
<td>2000000</td>
</tr>
<tr>
<td>specularFlag</td>
<td>0</td>
</tr>
<tr>
<td>specularCoeff</td>
<td>0.6</td>
</tr>
<tr>
<td>specularPower</td>
<td>10</td>
</tr>
<tr>
<td>specularColor</td>
<td>(255, 255, 255, 255)</td>
</tr>
<tr>
<td>doShadowing</td>
<td>0</td>
</tr>
<tr>
<td>shadowStrength</td>
<td>0.5</td>
</tr>
<tr>
<td>doDepthCueing</td>
<td>0</td>
</tr>
<tr>
<td>depthCueingAutomatic</td>
<td>1</td>
</tr>
<tr>
<td>startCuePoint</td>
<td>(-10, 0, 0)</td>
</tr>
<tr>
<td>endCuePoint</td>
<td>(10, 0, 0)</td>
</tr>
<tr>
<td>compressionActivationMode</td>
<td>Never, Always, Auto</td>
</tr>
<tr>
<td>colorTexturingFlag</td>
<td>1</td>
</tr>
<tr>
<td>compactDomainsActivationMode</td>
<td>Never, Always, Auto</td>
</tr>
<tr>
<td>compactDomainsAutoThreshold</td>
<td>256</td>
</tr>
<tr>
<td>osprayRendering</td>
<td>0</td>
</tr>
<tr>
<td>ospraySPP</td>
<td>1</td>
</tr>
<tr>
<td>osprayAO</td>
<td>0</td>
</tr>
<tr>
<td>osprayShadows</td>
<td>0</td>
</tr>
</tbody>
</table>
### 2.5.66 Replicate:

ReplicateAttributes()

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>useUnitCellVectors</td>
<td>0</td>
</tr>
<tr>
<td>xVector</td>
<td>(1, 0, 0)</td>
</tr>
<tr>
<td>yVector</td>
<td>(0, 1, 0)</td>
</tr>
<tr>
<td>zVector</td>
<td>(0, 0, 1)</td>
</tr>
<tr>
<td>xReplications</td>
<td>1</td>
</tr>
<tr>
<td>yReplications</td>
<td>1</td>
</tr>
<tr>
<td>zReplications</td>
<td>1</td>
</tr>
<tr>
<td>mergeResults</td>
<td>1</td>
</tr>
<tr>
<td>replicateUnitCellAtoms</td>
<td>0</td>
</tr>
<tr>
<td>shiftPeriodicAtomOrigin</td>
<td>0</td>
</tr>
<tr>
<td>newPeriodicOrigin</td>
<td>(0, 0, 0)</td>
</tr>
</tbody>
</table>

### 2.5.67 Resample:

ResampleAttributes()

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>useExtents</td>
<td>1</td>
</tr>
<tr>
<td>startX</td>
<td>0</td>
</tr>
<tr>
<td>endX</td>
<td>1</td>
</tr>
<tr>
<td>samplesX</td>
<td>10</td>
</tr>
<tr>
<td>startY</td>
<td>0</td>
</tr>
<tr>
<td>endY</td>
<td>1</td>
</tr>
<tr>
<td>samplesY</td>
<td>10</td>
</tr>
<tr>
<td>is3D</td>
<td>1</td>
</tr>
<tr>
<td>startZ</td>
<td>0</td>
</tr>
<tr>
<td>endZ</td>
<td>1</td>
</tr>
<tr>
<td>samplesZ</td>
<td>10</td>
</tr>
<tr>
<td>tieResolver</td>
<td>random, largest, smallest</td>
</tr>
<tr>
<td>tieResolverVariable</td>
<td>“default”</td>
</tr>
<tr>
<td>defaultValue</td>
<td>0</td>
</tr>
<tr>
<td>distributedResample</td>
<td>1</td>
</tr>
<tr>
<td>cellCenteredOutput</td>
<td>0</td>
</tr>
</tbody>
</table>
### 2.5.68 Revolve: \texttt{RevolveAttributes()}

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>meshType</td>
<td>Auto, XY, RZ, ZR</td>
</tr>
<tr>
<td>autoAxis</td>
<td>1</td>
</tr>
<tr>
<td>axis</td>
<td>(1, 0, 0)</td>
</tr>
<tr>
<td>startAngle</td>
<td>0</td>
</tr>
<tr>
<td>stopAngle</td>
<td>360</td>
</tr>
<tr>
<td>steps</td>
<td>30</td>
</tr>
</tbody>
</table>

### 2.5.69 SPHResample: \texttt{SPHResampleAttributes()}

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>minX</td>
<td>0</td>
</tr>
<tr>
<td>maxX</td>
<td>1</td>
</tr>
<tr>
<td>xnum</td>
<td>10</td>
</tr>
<tr>
<td>minY</td>
<td>0</td>
</tr>
<tr>
<td>maxY</td>
<td>1</td>
</tr>
<tr>
<td>ynum</td>
<td>10</td>
</tr>
<tr>
<td>minZ</td>
<td>0</td>
</tr>
<tr>
<td>maxZ</td>
<td>1</td>
</tr>
<tr>
<td>znum</td>
<td>10</td>
</tr>
<tr>
<td>tensorSupportVariable</td>
<td>“H”</td>
</tr>
<tr>
<td>weightVariable</td>
<td>“mass”</td>
</tr>
<tr>
<td>RK</td>
<td>1</td>
</tr>
</tbody>
</table>

### 2.5.70 SaveWindow: \texttt{SaveWindowAttributes()}

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>outputToCurrentDirectory</td>
<td>1</td>
</tr>
<tr>
<td>outputDirectory</td>
<td>.</td>
</tr>
<tr>
<td>fileName</td>
<td>“visit”</td>
</tr>
<tr>
<td>family</td>
<td>1</td>
</tr>
<tr>
<td>format</td>
<td>PNG, BMP, CURVE, JPEG, OBJ, POSTSCRIPT, POVRAY, PPM, RGB, STL, TIFF, ULTRA, VTK, PLY, EXR</td>
</tr>
<tr>
<td>width</td>
<td>1024</td>
</tr>
<tr>
<td>height</td>
<td>1024</td>
</tr>
<tr>
<td>screenCapture</td>
<td>0</td>
</tr>
<tr>
<td>saveTiled</td>
<td>0</td>
</tr>
<tr>
<td>quality</td>
<td>80</td>
</tr>
<tr>
<td>progressive</td>
<td>0</td>
</tr>
<tr>
<td>binary</td>
<td>0</td>
</tr>
<tr>
<td>stereo</td>
<td>0</td>
</tr>
<tr>
<td>compression</td>
<td>None, PackBits, Jpeg, Deflate, LZW</td>
</tr>
<tr>
<td>forceMerge</td>
<td>0</td>
</tr>
<tr>
<td>resConstraint</td>
<td>ScreenProportions, NoConstraint, EqualWidthHeight</td>
</tr>
<tr>
<td>pixelData</td>
<td>1</td>
</tr>
<tr>
<td>advancedMultiWindowSave</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win1.position</td>
<td>(0, 0)</td>
</tr>
<tr>
<td>subWindowAtts.win1.size</td>
<td>(128, 128)</td>
</tr>
<tr>
<td>subWindowAtts.win1.layer</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win1.transparency</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win1.omitWindow</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win2.position</td>
<td>(0, 0)</td>
</tr>
<tr>
<td>subWindowAtts.win2.size</td>
<td>(128, 128)</td>
</tr>
<tr>
<td>subWindowAtts.win2.layer</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win2.transparency</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win2.omitWindow</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win3.position</td>
<td>(0, 0)</td>
</tr>
<tr>
<td>subWindowAtts.win3.size</td>
<td>(128, 128)</td>
</tr>
<tr>
<td>subWindowAtts.win3.layer</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win3.transparency</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win3.omitWindow</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win4.position</td>
<td>(0, 0)</td>
</tr>
<tr>
<td>subWindowAtts.win4.size</td>
<td>(128, 128)</td>
</tr>
<tr>
<td>subWindowAtts.win4.layer</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win4.transparency</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win4.omitWindow</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win5.position</td>
<td>(0, 0)</td>
</tr>
<tr>
<td>subWindowAtts.win5.size</td>
<td>(128, 128)</td>
</tr>
<tr>
<td>subWindowAtts.win5.layer</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win5.transparency</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win5.omitWindow</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win6.position</td>
<td>(0, 0)</td>
</tr>
<tr>
<td>subWindowAtts.win6.size</td>
<td>(128, 128)</td>
</tr>
<tr>
<td>subWindowAtts.win6.layer</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win6.transparency</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win6.omitWindow</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win7.position</td>
<td>(0, 0)</td>
</tr>
</tbody>
</table>

Continued on next page
<table>
<thead>
<tr>
<th>subWindowAtts.win7.size</th>
<th>(128, 128)</th>
</tr>
</thead>
<tbody>
<tr>
<td>subWindowAtts.win7.layer</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win7.transparency</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win7.omitWindow</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win8.position</td>
<td>(0, 0)</td>
</tr>
<tr>
<td>subWindowAtts.win8.size</td>
<td>(128, 128)</td>
</tr>
<tr>
<td>subWindowAtts.win8.layer</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win8.transparency</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win8.omitWindow</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win9.position</td>
<td>(0, 0)</td>
</tr>
<tr>
<td>subWindowAtts.win9.size</td>
<td>(128, 128)</td>
</tr>
<tr>
<td>subWindowAtts.win9.layer</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win9.transparency</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win9.omitWindow</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win10.position</td>
<td>(0, 0)</td>
</tr>
<tr>
<td>subWindowAtts.win10.size</td>
<td>(128, 128)</td>
</tr>
<tr>
<td>subWindowAtts.win10.layer</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win10.transparency</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win10.omitWindow</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win11.position</td>
<td>(0, 0)</td>
</tr>
<tr>
<td>subWindowAtts.win11.size</td>
<td>(128, 128)</td>
</tr>
<tr>
<td>subWindowAtts.win11.layer</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win11.transparency</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win11.omitWindow</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win12.position</td>
<td>(0, 0)</td>
</tr>
<tr>
<td>subWindowAtts.win12.size</td>
<td>(128, 128)</td>
</tr>
<tr>
<td>subWindowAtts.win12.layer</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win12.transparency</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win12.omitWindow</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win13.position</td>
<td>(0, 0)</td>
</tr>
<tr>
<td>subWindowAtts.win13.size</td>
<td>(128, 128)</td>
</tr>
<tr>
<td>subWindowAtts.win13.layer</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win13.transparency</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win13.omitWindow</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win14.position</td>
<td>(0, 0)</td>
</tr>
<tr>
<td>subWindowAtts.win14.size</td>
<td>(128, 128)</td>
</tr>
<tr>
<td>subWindowAtts.win14.layer</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win14.transparency</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win14.omitWindow</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win15.position</td>
<td>(0, 0)</td>
</tr>
<tr>
<td>subWindowAtts.win15.size</td>
<td>(128, 128)</td>
</tr>
<tr>
<td>subWindowAtts.win15.layer</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win15.transparency</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win15.omitWindow</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win16.position</td>
<td>(0, 0)</td>
</tr>
<tr>
<td>subWindowAtts.win16.size</td>
<td>(128, 128)</td>
</tr>
<tr>
<td>subWindowAtts.win16.layer</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win16.transparency</td>
<td>0</td>
</tr>
<tr>
<td>subWindowAtts.win16.omitWindow</td>
<td>0</td>
</tr>
<tr>
<td>opts.types</td>
<td>()</td>
</tr>
</tbody>
</table>
### 2.5.71 Scatter: \( \text{ScatterAttributes()} \)

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>var1</td>
<td>“default”</td>
</tr>
<tr>
<td>var1Role</td>
<td>Coordinate0, Coordinate1, Coordinate2, Color, None</td>
</tr>
<tr>
<td>var1MinFlag</td>
<td>0</td>
</tr>
<tr>
<td>var1MaxFlag</td>
<td>0</td>
</tr>
<tr>
<td>var1Min</td>
<td>0</td>
</tr>
<tr>
<td>var1Max</td>
<td>1</td>
</tr>
<tr>
<td>var1Scaling</td>
<td>Linear, Log, Skew</td>
</tr>
<tr>
<td>var1SkewFactor</td>
<td>1</td>
</tr>
<tr>
<td>var2Role</td>
<td>Coordinate1, Coordinate0, Coordinate2, Color, None</td>
</tr>
<tr>
<td>var2</td>
<td>“default”</td>
</tr>
<tr>
<td>var2MinFlag</td>
<td>0</td>
</tr>
<tr>
<td>var2MaxFlag</td>
<td>0</td>
</tr>
<tr>
<td>var2Min</td>
<td>0</td>
</tr>
<tr>
<td>var2Max</td>
<td>1</td>
</tr>
<tr>
<td>var2Scaling</td>
<td>Linear, Log, Skew</td>
</tr>
<tr>
<td>var2SkewFactor</td>
<td>1</td>
</tr>
<tr>
<td>var3Role</td>
<td>None, Coordinate0, Coordinate1, Coordinate2, Color</td>
</tr>
<tr>
<td>var3</td>
<td>“default”</td>
</tr>
<tr>
<td>var3MinFlag</td>
<td>0</td>
</tr>
<tr>
<td>var3MaxFlag</td>
<td>0</td>
</tr>
<tr>
<td>var3Min</td>
<td>0</td>
</tr>
<tr>
<td>var3Max</td>
<td>1</td>
</tr>
<tr>
<td>var3Scaling</td>
<td>Linear, Log, Skew</td>
</tr>
<tr>
<td>var3SkewFactor</td>
<td>1</td>
</tr>
<tr>
<td>var4Role</td>
<td>None, Coordinate0, Coordinate1, Coordinate2, Color</td>
</tr>
<tr>
<td>var4</td>
<td>“default”</td>
</tr>
<tr>
<td>var4MinFlag</td>
<td>0</td>
</tr>
<tr>
<td>var4MaxFlag</td>
<td>0</td>
</tr>
<tr>
<td>var4Min</td>
<td>0</td>
</tr>
<tr>
<td>var4Max</td>
<td>1</td>
</tr>
<tr>
<td>var4Scaling</td>
<td>Linear, Log, Skew</td>
</tr>
<tr>
<td>var4SkewFactor</td>
<td>1</td>
</tr>
<tr>
<td>pointSize</td>
<td>0.05</td>
</tr>
</tbody>
</table>

Continued on next page
Table 2.11 – continued from previous page

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>originType</td>
<td>Intercept, Point, Percent, Zone, Node</td>
</tr>
<tr>
<td>originPoint</td>
<td>(0, 0, 0)</td>
</tr>
<tr>
<td>originIntercept</td>
<td>0</td>
</tr>
<tr>
<td>originPercent</td>
<td>0</td>
</tr>
<tr>
<td>originZone</td>
<td>0</td>
</tr>
<tr>
<td>originNode</td>
<td>0</td>
</tr>
<tr>
<td>normal</td>
<td>(0, -1, 0)</td>
</tr>
<tr>
<td>axisType</td>
<td>YAxis, XAxis, ZAxis, Arbitrary, ThetaPhi</td>
</tr>
<tr>
<td>upAxis</td>
<td>(0, 0, 1)</td>
</tr>
<tr>
<td>project2d</td>
<td>1</td>
</tr>
<tr>
<td>interactive</td>
<td>1</td>
</tr>
<tr>
<td>flip</td>
<td>0</td>
</tr>
<tr>
<td>originZoneDomain</td>
<td>0</td>
</tr>
<tr>
<td>originNodeDomain</td>
<td>0</td>
</tr>
<tr>
<td>meshName</td>
<td>“default”</td>
</tr>
<tr>
<td>theta</td>
<td>0</td>
</tr>
<tr>
<td>phi</td>
<td>0</td>
</tr>
</tbody>
</table>

2.5.72 Slice: 

`SliceAttributes()`
### 2.5.73 SmoothOperator:  
*SmoothOperatorAttributes()*

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>numIterations</td>
<td>20</td>
</tr>
<tr>
<td>relaxationFactor</td>
<td>0.01</td>
</tr>
<tr>
<td>convergence</td>
<td>0</td>
</tr>
<tr>
<td>maintainFeatures</td>
<td>1</td>
</tr>
<tr>
<td>featureAngle</td>
<td>45</td>
</tr>
<tr>
<td>edgeAngle</td>
<td>15</td>
</tr>
<tr>
<td>smoothBoundaries</td>
<td>0</td>
</tr>
</tbody>
</table>

### 2.5.74 SphereSlice:  
*SphereSliceAttributes()*

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>origin</td>
<td>(0, 0, 0)</td>
</tr>
<tr>
<td>radius</td>
<td>1</td>
</tr>
</tbody>
</table>

### 2.5.75 Spreadsheet:  
*SpreadsheetAttributes()*
### Attribute Table

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>subsetName</td>
<td>“Whole”</td>
</tr>
<tr>
<td>formatString</td>
<td>“%1.6f”</td>
</tr>
<tr>
<td>useColorTable</td>
<td>0</td>
</tr>
<tr>
<td>colorTableName</td>
<td>“Default”</td>
</tr>
<tr>
<td>showTracerPlane</td>
<td>1</td>
</tr>
<tr>
<td>tracerColor</td>
<td>(255, 0, 0, 150)</td>
</tr>
<tr>
<td>normal</td>
<td>Z, X, Y</td>
</tr>
<tr>
<td>sliceIndex</td>
<td>0</td>
</tr>
<tr>
<td>spreadsheetFont</td>
<td>“Courier,12,-1,5,50,0,0,0,0,0”</td>
</tr>
<tr>
<td>showPatchOutline</td>
<td>1</td>
</tr>
<tr>
<td>showCurrentCellOutline</td>
<td>0</td>
</tr>
<tr>
<td>currentPickType</td>
<td>0</td>
</tr>
<tr>
<td>currentPickLetter</td>
<td>“”</td>
</tr>
<tr>
<td>pastPickLetters</td>
<td>()</td>
</tr>
</tbody>
</table>

### 2.5.76 Stagger: `StaggerAttributes()`

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>offsetX</td>
<td>0</td>
</tr>
<tr>
<td>offsetY</td>
<td>0</td>
</tr>
<tr>
<td>offsetZ</td>
<td>0</td>
</tr>
</tbody>
</table>

### 2.5.77 StatisticalTrends: `StatisticalTrendsAttributes()`
### 2.5.78 SubdivideQuads:

**SubdivideQuadsAttributes()**

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>threshold</td>
<td>0.500002</td>
</tr>
<tr>
<td>maxSubdivs</td>
<td>4</td>
</tr>
<tr>
<td>fanOutPoints</td>
<td>1</td>
</tr>
<tr>
<td>doTriangles</td>
<td>0</td>
</tr>
<tr>
<td>variable</td>
<td>“default”</td>
</tr>
</tbody>
</table>

### 2.5.79 Subset:

**SubsetAttributes()**
<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>colorType</td>
<td>ColorByMultipleColors, ColorBySingleColor, ColorByColorTable</td>
</tr>
<tr>
<td>colorTableName</td>
<td>“Default”</td>
</tr>
<tr>
<td>invertColorTable</td>
<td>0</td>
</tr>
<tr>
<td>legendFlag</td>
<td>1</td>
</tr>
<tr>
<td>lineWidth</td>
<td>0</td>
</tr>
<tr>
<td>singleColor</td>
<td>(0, 0, 0, 255)</td>
</tr>
<tr>
<td>subsetNames</td>
<td>()</td>
</tr>
<tr>
<td>opacity</td>
<td>1</td>
</tr>
<tr>
<td>wireframe</td>
<td>0</td>
</tr>
<tr>
<td>drawInternal</td>
<td>0</td>
</tr>
<tr>
<td>smoothingLevel</td>
<td>0</td>
</tr>
<tr>
<td>pointSize</td>
<td>0.05</td>
</tr>
<tr>
<td>pointType</td>
<td>Point, Box, Icosahedron, Octahedron, Tetrahedron, SphereGeometry, Sphere</td>
</tr>
<tr>
<td>pointSizeVarEnabled</td>
<td>0</td>
</tr>
<tr>
<td>pointSizeVar</td>
<td>“default”</td>
</tr>
<tr>
<td>pointSizePixels</td>
<td>2</td>
</tr>
</tbody>
</table>

**2.5.80 SurfaceNormal:**

SurfaceNormalAttributes()

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>centering</td>
<td>Point, Cell</td>
</tr>
</tbody>
</table>

**2.5.81 Tensor:**

TensorAttributes()
### 2.5.82 ThreeSlice: *ThreeSliceAttributes()

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>x</td>
<td>0</td>
</tr>
<tr>
<td>y</td>
<td>0</td>
</tr>
<tr>
<td>z</td>
<td>0</td>
</tr>
<tr>
<td>interactive</td>
<td>1</td>
</tr>
</tbody>
</table>

### 2.5.83 Threshold: *ThresholdAttributes()

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>useStride</td>
<td>0</td>
</tr>
<tr>
<td>stride</td>
<td>1</td>
</tr>
<tr>
<td>nTensors</td>
<td>400</td>
</tr>
<tr>
<td>scale</td>
<td>0.25</td>
</tr>
<tr>
<td>scaleByMagnitude</td>
<td>1</td>
</tr>
<tr>
<td>autoScale</td>
<td>1</td>
</tr>
<tr>
<td>colorByEigenvalues</td>
<td>1</td>
</tr>
<tr>
<td>useLegend</td>
<td>1</td>
</tr>
<tr>
<td>tensorColor</td>
<td>(0, 0, 0, 255)</td>
</tr>
<tr>
<td>colorTableName</td>
<td>“Default”</td>
</tr>
<tr>
<td>invertColorTable</td>
<td>0</td>
</tr>
<tr>
<td>Attribute</td>
<td>Default/Allowed Values</td>
</tr>
<tr>
<td>--------------------</td>
<td>------------------------</td>
</tr>
<tr>
<td>outputMeshType</td>
<td>0</td>
</tr>
<tr>
<td>boundsInputType</td>
<td>0</td>
</tr>
<tr>
<td>listedVarNames</td>
<td>(&quot;default&quot;)</td>
</tr>
<tr>
<td>zonePortions</td>
<td>()</td>
</tr>
<tr>
<td>lowerBounds</td>
<td>()</td>
</tr>
<tr>
<td>upperBounds</td>
<td>()</td>
</tr>
<tr>
<td>defaultVarName</td>
<td>&quot;default&quot;</td>
</tr>
<tr>
<td>defaultVarIsScalar</td>
<td>0</td>
</tr>
<tr>
<td>boundsRange</td>
<td>()</td>
</tr>
</tbody>
</table>

### 2.5.84 Transform: $\text{TransformAttributes()}$

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>doRotate</td>
<td>0</td>
</tr>
<tr>
<td>rotateOrigin</td>
<td>(0, 0, 0)</td>
</tr>
<tr>
<td>rotateAxis</td>
<td>(0, 0, 1)</td>
</tr>
<tr>
<td>rotateAmount</td>
<td>0</td>
</tr>
<tr>
<td>rotateType</td>
<td>Deg, Rad</td>
</tr>
<tr>
<td>doScale</td>
<td>0</td>
</tr>
<tr>
<td>scaleOrigin</td>
<td>(0, 0, 0)</td>
</tr>
<tr>
<td>scaleX</td>
<td>1</td>
</tr>
<tr>
<td>scaleY</td>
<td>1</td>
</tr>
<tr>
<td>scaleZ</td>
<td>1</td>
</tr>
<tr>
<td>doTranslate</td>
<td>0</td>
</tr>
<tr>
<td>translateX</td>
<td>0</td>
</tr>
<tr>
<td>translateY</td>
<td>0</td>
</tr>
<tr>
<td>translateZ</td>
<td>0</td>
</tr>
<tr>
<td>transformType</td>
<td>Similarity, Coordinate, Linear</td>
</tr>
<tr>
<td>inputCoordSys</td>
<td>Cartesian, Cylindrical, Spherical</td>
</tr>
<tr>
<td>outputCoordSys</td>
<td>Spherical, Cartesian, Cylindrical</td>
</tr>
<tr>
<td>continuousPhi</td>
<td>0</td>
</tr>
<tr>
<td>m00</td>
<td>1</td>
</tr>
<tr>
<td>m01</td>
<td>0</td>
</tr>
<tr>
<td>m02</td>
<td>0</td>
</tr>
<tr>
<td>m03</td>
<td>0</td>
</tr>
<tr>
<td>m10</td>
<td>0</td>
</tr>
<tr>
<td>m11</td>
<td>1</td>
</tr>
<tr>
<td>m12</td>
<td>0</td>
</tr>
<tr>
<td>m13</td>
<td>0</td>
</tr>
</tbody>
</table>
Table 2.12 – continued from previous page

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>m20</td>
<td>0</td>
</tr>
<tr>
<td>m21</td>
<td>0</td>
</tr>
<tr>
<td>m22</td>
<td>1</td>
</tr>
<tr>
<td>m23</td>
<td>0</td>
</tr>
<tr>
<td>m30</td>
<td>0</td>
</tr>
<tr>
<td>m31</td>
<td>0</td>
</tr>
<tr>
<td>m32</td>
<td>0</td>
</tr>
<tr>
<td>m33</td>
<td>1</td>
</tr>
<tr>
<td>invertLinearTransform</td>
<td>0</td>
</tr>
<tr>
<td>vectorTransformMethod</td>
<td>AsDirection, None, AsPoint, AsDisplacement</td>
</tr>
<tr>
<td>transformVectors</td>
<td>1</td>
</tr>
</tbody>
</table>

2.5.85 TriangulateRegularPoints: \( \text{TriangulateRegularPointsAttributes()} \)

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>useXGridSpacing</td>
<td>0</td>
</tr>
<tr>
<td>xGridSpacing</td>
<td>1</td>
</tr>
<tr>
<td>useYGridSpacing</td>
<td>0</td>
</tr>
<tr>
<td>yGridSpacing</td>
<td>1</td>
</tr>
</tbody>
</table>

2.5.86 Truecolor: \( \text{TruecolorAttributes()} \)

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>opacity</td>
<td>1</td>
</tr>
<tr>
<td>lightingFlag</td>
<td>1</td>
</tr>
</tbody>
</table>

2.5. Attribute Reference
2.5.87 Tube: **TubeAttributes()**

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>scaleByVarFlag</td>
<td>0</td>
</tr>
<tr>
<td>tubeRadiusType</td>
<td>FractionOfBBox, Absolute</td>
</tr>
<tr>
<td>radiusFractionBBox</td>
<td>0.01</td>
</tr>
<tr>
<td>radiusAbsolute</td>
<td>1</td>
</tr>
<tr>
<td>scaleVariable</td>
<td>“default”</td>
</tr>
<tr>
<td>fineness</td>
<td>5</td>
</tr>
<tr>
<td>capping</td>
<td>0</td>
</tr>
</tbody>
</table>

2.5.88 Vector: **VectorAttributes()**
<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>glyphLocation</td>
<td>AdaptsToMeshResolution, UniformInSpace</td>
</tr>
<tr>
<td>useStride</td>
<td>0</td>
</tr>
<tr>
<td>stride</td>
<td>1</td>
</tr>
<tr>
<td>nVectors</td>
<td>400</td>
</tr>
<tr>
<td>lineWidth</td>
<td>0</td>
</tr>
<tr>
<td>scale</td>
<td>0.25</td>
</tr>
<tr>
<td>scaleByMagnitude</td>
<td>1</td>
</tr>
<tr>
<td>autoScale</td>
<td>1</td>
</tr>
<tr>
<td>headSize</td>
<td>0.25</td>
</tr>
<tr>
<td>headOn</td>
<td>1</td>
</tr>
<tr>
<td>colorByMag</td>
<td>1</td>
</tr>
<tr>
<td>useLegend</td>
<td>1</td>
</tr>
<tr>
<td>vectorColor</td>
<td>(0, 0, 0, 255)</td>
</tr>
<tr>
<td>colorTableName</td>
<td>“Default”</td>
</tr>
<tr>
<td>invertColorTable</td>
<td>0</td>
</tr>
<tr>
<td>vectorOrigin</td>
<td>Tail, Head, Middle</td>
</tr>
<tr>
<td>minFlag</td>
<td>0</td>
</tr>
<tr>
<td>maxFlag</td>
<td>0</td>
</tr>
<tr>
<td>limitsMode</td>
<td>OriginalData, CurrentPlot</td>
</tr>
<tr>
<td>min</td>
<td>0</td>
</tr>
<tr>
<td>max</td>
<td>1</td>
</tr>
<tr>
<td>lineStem</td>
<td>Line, Cylinder</td>
</tr>
<tr>
<td>geometryQuality</td>
<td>Fast, High</td>
</tr>
<tr>
<td>stemWidth</td>
<td>0.08</td>
</tr>
<tr>
<td>origOnly</td>
<td>1</td>
</tr>
<tr>
<td>glyphType</td>
<td>Arrow, Ellipsoid</td>
</tr>
<tr>
<td>animationStep</td>
<td>0</td>
</tr>
</tbody>
</table>

### 2.5.89 View: ViewAttributes()
### 2.5.90 View2D:

**View2DAttributes()**

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>windowCoords</td>
<td>(0, 1, 0, 1)</td>
</tr>
<tr>
<td>viewportCoords</td>
<td>(0.2, 0.95, 0.15, 0.95)</td>
</tr>
<tr>
<td>fullFrameActivationMode</td>
<td>Auto, On, Off</td>
</tr>
<tr>
<td>fullFrameAutoThreshold</td>
<td>100</td>
</tr>
<tr>
<td>xScale</td>
<td>LINEAR, LOG</td>
</tr>
<tr>
<td>yScale</td>
<td>LINEAR, LOG</td>
</tr>
<tr>
<td>windowValid</td>
<td>0</td>
</tr>
</tbody>
</table>

### 2.5.91 View3D:

**View3DAttributes()**

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>windowNormal</td>
<td>(0, 0, 1)</td>
</tr>
<tr>
<td>focus</td>
<td>(0, 0, 0)</td>
</tr>
<tr>
<td>viewUp</td>
<td>(0, 1, 0)</td>
</tr>
<tr>
<td>viewAngle</td>
<td>30</td>
</tr>
<tr>
<td>setScale</td>
<td>0</td>
</tr>
<tr>
<td>parallelScale</td>
<td>1</td>
</tr>
<tr>
<td>nearPlane</td>
<td>0.001</td>
</tr>
<tr>
<td>farPlane</td>
<td>100</td>
</tr>
<tr>
<td>imagePan</td>
<td>(0, 0)</td>
</tr>
<tr>
<td>imageZoom</td>
<td>1</td>
</tr>
<tr>
<td>perspective</td>
<td>1</td>
</tr>
<tr>
<td>windowCoords</td>
<td>(0, 0, 1, 1)</td>
</tr>
<tr>
<td>viewportCoords</td>
<td>(0.1, 0.1, 0.9, 0.9)</td>
</tr>
<tr>
<td>eyeAngle</td>
<td>2</td>
</tr>
<tr>
<td>Attribute</td>
<td>Default/Allowed Values</td>
</tr>
<tr>
<td>----------------</td>
<td>------------------------</td>
</tr>
<tr>
<td>viewNormal</td>
<td>(0, 0, 1)</td>
</tr>
<tr>
<td>focus</td>
<td>(0, 0, 0)</td>
</tr>
<tr>
<td>viewUp</td>
<td>(0, 1, 0)</td>
</tr>
<tr>
<td>viewAngle</td>
<td>30</td>
</tr>
<tr>
<td>parallelScale</td>
<td>0.5</td>
</tr>
<tr>
<td>nearPlane</td>
<td>-0.5</td>
</tr>
<tr>
<td>farPlane</td>
<td>0.5</td>
</tr>
<tr>
<td>imagePan</td>
<td>(0, 0)</td>
</tr>
<tr>
<td>imageZoom</td>
<td>1</td>
</tr>
<tr>
<td>perspective</td>
<td>1</td>
</tr>
<tr>
<td>eyeAngle</td>
<td>2</td>
</tr>
<tr>
<td>centerOfRotationSet</td>
<td>0</td>
</tr>
<tr>
<td>centerOfRotation</td>
<td>(0, 0, 0)</td>
</tr>
<tr>
<td>axis3DScaleFlag</td>
<td>0</td>
</tr>
<tr>
<td>axis3DScales</td>
<td>(1, 1, 1)</td>
</tr>
<tr>
<td>shear</td>
<td>(0, 0, 1)</td>
</tr>
<tr>
<td>windowValid</td>
<td>0</td>
</tr>
</tbody>
</table>

### 2.5.92 ViewAxisArray: 
*ViewAxisArrayAttributes()*

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>domainCoords</td>
<td>(0, 1)</td>
</tr>
<tr>
<td>rangeCoords</td>
<td>(0, 1)</td>
</tr>
<tr>
<td>viewportCoords</td>
<td>(0.15, 0.9, 0.1, 0.85)</td>
</tr>
</tbody>
</table>

### 2.5.93 ViewCurve: 
*ViewCurveAttributes()*
### Attribute Definitions

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>domainCoords</td>
<td>(0, 1)</td>
</tr>
<tr>
<td>rangeCoords</td>
<td>(0, 1)</td>
</tr>
<tr>
<td>viewportCoords</td>
<td>(0.2, 0.95, 0.15, 0.95)</td>
</tr>
<tr>
<td>domainScale</td>
<td>LINEAR, LOG</td>
</tr>
<tr>
<td>rangeScale</td>
<td>LINEAR, LOG</td>
</tr>
</tbody>
</table>

#### 2.5.94 Volume: VolumeAttributes()

### Attribute Definitions

<table>
<thead>
<tr>
<th>Attribute</th>
<th>Default/Allowed Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>osprayShadowsEnabledFlag</td>
<td>0</td>
</tr>
<tr>
<td>osprayUseGridAcceleratorFlag</td>
<td>0</td>
</tr>
<tr>
<td>osprayPreIntegrationFlag</td>
<td>0</td>
</tr>
<tr>
<td>ospraySingleShadeFlag</td>
<td>0</td>
</tr>
<tr>
<td>osprayOneSidedLightingFlag</td>
<td>0</td>
</tr>
<tr>
<td>osprayAoTransparencyEnabledFlag</td>
<td>0</td>
</tr>
<tr>
<td>ospraySpp</td>
<td>1</td>
</tr>
<tr>
<td>osprayAoSamples</td>
<td>0</td>
</tr>
<tr>
<td>osprayAoDistance</td>
<td>100000</td>
</tr>
<tr>
<td>osprayMinContribution</td>
<td>0.001</td>
</tr>
<tr>
<td>legendFlag</td>
<td>1</td>
</tr>
<tr>
<td>lightingFlag</td>
<td>1</td>
</tr>
<tr>
<td>colorControlPoints.GetControlPoints(0).colors</td>
<td>(0, 0, 255, 255)</td>
</tr>
<tr>
<td>colorControlPoints.GetControlPoints(0).position</td>
<td>0</td>
</tr>
<tr>
<td>colorControlPoints.GetControlPoints(1).colors</td>
<td>(0, 255, 255, 255)</td>
</tr>
<tr>
<td>colorControlPoints.GetControlPoints(1).position</td>
<td>0.25</td>
</tr>
<tr>
<td>colorControlPoints.GetControlPoints(2).colors</td>
<td>(0, 255, 0, 255)</td>
</tr>
<tr>
<td>colorControlPoints.GetControlPoints(2).position</td>
<td>0.5</td>
</tr>
<tr>
<td>colorControlPoints.GetControlPoints(3).colors</td>
<td>(255, 255, 0, 255)</td>
</tr>
<tr>
<td>colorControlPoints.GetControlPoints(3).position</td>
<td>0.75</td>
</tr>
<tr>
<td>colorControlPoints.GetControlPoints(4).colors</td>
<td>(255, 0, 0, 255)</td>
</tr>
<tr>
<td>colorControlPoints.GetControlPoints(4).position</td>
<td>1</td>
</tr>
<tr>
<td>colorControlPoints.smoothing</td>
<td>Linear, None, CubicSpline</td>
</tr>
<tr>
<td>colorControlPoints.equalSpacingFlag</td>
<td>0</td>
</tr>
<tr>
<td>colorControlPoints.discreteFlag</td>
<td>0</td>
</tr>
<tr>
<td>colorControlPoints.categoryFlag</td>
<td>&quot;&quot;</td>
</tr>
<tr>
<td>opacityAttenuation</td>
<td>1</td>
</tr>
<tr>
<td>opacityMode</td>
<td>FreeformMode, GaussianMode, ColorTableMode</td>
</tr>
</tbody>
</table>

controlPoints does not contain any GaussianControlPoint objects.

<p>| resampleFlag                           | 1                      |</p>
<table>
<thead>
<tr>
<th>EventName</th>
<th>ArgCount</th>
</tr>
</thead>
<tbody>
<tr>
<td>AMRStitchCellAttributes</td>
<td>1</td>
</tr>
<tr>
<td>ActivateDatabaseRPC</td>
<td>1</td>
</tr>
</tbody>
</table>

2.6 VisIt CLI Events

This chapter shows a table with all events that the VisIt GUI could potentially generate. Different plugins create different events, so the list will depend on the user configuration. The list in this section is generated from a call to the `GetCallbackNames()` function and will therefore list just the events that are applicable to the user that generates this documentation.

The list is alphabetically ordered. The left column, labeled `EventName` displays each event or callback name. The right column, labeled `ArgCount` displays the result of calling `GetCallbackArgumentCount(EventName)` for the corresponding event, which returns the number of arguments a callback function for that event should accept.

Continued on next page
<table>
<thead>
<tr>
<th>Method</th>
<th>Calls</th>
</tr>
</thead>
<tbody>
<tr>
<td>AddAnnotationObjectRPC</td>
<td>2</td>
</tr>
<tr>
<td>AddEmbeddedPlotRPC</td>
<td>1</td>
</tr>
<tr>
<td>AddInitializedOperatorRPC</td>
<td>1</td>
</tr>
<tr>
<td>AddOperatorRPC</td>
<td>2</td>
</tr>
<tr>
<td>AddPlotRPC</td>
<td>2</td>
</tr>
<tr>
<td>AddWindowRPC</td>
<td>0</td>
</tr>
<tr>
<td>AlterDatabaseCorrelationRPC</td>
<td>4</td>
</tr>
<tr>
<td>AnimationAttributes</td>
<td>1</td>
</tr>
<tr>
<td>AnimationPlayRPC</td>
<td>0</td>
</tr>
<tr>
<td>AnimationReversePlayRPC</td>
<td>0</td>
</tr>
<tr>
<td>AnimationSetNFramesRPC</td>
<td>1</td>
</tr>
<tr>
<td>AnimationStopRPC</td>
<td>0</td>
</tr>
<tr>
<td>AnnotationAttributes</td>
<td>1</td>
</tr>
<tr>
<td>ApplyNamedSelectionRPC</td>
<td>1</td>
</tr>
<tr>
<td>AXISAlignedSlice4DAttributes</td>
<td>1</td>
</tr>
<tr>
<td>BoundaryAttributes</td>
<td>1</td>
</tr>
<tr>
<td>BoundaryOpAttributes</td>
<td>1</td>
</tr>
<tr>
<td>BoxAttributes</td>
<td>1</td>
</tr>
<tr>
<td>CartographicProjectionAttributes</td>
<td>1</td>
</tr>
<tr>
<td>ChangeActivePlotsVarRPC</td>
<td>1</td>
</tr>
<tr>
<td>CheckForNewStatesRPC</td>
<td>1</td>
</tr>
<tr>
<td>ChooseCenterOfRotationRPC</td>
<td>2</td>
</tr>
<tr>
<td>ClearAllWindowsRPC</td>
<td>0</td>
</tr>
<tr>
<td>ClearCacheForAllEnginesRPC</td>
<td>0</td>
</tr>
<tr>
<td>ClearCacheRPC</td>
<td>2</td>
</tr>
<tr>
<td>ClearPickPointsRPC</td>
<td>0</td>
</tr>
<tr>
<td>ClearRefLinesRPC</td>
<td>0</td>
</tr>
<tr>
<td>ClearViewKeyframesRPC</td>
<td>0</td>
</tr>
<tr>
<td>ClearWindowRPC</td>
<td>1</td>
</tr>
<tr>
<td>ClipAttributes</td>
<td>1</td>
</tr>
<tr>
<td>CloneWindowRPC</td>
<td>0</td>
</tr>
<tr>
<td>CloseComputeEngineRPC</td>
<td>2</td>
</tr>
<tr>
<td>CloseDatabaseRPC</td>
<td>1</td>
</tr>
<tr>
<td>CloseRPC</td>
<td>0</td>
</tr>
<tr>
<td>ColorTableAttributes</td>
<td>1</td>
</tr>
<tr>
<td>ConeAttributes</td>
<td>1</td>
</tr>
<tr>
<td>ConnectToMetaDataServerRPC</td>
<td>2</td>
</tr>
<tr>
<td>ConnectedComponentsAttributes</td>
<td>1</td>
</tr>
<tr>
<td>ConstructDataBinningAttributes</td>
<td>1</td>
</tr>
<tr>
<td>ConstructDataBinningRPC</td>
<td>0</td>
</tr>
<tr>
<td>ContourAttributes</td>
<td>1</td>
</tr>
<tr>
<td>CoordSwapAttributes</td>
<td>1</td>
</tr>
<tr>
<td>CopyActivePlotsRPC</td>
<td>0</td>
</tr>
<tr>
<td>CopyAnnotationsToWindowRPC</td>
<td>2</td>
</tr>
<tr>
<td>CopyLightingToWindowRPC</td>
<td>2</td>
</tr>
<tr>
<td>CopyPlotsToWindowRPC</td>
<td>2</td>
</tr>
<tr>
<td>CopyViewToWindowRPC</td>
<td>2</td>
</tr>
<tr>
<td>CreateBondsAttributes</td>
<td>1</td>
</tr>
<tr>
<td>CreateDatabaseCorrelationRPC</td>
<td>4</td>
</tr>
<tr>
<td>CreateNamedSelectionRPC</td>
<td>1</td>
</tr>
</tbody>
</table>

Continued on next page
| CurveAttributes                           | 1 |
| CylinderAttributes                       | 1 |
| DDTConnectRPC                            | 1 |
| DDTFocusRPC                              | 1 |
| DataBinningAttributes                    | 1 |
| DatabaseMetaData                         | 1 |
| DeIconifyAllWindowsRPC                   | 0 |
| DeferExpressionAttributes                | 1 |
| DeleteActiveAnnotationObjectsRPC         | 0 |
| DeleteActivePlotsRPC                     | 0 |
| DeleteDatabaseCorrelationRPC             | 1 |
| DeleteNamedSelectionRPC                  | 1 |
| DeletePlotDatabaseKeyframeRPC            | 2 |
| DeletePlotKeyframeRPC                    | 2 |
| DeleteViewKeyframeRPC                    | 1 |
| DeleteWindowRPC                          | 0 |
| DemoteOperatorRPC                        | 1 |
| DetachRPC                                | 0 |
| DisableRedrawRPC                         | 0 |
| DisplaceAttributes                       | 1 |
| DrawPlotsRPC                             | 1 |
| DualMeshAttributes                       | 1 |
| EdgeAttributes                           | 1 |
| ElevateAttributes                        | 1 |
| EllipsoidSliceAttributes                 | 1 |
| EnableToolRPC                            | 2 |
| EnableToolbarRPC                         | 2 |
| ExplodeAttributes                        | 1 |
| ExportColorTableRPC                      | 1 |
| ExportDBAttributes                       | 1 |
| ExportDBRPC                              | 0 |
| ExportEntireStateRPC                     | 1 |
| ExportHostProfileRPC                     | 1 |
| ExportRPC                                | 1 |
| ExpressionList                           | 1 |
| ExternalSurfaceAttributes                | 1 |
| ExtrudeAttributes                        | 1 |
| FFTAttributes                            | 1 |
| FileOpenOptions                          | 1 |
| FilledBoundaryAttributes                 | 1 |
| FluxAttributes                           | 1 |
| GetProcInfoRPC                           | 3 |
| GetQueryParametersRPC                    | 1 |
| GlobalAttributes                         | 1 |
| GlobalLineoutAttributes                  | 1 |
| HideActiveAnnotationObjectsRPC           | 0 |
| HideActivePlotsRPC                       | 0 |
| HideAllWindowsRPC                        | 0 |
| HideToolbarsForAllWindowsRPC             | 0 |
| HideToolbarsRPC                          | 0 |

Continued on next page
<table>
<thead>
<tr>
<th>Table 2.14 – continued from previous page</th>
</tr>
</thead>
<tbody>
<tr>
<td>HistogramAttributes</td>
</tr>
<tr>
<td>IconifyAllWindowsRPC</td>
</tr>
<tr>
<td>ImportEntireStateRPC</td>
</tr>
<tr>
<td>ImportEntireStateWithDifferentSourcesRPC</td>
</tr>
<tr>
<td>IndexSelectAttributes</td>
</tr>
<tr>
<td>InitializeNamedSelectionVariablesRPC</td>
</tr>
<tr>
<td>IntegralCurveAttributes</td>
</tr>
<tr>
<td>InteractorAttributes</td>
</tr>
<tr>
<td>InverseGhostZoneAttributes</td>
</tr>
<tr>
<td>InvertBackgroundRPC</td>
</tr>
<tr>
<td>IsosurfaceAttributes</td>
</tr>
<tr>
<td>IsovolumeAttributes</td>
</tr>
<tr>
<td>KeyframeAttributes</td>
</tr>
<tr>
<td>LCSAttributes</td>
</tr>
<tr>
<td>LabelAttributes</td>
</tr>
<tr>
<td>LagrangianAttributes</td>
</tr>
<tr>
<td>LimitCycleAttributes</td>
</tr>
<tr>
<td>LineoutAttributes</td>
</tr>
<tr>
<td>LoadNamedSelectionRPC</td>
</tr>
<tr>
<td>LowerActiveAnnotationObjectsRPC</td>
</tr>
<tr>
<td>MaterialAttributes</td>
</tr>
<tr>
<td>MenuQuitRPC</td>
</tr>
<tr>
<td>MeshAttributes</td>
</tr>
<tr>
<td>MeshManagementAttributes</td>
</tr>
<tr>
<td>ModelFitAtts</td>
</tr>
<tr>
<td>MoleculeAttributes</td>
</tr>
<tr>
<td>MoveAndResizeWindowRPC</td>
</tr>
<tr>
<td>MovePlotDatabaseKeyframeRPC</td>
</tr>
<tr>
<td>MovePlotKeyframeRPC</td>
</tr>
<tr>
<td>MovePlotOrderTowardFirstRPC</td>
</tr>
<tr>
<td>MovePlotOrderTowardLastRPC</td>
</tr>
<tr>
<td>MoveViewKeyframeRPC</td>
</tr>
<tr>
<td>MoveWindowRPC</td>
</tr>
<tr>
<td>MultiCurveAttributes</td>
</tr>
<tr>
<td>MultiresControlAttributes</td>
</tr>
<tr>
<td>OnionPeelAttributes</td>
</tr>
<tr>
<td>OpenCLIClientRPC</td>
</tr>
<tr>
<td>OpenClientRPC</td>
</tr>
<tr>
<td>OpenComputeEngineRPC</td>
</tr>
<tr>
<td>OpenDatabaseRPC</td>
</tr>
<tr>
<td>OpenGUIClientRPC</td>
</tr>
<tr>
<td>OpenMDServerRPC</td>
</tr>
<tr>
<td>OverlayDatabaseRPC</td>
</tr>
<tr>
<td>ParallelCoordinatesAttributes</td>
</tr>
<tr>
<td>PersistentParticlesAttributes</td>
</tr>
<tr>
<td>PickAttributes</td>
</tr>
<tr>
<td>PlotDDTVispointVariablesRPC</td>
</tr>
<tr>
<td>PlotList</td>
</tr>
<tr>
<td>PoincareAttributes</td>
</tr>
<tr>
<td>PrintWindowRPC</td>
</tr>
</tbody>
</table>

Continued on next page
### Table 2.14 – continued from previous page

<table>
<thead>
<tr>
<th>Category</th>
<th>Count</th>
</tr>
</thead>
<tbody>
<tr>
<td>PrinterAttributes</td>
<td>1</td>
</tr>
<tr>
<td>ProcessAttributes</td>
<td>1</td>
</tr>
<tr>
<td>ProcessExpressionsRPC</td>
<td>0</td>
</tr>
<tr>
<td>ProjectAttributes</td>
<td>1</td>
</tr>
<tr>
<td>PromoteOperatorRPC</td>
<td>1</td>
</tr>
<tr>
<td>PseudocolorAttributes</td>
<td>1</td>
</tr>
<tr>
<td>QueryAttributes</td>
<td>1</td>
</tr>
<tr>
<td>QueryOverTimeAttributes</td>
<td>1</td>
</tr>
<tr>
<td>QueryRPC</td>
<td>1</td>
</tr>
<tr>
<td>RadialResampleAttributes</td>
<td>1</td>
</tr>
<tr>
<td>RaiseActiveAnnotationObjectsRPC</td>
<td>0</td>
</tr>
<tr>
<td>ReOpenDatabaseRPC</td>
<td>2</td>
</tr>
<tr>
<td>ReadHostProfilesFromDirectoryRPC</td>
<td>1</td>
</tr>
<tr>
<td>RecenterViewRPC</td>
<td>0</td>
</tr>
<tr>
<td>RedoViewRPC</td>
<td>0</td>
</tr>
<tr>
<td>RedrawRPC</td>
<td>0</td>
</tr>
<tr>
<td>ReflectAttributes</td>
<td>1</td>
</tr>
<tr>
<td>ReleaseToDDTRPC</td>
<td>1</td>
</tr>
<tr>
<td>RemapAttributes</td>
<td>1</td>
</tr>
<tr>
<td>RemoveAllOperatorsRPC</td>
<td>0</td>
</tr>
<tr>
<td>RemoveLastOperatorRPC</td>
<td>0</td>
</tr>
<tr>
<td>RemoveOperatorRPC</td>
<td>1</td>
</tr>
<tr>
<td>RemovePicksRPC</td>
<td>1</td>
</tr>
<tr>
<td>RenamePickLabelRPC</td>
<td>1</td>
</tr>
<tr>
<td>RenderingAttributes</td>
<td>1</td>
</tr>
<tr>
<td>ReplaceDatabaseRPC</td>
<td>2</td>
</tr>
<tr>
<td>ReplicateAttributes</td>
<td>1</td>
</tr>
<tr>
<td>RequestMetaDataRPC</td>
<td>2</td>
</tr>
<tr>
<td>ResampleAttributes</td>
<td>1</td>
</tr>
<tr>
<td>ResetAnnotationAttributesRPC</td>
<td>0</td>
</tr>
<tr>
<td>ResetAnnotationObjectListRPC</td>
<td>0</td>
</tr>
<tr>
<td>ResetInteractorAttributesRPC</td>
<td>0</td>
</tr>
<tr>
<td>ResetLightListRPC</td>
<td>0</td>
</tr>
<tr>
<td>ResetLineoutColorRPC</td>
<td>0</td>
</tr>
<tr>
<td>ResetMaterialAttributesRPC</td>
<td>0</td>
</tr>
<tr>
<td>ResetMeshManagementAttributesRPC</td>
<td>0</td>
</tr>
<tr>
<td>ResetOperatorOptionsRPC</td>
<td>1</td>
</tr>
<tr>
<td>ResetPickAttributesRPC</td>
<td>0</td>
</tr>
<tr>
<td>ResetPickLetterRPC</td>
<td>0</td>
</tr>
<tr>
<td>ResetPlotOptionsRPC</td>
<td>1</td>
</tr>
<tr>
<td>ResetQueryOverTimeAttributesRPC</td>
<td>0</td>
</tr>
<tr>
<td>ResetViewRPC</td>
<td>0</td>
</tr>
<tr>
<td>ResizeWindowRPC</td>
<td>3</td>
</tr>
<tr>
<td>RevolveAttributes</td>
<td>1</td>
</tr>
<tr>
<td>SPHResampleAttributes</td>
<td>1</td>
</tr>
<tr>
<td>SaveNamedSelectionRPC</td>
<td>1</td>
</tr>
<tr>
<td>SaveViewRPC</td>
<td>0</td>
</tr>
<tr>
<td>SaveWindowAttributes</td>
<td>1</td>
</tr>
<tr>
<td>SaveWindowRPC</td>
<td>0</td>
</tr>
<tr>
<td>ScatterAttributes</td>
<td>1</td>
</tr>
</tbody>
</table>

Continued on next page
<table>
<thead>
<tr>
<th>Method</th>
<th>Count</th>
</tr>
</thead>
<tbody>
<tr>
<td>SendSimulationCommandRPC</td>
<td>4</td>
</tr>
<tr>
<td>SetActivePlotsRPC</td>
<td>2</td>
</tr>
<tr>
<td>SetActiveTimeSliderRPC</td>
<td>1</td>
</tr>
<tr>
<td>SetActiveWindowRPC</td>
<td>1</td>
</tr>
<tr>
<td>SetAnimationAttributesRPC</td>
<td>0</td>
</tr>
<tr>
<td>SetAnnotationAttributesRPC</td>
<td>0</td>
</tr>
<tr>
<td>SetAnnotationObjectOptionsRPC</td>
<td>0</td>
</tr>
<tr>
<td>SetAppearanceRPC</td>
<td>0</td>
</tr>
<tr>
<td>SetBackendTypeRPC</td>
<td>1</td>
</tr>
<tr>
<td>SetCenterOfRotationRPC</td>
<td>1</td>
</tr>
<tr>
<td>SetCreateMeshQualityExpressionsRPC</td>
<td>1</td>
</tr>
<tr>
<td>SetCreateTimeDerivativeExpressionsRPC</td>
<td>1</td>
</tr>
<tr>
<td>SetCreateVectorMagnitudeExpressionsRPC</td>
<td>1</td>
</tr>
<tr>
<td>SetDefaultAnnotationAttributesRPC</td>
<td>0</td>
</tr>
<tr>
<td>SetDefaultAnnotationObjectListRPC</td>
<td>0</td>
</tr>
<tr>
<td>SetDefaultFileOpenOptionsRPC</td>
<td>0</td>
</tr>
<tr>
<td>SetDefaultInteractorAttributesRPC</td>
<td>0</td>
</tr>
<tr>
<td>SetDefaultLightListRPC</td>
<td>0</td>
</tr>
<tr>
<td>SetDefaultMaterialAttributesRPC</td>
<td>0</td>
</tr>
<tr>
<td>SetDefaultMeshManagementAttributesRPC</td>
<td>0</td>
</tr>
<tr>
<td>SetDefaultOperatorOptionsRPC</td>
<td>1</td>
</tr>
<tr>
<td>SetDefaultPickAttributesRPC</td>
<td>0</td>
</tr>
<tr>
<td>SetDefaultPlotOptionsRPC</td>
<td>1</td>
</tr>
<tr>
<td>SetDefaultQueryOverTimeAttributesRPC</td>
<td>0</td>
</tr>
<tr>
<td>SetGlobalLineoutAttributesRPC</td>
<td>0</td>
</tr>
<tr>
<td>SetInteractorAttributesRPC</td>
<td>0</td>
</tr>
<tr>
<td>SetKeyframeAttributesRPC</td>
<td>0</td>
</tr>
<tr>
<td>SetLightListRPC</td>
<td>0</td>
</tr>
<tr>
<td>SetMaterialAttributesRPC</td>
<td>0</td>
</tr>
<tr>
<td>SetMeshManagementAttributesRPC</td>
<td>0</td>
</tr>
<tr>
<td>SetNamedSelectionAutoApplyRPC</td>
<td>1</td>
</tr>
<tr>
<td>SetOperatorOptionsRPC</td>
<td>1</td>
</tr>
<tr>
<td>SetPickAttributesRPC</td>
<td>0</td>
</tr>
<tr>
<td>SetPlotDatabaseStateRPC</td>
<td>3</td>
</tr>
<tr>
<td>SetPlotDescriptionRPC</td>
<td>1</td>
</tr>
<tr>
<td>SetPlotFollowsTimeRPC</td>
<td>0</td>
</tr>
<tr>
<td>SetPlotFrameRangeRPC</td>
<td>3</td>
</tr>
<tr>
<td>SetPlotOptionsRPC</td>
<td>1</td>
</tr>
<tr>
<td>SetPlotOrderToFirstRPC</td>
<td>1</td>
</tr>
<tr>
<td>SetPlotOrderToLastRPC</td>
<td>1</td>
</tr>
<tr>
<td>SetPlotSILRestrictionRPC</td>
<td>0</td>
</tr>
<tr>
<td>SetPrecisionTypeRPC</td>
<td>1</td>
</tr>
<tr>
<td>SetQueryFloatFormatRPC</td>
<td>1</td>
</tr>
<tr>
<td>SetQueryOverTimeAttributesRPC</td>
<td>0</td>
</tr>
<tr>
<td>SetRemoveDuplicateNodesRPC</td>
<td>1</td>
</tr>
<tr>
<td>SetRenderingAttributesRPC</td>
<td>0</td>
</tr>
<tr>
<td>SetStateLoggingRPC</td>
<td>0</td>
</tr>
<tr>
<td>SetSuppressMessagesRPC</td>
<td>1</td>
</tr>
<tr>
<td>SetTimeSliderStateRPC</td>
<td>1</td>
</tr>
<tr>
<td>SetToolUpdateModeRPC</td>
<td>1</td>
</tr>
</tbody>
</table>

Continued on next page
<table>
<thead>
<tr>
<th>Event Description</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>SetToolBarIconSizeRPC</td>
<td>0</td>
</tr>
<tr>
<td>SetTreatAllDBsAsTimeVaryingRPC</td>
<td>1</td>
</tr>
<tr>
<td>SetTryHarderCyclesTimesRPC</td>
<td>1</td>
</tr>
<tr>
<td>SetView2DRPC</td>
<td>0</td>
</tr>
<tr>
<td>SetView3DRPC</td>
<td>0</td>
</tr>
<tr>
<td>SetViewAxisArrayRPC</td>
<td>1</td>
</tr>
<tr>
<td>SetViewCurveRPC</td>
<td>0</td>
</tr>
<tr>
<td>SetViewExtentsTypeRPC</td>
<td>1</td>
</tr>
<tr>
<td>SetViewKeyframeRPC</td>
<td>0</td>
</tr>
<tr>
<td>SetWindowAreaRPC</td>
<td>1</td>
</tr>
<tr>
<td>SetWindowLayoutRPC</td>
<td>1</td>
</tr>
<tr>
<td>SetWindowModeRPC</td>
<td>1</td>
</tr>
<tr>
<td>ShowAllWindowsRPC</td>
<td>0</td>
</tr>
<tr>
<td>ShowToolbarsForAllWindowsRPC</td>
<td>0</td>
</tr>
<tr>
<td>ShowToolbarsRPC</td>
<td>0</td>
</tr>
<tr>
<td>SliceAttributes</td>
<td>1</td>
</tr>
<tr>
<td>SmoothOperatorAttributes</td>
<td>1</td>
</tr>
<tr>
<td>SphereSliceAttributes</td>
<td>1</td>
</tr>
<tr>
<td>SpreadsheetAttributes</td>
<td>1</td>
</tr>
<tr>
<td>StaggerAttributes</td>
<td>1</td>
</tr>
<tr>
<td>StartPlotAnimationRPC</td>
<td>1</td>
</tr>
<tr>
<td>StatisticalTrendsAttributes</td>
<td>1</td>
</tr>
<tr>
<td>StopPlotAnimationRPC</td>
<td>1</td>
</tr>
<tr>
<td>SubdivideQuadsAttributes</td>
<td>1</td>
</tr>
<tr>
<td>SubsetAttributes</td>
<td>1</td>
</tr>
<tr>
<td>SuppressQueryOutputRPC</td>
<td>1</td>
</tr>
<tr>
<td>SurfaceNormalAttributes</td>
<td>1</td>
</tr>
<tr>
<td>TensorAttributes</td>
<td>1</td>
</tr>
<tr>
<td>ThreeSliceAttributes</td>
<td>1</td>
</tr>
<tr>
<td>ThresholdAttributes</td>
<td>1</td>
</tr>
<tr>
<td>TimeSliderNextStateRPC</td>
<td>0</td>
</tr>
<tr>
<td>TimeSliderPreviousStateRPC</td>
<td>0</td>
</tr>
<tr>
<td>ToggleAllowPopupRPC</td>
<td>1</td>
</tr>
<tr>
<td>ToggleBoundingBoxModeRPC</td>
<td>0</td>
</tr>
<tr>
<td>ToggleCameraViewModeRPC</td>
<td>0</td>
</tr>
<tr>
<td>ToggleFullFrameRPC</td>
<td>0</td>
</tr>
<tr>
<td>ToggleLockTimeRPC</td>
<td>0</td>
</tr>
<tr>
<td>ToggleLockToolsRPC</td>
<td>0</td>
</tr>
<tr>
<td>ToggleLockViewModeRPC</td>
<td>0</td>
</tr>
<tr>
<td>ToggleMaintainViewModeRPC</td>
<td>0</td>
</tr>
<tr>
<td>TogglePerspectiveViewRPC</td>
<td>0</td>
</tr>
<tr>
<td>ToggleSpinModeRPC</td>
<td>0</td>
</tr>
<tr>
<td>TransformAttributes</td>
<td>1</td>
</tr>
<tr>
<td>TriangulateRegularPointsAttributes</td>
<td>1</td>
</tr>
<tr>
<td>TruecolorAttributes</td>
<td>1</td>
</tr>
<tr>
<td>TubeAttributes</td>
<td>1</td>
</tr>
<tr>
<td>TurnOffAllLocksRPC</td>
<td>0</td>
</tr>
<tr>
<td>UndoViewRPC</td>
<td>0</td>
</tr>
<tr>
<td>UpdateColorTableRPC</td>
<td>1</td>
</tr>
<tr>
<td>UpdateDBPluginInfoRPC</td>
<td>1</td>
</tr>
</tbody>
</table>

Continued on next page
2.7 Contributing To VisIt CLI Documentation

Note: We are still refining this procedure!

The Python doc strings for most functions in VisIt’s cli are generated from the examples embedded in the cli_manual/functions.rst file. This allows us to have a single source for both our CLI sphinx docs and the doc strings embedded in VisIt’s compiled Python module. The functions_to_method_doc.py helper script generates MethodDoc.C from the examples embedded in the rst source.

The Python doc strings for Attribute objects and Events are extracted from the CLI for use in the CLI sphinx docs. The sphinx_cli_extractor.py runs VisIt to generate cli_manual/attributes.rst and cli_manual/events.rst

2.7.1 Steps to update the CLI Manual

1. Modify cli_manual/functions.rst
2. Run functions_to_plain.py to generate PY_RST_FUNCTIONS_TO_PYTHON.py
3. Run 2to3 -p PY_RST_FUNCTIONS_TO_PYTHON.py to check for Python syntax errors and Python 3 compatibly
4. Run functions_to_method_doc.py to regenerate MethodDoc.C
5. Build and run the VisIt cli and assure yourself help(<your-new-func-doc>) produces the desired output
6. Run the sphinx_cli_extractor.py tool producing new attributes.rst and events.rst files. To do so, you may need to use a combination of the PATH and PYTHONPATH environment variables to tell the sphinx_cli_extractor.py script where to find the VisIt module, visit in VisIt’s site-packages and where to find the Python installation that that module is expecting to run with. In addition, you may need to use the PYTHONHOME environment variable to tell VisIt’s visit module where to find standard Python libraries. For example, to use an installed version of VisIt on my OSX machine, the command would look like...

```bash
env PATH=~/Applications/VisIt.app/Contents/Resources/2.13.3/darwin-x86_64/bin:/Applications/VisIt.app/Contents/Resources/bin:$PATH
PYTHONHOME=~/Applications/VisIt.app//Contents/Resources/2.13.3/darwin-x86_64/lib/python
```

(continues on next page)
7. Assuming you successfully ran the above command, producing new `attributes.rst` and `events.rst` files, then do a local build of the documentation here and confirm there are no errors in the build

```
sphinx-build -b html . _build -a
```

8. Then open the file, `_build/index.html`, in your favorite browser to view.

9. Add all the changed files to a commit and push to GitHub

10. The GitHub integration with ReadTheDocs should result in your documentation updates going live a short while (<15 mins) after it has been merged to develop.

# 2.8 Acknowledgments

This document is primarily based on the excellent manual put together by Brad Whitlock of Lawrence Livermore in 2005. Several years afterwards, the content from that manual was converted to serve as online help for the command line interpreter itself. As new routines were added, this online help was updated. In 2010, Jakob van Bethlehem of the University of Groningen wrote a wonderful script to convert the online help to manual form. In 2011, Hank Childs of Lawrence Berkeley merged the descriptions from Brad Whitlock’s original manual with the function definitions produced by Jakob’s conversion of the online help. In 2018, Alister Maguire of Lawrence Livermore wrote a script for converting this manual to restructuredText format to be used with Sphinx. The result is this manual.
This manual contains a series of hands on tutorials that expose the user to the features in VisIt. The first three tutorials form a good basis for using VisIt, including the basics of using the Graphical User Interface (GUI), performing data analysis and using Python to script and automate tasks in VisIt. After that are a series of tutorials that cover advanced topics in detail.

Here are links to the datasets used in the VisIt Basics, Data Analysis and Scripting tutorials.

- Tarfile: visit_tutorial_data.tar.gz (59.8 MB)
- Zip: visit_tutorial_data.zip (59.8 MB)

Here are links to the data used for Aneurysm (Blood Flow) simulation tutorial, which is also used in one of the examples in the Scripting tutorial.

- Tarfile: aneurysm_tutorial_data.tar.gz (361 MB)
- Zip: aneurysm_tutorial_data.zip (361 MB)

Here are links to data used for the Potential Flow tutorial.

- Tarfile: potential_flow_tutorial_data.tar.gz (161 MB)
- Zip: potential_flow_tutorial_data.zip (160 MB)

Contents:

### 3.1 VisIt Basics

#### 3.1.1 Starting

The way you start VisIt depends on the platform you are on:

- On Windows, double click on the VisIt desktop icon
- On Mac, double click on the VisIt icon where you installed it (generally in the /Applications folder).
- On Unix, invoke: /path/to/visit/bin/visit
Most people ultimately put /path/to/visit/bin in their $PATH and then just say visit.

### 3.1.2 What you see

![The VisIt graphical user interface and visualization window](image)

- The tall grey window on the left is called the **Graphical User Interface**, which will be referred to from here on as the **GUI**. It is the primary mechanism for driving VisIt.
- The window on the right is called the **visualization window**. It displays results.

### 3.1.3 Opening files

The first thing to do is to open files.

1. Go to the **GUI** and click on the **Open** icon.
2. This brings up the File open window.
3. Change the **Path** field to the “tutorial_data” folder.
4. Highlight the file “example.silo” and then click **OK**.

You’ve opened a file!

#### Advanced file opening features

1. In the File open window:
   - There is a field for **Host**. That is how you open a file on another system and run in client/server mode.
   - There is a **Filter**. That is provided to subset the file list to only the files VisIt may want.
Fig. 3.2: The File open window
Example filter: “*.silo *.vtk”

2. VisIt uses heuristics to determine the file type.
   - You can explicitly set the file type by setting the *Open file as type:* to the appropriate type.

3. You can also open files on the command line. For example, *visit -o file.ext* opens the file “file.ext”.

### 3.1.4 Making a plot

1. Click on the *Add* icon to access various plots. This is located about half way down the Main window.
2. Select *Pseudocolor-*temp* to add a Pseudocolor plot.
3. After adding a plot, you will see a green entry added to the “Plot list”, which is located half way down the *GUI*.
   - This means VisIt will draw this plot after you click *Draw*.

![Plot list with a Pseudocolor plot](image)

**Fig. 3.3:** The plot list with a Pseudocolor plot in it

4. Click *Draw*.
5. You should see a plot appear in the visualization window.
6. Go to *Add-*Mesh-*Mesh*.
7. Click *Draw*.
8. You should now see both a Pseudocolor and Mesh plot.
9. Highlight the Pseudocolor plot in the Plot list.
10. Click the *Hide/Show* button.
    - This will hide the Pseudocolor plot. You should now see only the Mesh plot.
11. Highlight the Mesh plot and click *Delete*.
    - You should now have an empty visualization window.
Fig. 3.4: A Pseudocolor and mesh plot displayed in a visualization window
3.1.5 Modifying the plot attributes

1. Go to `PlotAtts->Pseudocolor`. This is located in the menu bar at the top of the Main menu.
2. This brings up the Pseudocolor plot attributes window.

![Pseudocolor plot attributes window](image)

3. Change the `Scale` from `Linear` to `Log`.
4. Click `Apply`.
   - The colors changed.
5. Click `Minimum` on and change the value to “3”.
6. Click `Maximum` on and change the value to “4”.

- The Pseudocolor plot should now be selected.

12. Click `Hide/Show`.
   - The Pseudocolor plot should reappear.
7. Click Apply.
   • The colors change again.

8. Change the **Opacity** mode to **Constant**.
   • Change the opacity slider to 50%.

9. Click **Apply**.
   • You can now see through the plot. Note that you only see the external faces. If you want to see the data from
     the whole volume, that will be with the volume plot.

10. Change back the **Scale**, **Limits**, and **Opacity** back to their original settings and click **Apply**.

11. Dismiss the Pseudocolor plot attributes window.

### 3.1.6 Applying an operator

1. Click on the **Operators** button to access various operators. This is located next to the *Add* button.

2. Select **Slicing->Slice** to add a Slice operator
   • The visualization window will go blank and the Pseudocolor entry in the Plot list will turn green.
   • This allows you to change the slice attributes before applying the Slice operator.
   • We will apply the operator with the default attributes.

3. Click **Draw**.
   • You are now looking at a 2D slice.

4. Go to **OpAtts->Slicing->Slice**.

5. This brings up the Slice operator attributes window.

6. There are many controls for setting the slice plane … play with them.

7. Operators can be removed by clicking on an expansion arrow in the Plot list, then clicking on the red X icon
   next to an operator.

### 3.1.7 VisIt interaction modes

There are six basic interaction modes:

1. Navigate
2. Zoom
3. Zone pick
4. Node pick
5. Spreadsheet pick
6. Lineout

The interaction mode is controlled by the toolbar, which is located at the top of the visualization window. The six
interaction modes are all located together on the toolbar, towards the bottom.
Fig. 3.6: The Slice operator attributes window

Fig. 3.7: The visualization tool bar with the icons for setting the interaction mode
Using **navigate** mode

You always start in Navigate mode. Navigate mode is indicated by the Navigate icon, represented by a compass, being indented. It allows you to pan and rotate the data set.

1. Put the cursor in the visualization window.
2. Left click (or single click if you do not have a 3 button mouse) and move the mouse.
3. The data set will pan with the mouse.
   - In 3D, the data set rotates.

Using **zoom** mode

Zoom mode is indicated by the Zoom icon, represented by a magnifying glass, being indented. It allows you to zoom the image by selecting a rectangular region.

1. Click on the Zoom icon.
2. Go to the visualization window and left click (single click) and HOLD IT DOWN.
3. Move the mouse a bit.
   - You should see a rubber band.
4. Lift up the mouse button.
   - You should now be zoomed in so that the viewport matches what was previously inside the rubber band.

Using **lineout** mode

Lineout mode is indicated by the Lineout icon, represented by a curve plot of red and blue curves, being indented. It allows the user to create a plot of a scalar variable as a function of distance along a line.

1. First we will reset the view. Click on the Reset view icon, represented by a camera that has a green “X” around it (The camera is mostly obscured by the X).
   - This will reset your view.
2. Click on the Lineout icon.
3. Put the cursor over the data and left click (single click) and HOLD IT DOWN.
4. Move the mouse a bit.
   - You should see a single line moving around.
5. Lift up the mouse button.
6. The window layout changes. You now have two windows. The first window is the same, but the second now contains a “Lineout”, which has temp as a function of distance over the line.
7. On the window that has the curve, find the Delete window icon, represented by a window with a red circle with a line through it.
8. Click this button.
   - The new window will disappear and you should now have only one window.
Using pick mode

Pick mode is indicated by the Zone pick or Node pick icon, represented by a “+” with a small Z or a “+” with a small N, being indented. It allows the user to query a variable associated with a zone or node.

1. Click on the Zone pick icon.
2. Put the cursor over the data set and left click (single click).
3. This brings up the Pick window.
   • The Pick window contains information about the zone (i.e. cell or element) that you just picked in.
   
   Pick can return a lot more information than what it just did if you use the Pick window.

4. Go to the Variables drop down menu and select Scalars/pressure.
5. Turn on Physical Coords under For Nodes.
6. Turn on Domain-Logical Coords under For Zones.
7. Click Apply.
8. Make another pick.
   • You get information about pressure, the coordinates of each node, and the logical coordinates for the zone.

3.1.8 Other plots

1. We will experiment with the Contour, Filled Boundary, Label, Vector and Volume plots.

3.1.9 Other operators

1. We will experiment with the Clip and Threshold operators.

3.1.10 Saving an image

1. With a current plot, go to File->Save window.
   • This saves an image to the filesystem.

   On Windows, the default location for saved images is in Documents/VisIt/My images.

3.1.11 Saving a database

VisIt can be part of a larger tool chain.

1. If you do not already have one, make a Pseudocolor plot of temp from the “example.silo” database.
2. Apply the Threshold operator and change the range to be 3->max.
3. Click Draw.
4. Go to File->Export database.
5. This brings up the Export Database window.
6. Change Export to VTK.
Fig. 3.8: The Pick output window
Fig. 3.9: The Export Database window
7. Be sure to set the output directory or the exported file will be written to the working directory (on Windows that would be the directory where VisIt is installed).

8. Click Export.
   - The Export options for VTK writer window will pop up at this point. It allows you to specify the options for the VTK writer. We will use the default options.

9. Click Ok.
   - A file named “visit_ex_db.vtk” has been saved to the file system.

### 3.1.12 Subsetting

1. Delete any plots in your visualization window.
2. Open the file “multi_ucd3d.silo”.
3. Make a Subset plot of “domains(mesh1)”.
   - The plot is colored by “domains”, which normally correspond to a simulation’s processors.
4. In the Plot list, find the overlapping transparent black and white ovals (like a Venn diagram) and click on it.
5. This brings up the Subset window.

![Subset window](image.png)

Fig. 3.10: The Subset window

6. Click on domains in the left most panel.
   - This will expand the list of domains in the center panel.
7. Turn off some domains and click Apply.
   - You will see some of the domains disappear.
   - Subsetting works with any plot type.
8. Turn all the domains back on.
9. Click on mat1 in the left most panel.
   • This will expand the list of materials in the center panel.
10. Turn off materials 1 and 3.
    • You will see material 2 only, colored by domain.

This mechanism is used to expose subsetting for materials, domains, AMR levels, and other custom subsettable parts.

### 3.2 Data Analysis

This section describes two important abstractions in VisIt: Queries and Expressions.

#### 3.2.1 Queries

**What are queries**

Queries are the mechanism to do data analysis, to pull out a number or curve that describes the data set.

**Experiment with queries**

1. Go to Controls->Query.
2. This brings up the Query window.

**Variable-related**

Variable related queries provide information about variables.

1. Change the Display in the Query window to be Variable-related.
2. Go back to the GUI, delete any plots, open up “example.silo”, create a Pseudocolor plot of temp and click Draw.
3. Highlight MinMax and click Query.
   • The result will be displayed in the Query results. It will tell you the minimum, maximum and their locations.
4. Apply the Slice operator to your plot.
5. Do another MinMax query.
   • It gives you the different results. This is because the Query parameter Actual Data is selected. This means the answer will be the minimum and maximum constrained to the slice.
6. Change the Query parameter to be Original Data.
7. Do another MinMax query.
   • This time the answer will match the result of the first query. It will display the minimum and maximum for what is in the file, not what is on the screen.
8. Now highlight Variable Sum and click Query.
   • This will sum up all of the values in the data set.
9. Now highlight Weighted Variable Sum and click Query.
Fig. 3.11: The MinMax query
• This will sum up all of the values, but it will weight by area (since you have a slice).
• For 3D, it will weight by volume.
• For axi-symmetric 2D calculations, it will weight by revolved volume.

3. Note that both queries have options for doing queries over time (grayed out because we don’t have a time varying data set).
• This is for time varying data and will produce a curve in a separate window.
1. Now highlight Lineout.
• Note that you must have left Project to 2D enabled in the Slice operator for this next one to work correctly.
2. Change the start point to “-5 -5 0” and the end point to “5 5 0”.
3. Click Query.
4. This is a way to get exact lineouts.
5. You can also take 3D lineouts this way.
1. Now highlight Pick.
2. Click Query.
• This will provide information about the zone containing the coordinate “0 0 0”.
3. Change the mode to Pick using coordinate to determine node.
4. Click Query.
• This will provide information about the node nearest the coordinate “0 0 0”.
5. Change the mode to Pick using domain and element Id.
6. Click Query.
• This will provide information about the node or zone in the specified domain.

You can also perform a query using the global element id by selecting Pick using global element Id. This only works if the file contains global element id information, which this file does not.

Mesh-related

1. Change the Display in the Query window to be Mesh-related.
2. Experiment with the 2D area, SpatialExtents, NumZones, and Zone Center queries.
• For the Zone Center query, you will set the Domain to “0”.
• The domain is used for when you have a parallel file, where the data has been “domain decomposed” for parallel processing.

ConnectedComponents-related

1. If you haven’t already removed the slice operator, do that now, so you have just a Pseudocolor plot of temp.
2. Apply the Isovolume operator. Change the Lower bound of the Isovolume operator attributes to be “4”.
3. You will now see a bunch of blobs in space.
4. Change the Display in the Query window to be ConnectedComponents-related.
5. Perform the *Number of Connected Components* query.
   • It should tell you that there are 15 components.
6. Apply the Clip operator with the default settings.
7. Perform the *Number of Connected Components* query again.
   • It should now say there are 14 components.
   • Operators affect queries.

### 3.2.2 Queries over Time

Queries over time perform analysis through time and generate a time-curve.

<table>
<thead>
<tr>
<th>What are queries over time perform analysis through time and generate a time-curve.</th>
</tr>
</thead>
</table>

<table>
<thead>
<tr>
<th>Experiment with queries over time</th>
</tr>
</thead>
<tbody>
<tr>
<td>Weighted Variable Sum</td>
</tr>
</tbody>
</table>

1. Go to *Controls->Query*.
2. This brings up the Query window.
3. Go back to the *GUI*, delete any existing plots, open up “wave.visit”, and make a Pseudocolor plot of *pressure*.
4. Find and Highlight *Weighted Variable Sum* and click *Do Time Query*.
5. Options for changing the *Starting timestep*, *Ending timestep* and *Stride* will be available.
   • Note that these are 0-origin timestate indices and not cycles or times.
6. Click *Query*.
   • The result will be displayed in a new Window. By default the x-axis will be cycle and the y-axis will be the weighted summation of the *pressure*.

### Pick

1. Pick can do multiple-variable time curves.
2. Make *Window 2* active, delete the plot, and make *Window 1* active again.
3. Find and Highlight *Pick* in the Query window and click *Do Time Query* to enable time-curve options.
4. Change the *Variables* option to add v using the *Variables->Scalars* dropdown menu.
5. Select *Pick using domain and element Id*. Leave the defaults for *Node Id* and *Domain Id* as “0”.
6. Select *Preserve Picked Element Id*.
7. Click *Query*.
   • The result will be two curves in a single xy plot.
8. Make *Window 2* active, delete the plot, and make *Window 1* active again.
9. Change the *Multiple-variable Time Curve options* to *Create Multiple Y-Axes plot*.
10. Click *Query*.  

### 3.2. Data Analysis
Fig. 3.12: The Weighted Variable Sum query
3.2. Data Analysis

Fig. 3.13: The output of the Weighted Variable Sum query over time
Fig. 3.14: The Pick query
• The result will be a Multi-curve plot (multiple axes) in Window 2.

Fig. 3.15: The Pick query output

11. **NOTE:** Time Pick can also be performed via the mouse by first setting things up on the *Time Pick* tab in the Pick window (*Controls-*->*Pick*).

**Changing global options**

1. Go to *Controls-*->*Query over time options*.
2. This brings up the QueryOverTime window.
3. Here you can change the values displayed in the x-axis for all subsequent queries over time.
4. You can also change the window used to display time-curves. By default, the first un-used window becomes the time-curve window, and all subsequent time-curves are generated in the same window.
3.2.3 Built-in queries

3.2.4 Expressions

Expressions in VisIt create new mesh variables from existing ones. These are also known as derived quantities. VisIt’s expression system supports only derived quantities that create a new mesh variable defined over the entire mesh. Given a mesh on which a variable named pressure is defined, an example of a very simple expression is “2*pressure”. On the other hand, suppose one wanted to sum (or integrate) “pressure” over the entire mesh (maybe the mesh represents some surface area over which a force calculation is desired). Such an operation is not an expression in VisIt because it does not result in a new variable defined over the entire mesh. In this example, summing pressure over the entire mesh results in a single, scalar, number, like “25.6”. Such an operation is supported instead by VisIt’s Variable Sum Query. This tends to be true in general; Expressions define whole mesh variables while Queries define single numerical values (there are, however, some Queries for which this is not strictly true).

A simple algebraic expression, “2*radial”

1. Open up “noise2d.silo”.
2. Create a Pseudocolor plot of the variable radial.
   • Take note of the legend range, “0...28.28”
3. Go to Controls->Expressions.
4. Click on New in the bottom left.
   • This will create an expression and give it a default name, “unnamed1”.
5. Rename this expression by typing “radial2” into the Name field.
   • Take note of the Type of the variable. By default, VisIt assumes the type of the new variable you are creating is a scalar mesh variable (e.g., a single numerical value for each node or zone/cell in the mesh). Here, we are
indeed creating a scalar variable and so there is no need to adjust the Type. However, in some of the examples that follow, we’ll be creating vector mesh variables and if we don’t specify the correct type, we’ll get an error message.

6. Place the cursor in the Definition pane of the Expressions dialog.
7. Type the number “2” followed by the C/C++ language symbol for multiplication, “*”.
8. Now, you can either type the name “radial” or you can go to the Insert Variable... pulldown menu and find and select the radial variable there (see picture at right).

9. Click Apply.
10. Now, go to the main VisIt GUI Panel to the Variables pulldown.
   - Note that radial2 now appears in the list of variables there.
11. Select radial2 from the pull down and click Draw.
   - Visually, the image will not look any different. But, if you take a close look at the legend you will see it is now showing “0...56.57”.

Visit supports several unary and binary algebraic expressions including +, -, /, \*, bitwise-^, bitwise-&; sqrt(), abs(), ciel(), floor(), ln(), log10(), exp() and more.
Accessing coordinates (of a mesh) in expressions

Here, we’ll use the category of Mesh expressions to access the coordinates of a mesh, again, working with “noise2d.silo”.

1. Go to Controls->Expressions.
2. Click the New button and name this expression “Coords”.
3. Set the Type to Vector mesh variable (because coordinates, at least in this 2D example, are a vector quantity).
4. Put the cursor in the Definition pane.
5. Go to Insert Function... and find the Mesh category of expressions and then, within it, find the coord function expression.
   • This should result in the insertion of “coord()” in the Definition pane and place the cursor between the two parenthesis characters.
   • Note that in almost all cases, the category of Mesh expressions expect one or more mesh variables as operands.
6. Now, go to Insert Variable... pull down and then to the Meshes submenu and select Mesh.
   • This should result in Mesh being inserted between the parentheses in the definition.
7. Click Apply.
8. Now, we’ll define two scalar expressions for the “X” and “Y” coordinates of the mesh. While still in the Expressions window,
   1. Click New.
   2. Name the new expression “X”.
      • Note that VisIt’s expression system is case sensitive so “x” and “X” can be different variable names.
   3. Leave the type as Scalar mesh variable

Fig. 3.18: Expression variable appears in the plot menus
4. Type into the definition pane, “Coords[0]”
   • This expression uses the array bracket dereference operator “[ ]” to specify a particular component of an array. In this case, the array being dereferenced is the vector variable defined by “Coords”.
   • Note that VisIt’s expression system always numbers its array indices starting from zero.
5. Click Apply.
6. Now, repeat these steps to define a “Y” expression for the “Y” coordinates as “Coords[1]”.

9. Finally, we’ll define the “distance” expression
   1. Click the New button.
   2. Give the new variable the name “Dist” (Type should be Scalar mesh variable).
   3. Type in the definition “sqrt(X*X+Y*Y)”.
   4. Click Apply.

   Now, we’ll use the new “Dist” variable we’ve just defined to display some data.
   1. Delete any existing plots from the plot list.
   2. Add a Pseudocolor plot of shepardglobal.
   3. Add an Isovolume operator.
   • Although this example is a 2D example and so volume doesn’t seem to apply, VisIt’s Isovolume operator performs the equivalent operation for 2D data.
4. Bring up the Isovolume operator attributes (either expand the plot by clicking on the triangle to the left of its name in the plot list and double clicking on the Isovolume operator there or go to the OpAtts menu and bring up Isovolume operator attributes that way).
5. Set the variable to Dist.
6. Set the Lower bound to “5” and the Upper bound to “7”.
7. Click Apply.
8. Click Draw.

   You should get the picture below. In this picture, we are displaying a Pseudocolor plot of shepardglobal, but Isovolumed by our Dist expression in the range “[5 . . 7]”.

   This example also demonstrates the use of an expression function, coord() to operate on a mesh and return its coordinates as a vector variable on the mesh.

   VisIt has a variety of expression functions that operate on a Mesh including area (for 2D meshes), volume (for 3D meshes, revolved_volume (for 2D cylindrically symmetric meshes), zonetype, and more. In addition, VisIt includes the entire suite of Mesh quality expressions from the Verdict Library.

Creating vector and tensor valued variables from scalars

If the database contains scalar variables representing the individual components of a vector or tensor, VisIt’s Expression system allows you to construct the associated vector (or tensor). You create vectors in VisIt’s Expression system using the curly bracket vector compose “[ {} ]” operator. For example, using “noise2d.silo” again as an example, suppose we want to compose a Vector valued expression that has “shepardglobal” and “hardyglobal” as components.

   Here are the steps.

1. Go to Controls->Expressions.
2. Click the New button and set Name to “randvec”.

3.2. Data Analysis 691
Fig. 3.19: Example of using the radial expression
3. Be sure to also set the Type to Vector mesh variable.

4. Place the cursor in Definition pane and type “{shepardglobal, hardyglobal}”.

5. Click Apply.

6. Go to Plots->Vector.

   • You should now see randvec appear there as a variable name to plot.

7. Add the Vector plot of randvec.

   In the example above, we used the vector compose operator, “{ }” to create a vector variable from multiple scalar variables. We can do the same to create a tensor variable. Recall from calculus that a rank 0 tensor is a scalar, a rank 1 tensor is a vector and a rank 2 tensor is a matrix. So, to create a tensor variable, we use multiple vector compose operators nesting within another vector compose operator. Here, solely for the purposes of illustration (e.g. this isn't a physically meaningful tensor) we’ll use the “X” and “Y” coordinate component scalars we defined earlier together with the shepardglobal and hardyglobal.

1. Go to Controls->Expressions.

2. Click New and set the Name to “tensor”.

3. Be sure to also set the Type to Tensor mesh variable.

4. Place the cursor in Definition pane and type “{ {she ShepardGlobal, hardyglobal}, {X,Y} }”.

   • Note the two levels of curly braces. The outer level is the whole rank 2 tensor matrix and the inner curly braces are each row of the matrix.

   • Note that you could also have defined the same tensor expression using two vector expressions like so, “{randvec, Coords}”.

5. Click Apply.

6. Add a Tensor plot of tensor variable.

Variable compatibility gotchas (tensor rank, centering, mesh)

VisIt will allow you to define expressions that it winds up determining to be invalid later when it attempts to execute those expressions. Some common issues are the mixing of incompatible mesh variables in the same expression without the necessary additional functions to make them compatible.

Tensor rank compatibility

For example, what happens if you mix scalar and vector mesh variables (e.g. variables of different Tensor rank) in the same expression? Again, using “noise2d.silo”.

1. Define the expression, “foo” as “grad+shepardglobal” with the Type Vector mesh variable.

   • Note that grad is a Vector mesh variable and shepardglobal is a Scalar mesh variable.

2. Now, attempt to do a Vector plot of foo. This works because VisIt will add the scalar to each component of the vector resulting a new vector mesh variable

3. But, suppose you instead defined foo to be of Type Scalar mesh variable.

   • VisIt will allow you to define this expression. But, when you go to plot it, the plot will fail.

   As an aside, as you go back and forth between the Expressions window creating and/or adjusting expression definitions, VisIt makes no attempt to keep track of all the changes you've made in expressions and automatically
Fig. 3.20: Example of using vector and tensor expressions
update plots as expressions change. You will have to manually clear or delete plots to force VisIt to re-draw plots in
which you’ve changed expressions.

In the above example, if on the other hand, you had set type of “foo” to Scalar Mesh Variable, then VisIt would have
failed to plot it because it is adding a scalar and a vector variable and the result of such an operation is always a
Vector mesh variable. If what you really intended was a scalar mesh variable, then use one of the expression
functions that converts a vector to a scalar (e.g. \texttt{magnitude()} function or array dereference operator \([])\) to convert the
Vector mesh variable in your expression to a scalar mesh variable. So, “\texttt{grad[i]}+\texttt{shephardglobal}” where “\(i\)” is “0” or
“1” would work to define a scalar mesh variable. Or, “\texttt{magnitude(\text{grad})+shephardglobal}” would also have worked.

Centering compatibility

In “noise2d.silo”, some variables are zone centered and some are node centered. What happens if you combine these
in an expression? VisIt will default to zone centering for the result. If this is not the desired result, use the
“\texttt{recenter()}” expression function, where appropriate, to adjust centering of some of the terms in your expression. For
example, again using “noise2d.silo”.

1. Define the Scalar mesh variable expression “\texttt{bar}” as “\texttt{shephardglobal+airVf}”.
   • For reference, in “noise2d.silo”, “\texttt{shephardglobal}” is node centered while “\texttt{airVf}” is zone centered.
2. Do a Pseudocolor plot of “\texttt{bar}”.
   • Note that “\texttt{bar}” displays as a zone centered quantity.
3. Now, go back to the expression and recenter “\texttt{airVf}” by adjusting the definition to “\texttt{shephardglobal+recenter(airVf)}”.
   • The \texttt{recenter()} expression function is a toggle in that it will take whatever the variable’s centering is and swap it
     (node->zone and zone->node).
   • The \texttt{recenter()} expression function also takes a second argument, a string of one of the values toggle, zonal, nodal
to force a particular behavior.
   • Note that when you click \texttt{Apply}, the current plot of “\texttt{bar}” does not change. You need to manually delete and
     re-create the plot (or clear and re-draw the plots).

   Finally, note that these two expressions...
   • “\texttt{shephardglobal+recenter(airVf)}”
   • “\texttt{recenter(shephardglobal+airVf)}”

both achieve a node-centered result. But, each expression is subtly (and numerically) different. The first recenter’s
“\texttt{airVf}” to the nodes and then performs the summation operator at each node. In the second, there is an implied
recentering of “\texttt{shephardglobal}” to the zones first. Then, the summation operator is applied at each zone center and
finally the results are recentered back to the nodes. In all likelihood this results in a numerically lower quality result.

The moral is that in a complex series of expressions be sure to take care where you want recentering to occur.

Mesh compatibility

In many cases, especially in Silo databases, all the available variables in a database are not always defined on the
same mesh. This can complicate matters involving expressions in variables from different meshes.

Just as in the previous two examples of incompatible variables where the solution was to apply some functions to
make the variables compatible, we have to do the same thing when variables from different meshes are combined in
an expression. The key expression functions which enable this are called \textit{Cross Mesh Field Evaluation} or \textit{CMFE}
functions. We will only briefly touch on these here. CMFEs will be discussed in much greater detail in a tutorial
devoted to that topic.
Again, using “noise2d.silo”

1. Define the expression “gorf” with definition “PointVar + shepardglobal”.
   - Note that PointVar is defined on a mesh named PointMesh while shepardglobal is defined on a mesh named Mesh.

2. Try to do a Pseudocolor plot of “gorfo”. You will get a plot of points and a warning message like this one…
   The compute engine running on host somehost.com issued the following warning: In domain 0, your nodal variable “shepardglobal” has 2500 values, but it should have 100. Some values were removed to ensure VisIt runs smoothly.

So, what’s happening here? VisIt is deciding to perform the summation operation on the PointVar’s mesh. That mesh consists of 100 points. So, when it encounters the shepardglobal variable (defined on Mesh with 50x50 nodes), it simply ignores any values in “shepardglobal” after the first 100. Most likely, this is not the desired outcome.

We have two options each of which involves mapping one of the variables onto the other variable’s mesh using one of the CMFE expression functions. We can map shepardglobal onto PointMesh or we can map PointVar onto Mesh. We’ll do both here.

**Mapping shepardglobal onto PointMesh**

1. Define a new expression named “shepardglobal_mapped”.
2. Go to Insert Function…, then to the Comparisons submenu and select pos_cmfe.
   - This defines a position based cross-mesh field evaluation function. The other option is a conn_cmfe or connectivity-based which is faster but requires both meshes to be topologically congruent and is not appropriate here.
3. A template for the arguments to the pos_cmfe will appear in the Definition pane.
4. Replace “<filename:var>” with “<./noise2d.silo:shepardglobal>”.
   - This assumes the “noise2d.silo” file is in the same directory from which VisIt was started.
   - This defines the source or donor variable to be mapped onto a new mesh.
5. Replace “<meshname>” with “PointMesh”.
   - This defines the destination or target mesh the variable is to be mapped onto.
6. Replace “<fill-var-for-uncovered-regions>” with “-1”.
   - This is needed for position-based CMFE’s because the donor variable’s mesh and target mesh may not always volumetrically overlap 100%. In places where this winds up being the case, VisIt will use this value to fill in.
7. Now with “shepardglobal_mapped” defined, you can define the desired expression, “PointVar + shepardglobal_mapped” and this will achieve the desired result and is shown below.

**Mapping PointVar onto Mesh**

To be completed. But, cannot map point mesh onto a volumetric mesh. VisIt always returns zero overlap.

**Combining expressions and queries is powerful**

Suppose you have a database generated by some application code simulating some object being blown apart. Maybe its a 2D, cylindrically symmetric calculation. Next, suppose the code produced a “density” and “velocity” variable. However, what you want to compute is the total mass of some (portion of) the object that has velocity (magnitude)
Fig. 3.21: The variable Shepardglobal mapped onto a point mesh
greater than some threshold, say 5 meters/second. You can use a combination of Expressions, Queries and the Threshold operator to achieve this.

Mass is “density * volume”. You have a 2D mesh, so how do you get volume from something that has only 2 dimensions? You know the mesh represents a calculation that is cylindrically symmetric (revolved around the y-axis). You can use the revolved_volume() Expression function to obtain the volume of each zone in the mesh. Then, you can multiply the result of revolved_volume() by density to get mass of each zone in the mesh. Once you have that, you can use threshold operator to display only those zones with velocity (magnitude) greater than 5 and then a variable sum query to add up all the mass moving at that velocity.

Here, we demonstrate the steps involved using the “noise2d.silo” database. Because that database does not quite match the problem assumption described in the preceding paragraphs, we simply re-purpose a few of the variables in the database to serve as our density and velocity variables in this example. Namely, we define the expression density as an alias for shepardglobal and velocity as an alias for grad.

Here are the steps involved.

1. Go to Controls->Expressions.
2. Click New.
3. Set the Name to “density”.
4. Make sure the Type is set to Scalar mesh variable.
5. Set the Definition to “shepardglobal”.
6. Click Apply.
7. Click New.
8. Set the Name to “velocity”.
9. Make sure the Type is set to Vector mesh variable.
10. Set the Definition to “grad”.
11. Click Apply.
12. Click New.
13. Set the Name to “mass”.
14. Make sure the Type is set to Scalar mesh variable.
15. Set the Definition to “revolved_volume(Mesh) * density”.
16. Click Apply.
17. Click the New button again (for a new expression).
18. Set the Name to “velmag” (for velocity magnitude).
19. Set the Definition to “magnitude(velocity)”.
20. Go to Plot->Pseudocolor->mass.
21. Click Draw.
22. Add Operator->Threshold.
23. Open the Threshold operator attributes window.
24. Select the default variable and then click Delete selected variable.
25. Go to Add Variable and select velmag from the list of Scalars.
26. Set Lower Bound to “5”.
27. Click Apply.
   • Now the displayed plot changes to show only those parts of the mesh that are moving with velocity greater than 5.
28. Go to Controls->Query.
29. Find the Variable sum query from the list of queries.
30. Click the Query button. The computed result will be a sum of all the individual zones’ masses in the mesh for those zones that are moving with velocity greater than 5.

**Automatic, saved and database expressions**

*VisIt* defines several types of expressions automatically. For all vector variables from a database, *VisIt* will automatically define the associated magnitude expressions. For unstructured meshes, *VisIt* will automatically define mesh quality expressions. For any databases consisting of multiple time states, *VisIt* will define time derivative expressions. This behavior can be controlled by going to *VisIt’s Preferences* dialog and enabling or disabling various kinds of automatic expressions.

If you save settings, any expressions you have defined are also saved with the settings. And, they will appear (and sometimes pollute) your menus whether or not they are valid expressions for the currently active database.

Finally, databases are also free to define expressions. In fact, many databases define a large number of expressions for the convenience of their users who often use the expressions in their post-processing workflows. Ordinarily, you never see *VisIt’s* automatic expressions or a database’s expressions in the Expression window because they are not editable. However, you can check the display expressions from database check box in the Expressions window and *VisIt* will also show these expressions.
Fig. 3.23: Mass plot
Fig. 3.24: Threshold attributes
Fig. 3.25: Mass plot after threshold
Fig. 3.26: The variable sum query result
3.3 Scripting

This section describes the VisIt Command Line Interface (CLI).

3.3.1 Command line interface overview

VisIt includes a rich command line interface that is based on Python 2.7.

There are several ways to use the CLI:

1) Launch VisIt in a batch mode and run scripts.
   • Linux: /path/to/visit/bin/visit -nowin -cli -s <script.py>
   • OSX: /path/to/VisIt.app/Contents/Resources/bin/visit -nowin -cli -s <script.py>

2) Launch VisIt so that a visualization window is visible and interactively issue CLI commands.

3) Use both the standard GUI and CLI simultaneously.

3.3.2 Launching the CLI

We will focus on the use case where we have the graphical user interface and CLI running simultaneously.

To launch the CLI from the graphical user interface:

1) Go to Controls->Command.

This will bring up the Commands window. The Command window provides a text editor with Python syntax highlighting and an Execute button that tells VisIt to execute the script. Finally, the Command window lets you record your GUI actions into Python code that you can use in your scripts.

3.3.3 A first action in the CLI

1) Open “example.silo” in the GUI if it not already open.

2) Cut-and-paste the following Python commands into the first tab of the Commands window.

AddPlot("Pseudocolor", "temp")
‘You will see the active plots list in the GUI update, since the CLI and GUI communicate.
DrawPlots()’
‘You should see your plot.

3) Click Execute.

3.3.4 Tips about Python

1) Python is whitespace sensitive! This is a pain, especially when you are cut-n-pasting things.

2) Python has great constructs for control and iteration, here are some examples:
for i in range(100):
    # use i
    # strided range
for i in range(0, 100, 10):
    # use i
if (cond):
    # stmt
import sys
...
sys.exit()

3.3.5 Example scripts

We will be using Python scripts in each of the following sections: You can get execute them by:

1) Cut-n-paste-ing them into a tab in the Commands window and executing it.

For all of these scripts, make sure “example.silo” is currently open unless otherwise noted.

Setting attributes

Each of VisIt’s Plots and Operators expose a set of attributes that control their behavior. In VisIt’s GUI, these attributes are modified via options windows. VisIt’s CLI provides a set of simple Python objects that control these attributes. Here is an example setting the minimum and maximum for the Pseudocolor plot.

```python
DeleteAllPlots()
AddPlot("Pseudocolor", "temp")
DrawPlots()
p = PseudocolorAttributes()
p.minFlag = 1
p.maxFlag = 1
p.min = 3.5
p.max = 7.5
SetPlotOptions(p)
```

Animating an isosurface

This example demonstrates sweeping an isosurface operator to animate the display of a range of isovalues from “example.silo”.

```python
DeleteAllPlots()
AddPlot("Pseudocolor", "temp")
iso_att = IsosurfaceAttributes()
iso_att.contourMethod = iso_att.Value
iso_att.variable = "temp"
AddOperator("Isosurface")
DrawPlots()
for i in range(30):
    iso_att.contourValue = (2 + 0.1*i)
SetOperatorOptions(iso_att)
```

(continues on next page)
# For moviemaking, you'll need to save off the image
# SaveWindow()

Using all of VisIt's building blocks

This example uses a Pseudocolor plot with a ThreeSlice operator applied to display temp on the exterior of the grid along with streamlines of the gradient of temp.

Note that the script below may not work the first time you execute it. In that case delete all the plots and execute the script again.

```plaintext
# Clear any previous plots
DeleteAllPlots()

# Create a plot of the scalar field 'temp'
AddPlot("Pseudocolor","temp")
```

Fig. 3.27: Streamlines
# Slice the volume to show only three
# external faces.
AddOperator("ThreeSlice")
tatts = ThreeSliceAttributes()
tatts.x = -10
tatts.y = -10
tatts.z = -10
SetOperatorOptions(tatts)
DrawPlots()

# Find the maximum value of the field 'temp'
Query("Max")
val = GetQueryOutputValue()
print "Max value of 'temp' = ", val

# Create a streamline plot that follows
# the gradient of 'temp'
DefineVectorExpression("g","gradient(temp)")
AddPlot("Pseudocolor", "operators/IntegralCurve/g")
iatts = IntegralCurveAttributes()
iatts.sourceType = iatts.SpecifiedBox
iatts.sampleDensity0 = 7
iatts.sampleDensity1 = 7
iatts.sampleDensity2 = 7
iatts.dataValue = iatts.SeedPointID
iatts.integrationType = iatts.DormandPrince
iatts.issueStiffnessWarnings = 0
iatts.issueCriticalPointsWarnings = 0
SetOperatorOptions(iatts)

# set style of streamlines
patts = PseudocolorAttributes()
patts.lineType = patts.Tube
patts.tailStyle = patts.Spheres
patts.headStyle = patts.Cones
patts.endPointRadiusBBox = 0.01
SetPlotOptions(patts)
DrawPlots()

Creating a movie of animated streamline paths

This example extends the “Using all of VisIt’s Building Blocks” example by

- animating the paths of the streamlines
- saving images of the animation
- finally, encoding those images into a movie

(Note: Encoding requires ffmpeg is installed and available in your PATH)

# import visit_utils, we will use it to help encode our movie
from visit_utils import *

# Set a better view
ResetView()
v = GetView3D()
v.RotateAxis(0,44)
v.RotateAxis(1,-23)
SetView3D(v)

# Disable annotations
aatts = AnnotationAttributes()
aatts.axes3D.visible = 0
aatts.axes3D.triadFlag = 0
aatts.axes3D.bboxFlag = 0
aatts.userInfoFlag = 0
aatts.databaseInfoFlag = 0
aatts.legendInfoFlag = 0
SetAnnotationAttributes(aatts)

# Set basic save options
swatts = SaveWindowAttributes()
# The 'family' option controls if visit automatically adds a frame number to
# the rendered files. For this example we will explicitly manage the output name.
#
swatts.family = 0
#
swatts.format = swatts.PNG
#
swatts.width = 1024
#
swatts.height = 1024

####
# Crop streamlines to render them at increasing time values over 50 steps
####
iatts.cropValue = iatts.Time
iatts.cropEndFlag = 1
iatts.cropBeginFlag = 1
iatts.cropBegin = 0
for ts in range(0,50):
    # set the integral curve attributes to change the where we crop the streamlines
    iatts.cropEnd = (ts + 1) * .5

    # update streamline attributes and draw the plot
    SetOperatorOptions(iatts)
    DrawPlots()
    #before we render the result, explicitly set the filename for this render
    swatts.fileName = "streamline_crop_example_%04d.png" % ts
    SetSaveWindowAttributes(swatts)
    # render the image to a PNG file
    SaveWindow()

################
# use visit_utils.encoding to encode these images into a "wmv" movie
Rendering each time step of a dataset to a movie

This example assumes the "aneurysm.visit" is already opened.

- Create a plot, render all timesteps and encode a movie.

(Note: Encoding requires that ffmpeg is installed and available in your PATH)
# swatts.format = swatts.PNG
# set the width of the output image
# swatts.width = 1024
# set the height of the output image
# swatts.height = 1024

# the encoder expects file names with an integer sequence
# 0,1,2,3 .... N-1

file_idx = 0

for ts in range(0, nts, 10):  # look at every 10th frame
    # Change to the next timestep
    TimeSliderSetState(ts)
    # before we render the result, explicitly set the filename for this render
    swatts.fileName = "blood_flow_example_%04d.png" % file_idx
    SetSaveWindowAttributes(swatts)
    # render the image to a PNG file
    SaveWindow()
    file_idx += 1

##############
# use visit_utils.encoding to encode these images into a "wmv" movie
#
# The encoder looks for a printf style pattern in the input path to identify the frames of the movie.
# The frame numbers need to start at 0.
#
# The encoder selects a set of decent encoding settings based on the extension of the output movie file (second argument). In this case we will create a "wmv" file.
#
# Other supported options include ".mpg", ".mov".
# "wmv" is usually the best choice and plays on all most all platforms (Linux , OSX, Windows).
# "mpg" is lower quality, but should play on any platform.
#
# 'fdup' controls the number of times each frame is duplicated.
# Duplicating the frames allows you to slow the pace of the movie to something reasonable.
#
##############

input_pattern = "blood_flow_example_%04d.png"
output_movie = "blood_flow_example.wmv"
encoding.encode(input_pattern, output_movie, fdup=4)

Animating the camera

See Python fly through.
Automating data analysis

See Python analysis curves.

Extracting a per-material aggregate value at each timestep

See example Python aggregate curves.

3.3.6 Recording GUI actions to Python scripts

VisIt’s Commands window provides a mechanism to translate GUI actions into their equivalent Python commands.

1) Open the Commands Window by selecting “Controls Menu->Command”

2) Click the Record button.

3) Perform GUI actions.

4) Return to the Commands Window.

5) Select a tab to hold the python script of your recorded actions.

Fig. 3.28: The Commands window

2) Click the Record button.

3) Perform GUI actions.

4) Return to the Commands Window.

5) Select a tab to hold the python script of your recorded actions.
6) Click the Stop button.

7) The equivalent Python script will be placed in the tab in the Commands window.
   • Note that the scripts are very verbose and contain some unnecessary commands, which can be edited out.

3.3.7 Learning the CLI

Here are some tips to help you quickly learn how to use VisIt’s CLI:

1) From within VisIt’s python CLI, you can type “dir()” to see the list of all commands.
   • Sometimes, the output from “dir()” within VisIt’s python CLI is a little hard to look through. So, a useful thing on Linux to get a nicer list of methods is the following shell command (typed from outside VisIt’s python CLI)...

   ```bash
   echo "dir()" | visit -cli -nowin -forceinteractivecli | tr ',' '
' | tr -d " '"
   →| sort
   ```
   • Or, if you are looking for CLI functions having to do with a specific thing...

   ```bash
   echo "dir()" | visit -cli -nowin -forceinteractivecli | tr ',' '
' | tr -d " '"
   →| grep -i material
   ```

2) You can learn the syntax of a given method by typing “help(MethodName)”
   • Type “help(AddPlot)” in the Python interpreter.

3) Use the GUI to Python recording featured outlined in Recording GUI actions to Python scripts.

4) Use “WriteScript()” function, which will create a python script that describes all of your current plots.
   • For more details, see WriteScript.

5) When you have a Python object, you can see all of its attributes by printing it.

```python
s = SliceAttributes()
print s
# Output:
originType = Intercept  # Point, Intercept, Percent, Zone, Node
originPoint = (0, 0, 0)
originIntercept = 0
originPercent = 0
originZone = 0
originNode = 0
normal = (0, -1, 0)
axisType = YAxis       # XAxis, YAxis, ZAxis, Arbitrary, ThetaPhi
upAxis = (0, 0, 1)
project2d = 1
interactive = 1
flip = 0
originZoneDomain = 0
originNodeDomain = 0
meshName = "default"
theta = 0
phi = 0
```
Tips for searching for help

VisIt’s CLI provides a large set of functions. To can limit the scope of your search using a helper functions. One such helper is the `lsearch()` function in the visit_utils module:

```python
from visit_utils.common import lsearch
lsearch(dir(), "Material")
```

`lsearch()` returns a python list of strings with the names that match the given pattern. Here is another example that prints each of the result strings on a separate line.

```python
from visit_utils.common import lsearch
for value in lsearch(dir(), "Material"):  
    print value
```

3.3.8 Advanced features

1) You can set up your own buttons in the VisIt gui using the CLI. See visitrc file.
2) You can set up callbacks in the CLI that get called whenever events happen in VisIt. See Python callbacks.
3) You can create your own custom Qt GUI that uses VisIt for plotting. See PySide recipes.

3.4 Aneurysm

This tutorial provides a short introduction to VisIt’s features while exploring a finite element blood flow simulation of an aneurysm. The simulation was run using the LifeV finite element solver and made available for this tutorial thanks to Gilles Fourestey and Jean Favre, Swiss National Supercomputing Centre.

3.4.1 Open the dataset

1. Click on the Open icon to bring up the File open window.
2. Navigate your file system to the folder containing “aneurysm.visit”.
3. Highlight the file “aneurysm.visit” and then click OK.

3.4.2 Plotting the mesh topology

First we will examine the finite element mesh used in the blood flow simulation.

Create a Mesh plot

1. Go to Add->Mesh->Mesh.
2. Click Draw.

After this, the mesh plot is rendered in VisIt’s Viewer window. Modify the view by rotating and zooming in the viewer window.
Modify the Mesh plot settings

1. Double click on the Mesh plot to open the Mesh plot attributes window.
2. Experiment with settings for:
   - Mesh color
   - Opaque color
   - Opaque mode
     - When the Mesh plot’s opaque mode is set to automatic, the Mesh plot will be drawn in opaque mode unless it is forced to share the visualization window with other plots, at which point the Mesh plot is drawn in wireframe mode. When the Mesh plot is drawn in wireframe mode, only the edges of each externally visible cell face are drawn, which prevents the Mesh plot from interfering with the appearance of other plots. In addition to having an automatic opaque mode, the Mesh plot can be forced to be drawn in opaque mode or wireframe mode by selecting the On or Off. This is best demonstrated with the Pseudocolor plot of pressure present.
   - Show internal zones
     - You will need to click Apply to commit the settings to your plot.

Query the mesh properties

VisIt’s Query interface provides several quantitative data summarization operations. We will use the query interface to learn some basic information about the simulation mesh.

1. Go to Controls->Query to bring up the Query window.
2. Select NumZones and click Query.
   - This returns the number of elements in the mesh.
3. Select NumNodes and click Query.
   - This returns the number of vertices in the mesh.

Note: The terms “zones”, “elements”, and “cells” are overloaded in scientific visualization, as are the terms “nodes”, “points”, and “vertices.”
Fig. 3.30: The mesh plot attributes window.
Fig. 3.31: The mesh plot of the aneurysm.
3.4.3 Examining scalar fields

In addition to the mesh topology, this dataset provides two mesh fields:

- A scalar field “pressure”, associated with the mesh vertices.
- A vector field “velocity”, associated with the mesh vertices.

*VisIt* automatically defines an expression that allows us to use the magnitude of the “velocity” vector field as a scalar field on the mesh. The result of the expression is a new field named “velocity_magnitude”.

We will use Pseudocolor plots to examine the “pressure” and “velocity_magnitude” fields.

1. Go to Add->Pseudocolor->Pressure.
2. Click Draw.
3. Double click on the Pseudocolor plot to bring up the Pseudocolor plot attributes window.
4. Change the color table to Spectral and check the Invert button.
5. Click Apply.
6. Click Draw.
7. Click Play in the Time animation controls above the plot list on the main GUI window.

You will see the pressure field animate on the exterior of the mesh as the simulation evolves.

Experiment with:

- Setting the Pseudocolor plot limits.
- Hiding and showing the Mesh plot.

When you are done experimenting, stop animating over time steps using the Stop button.

Query the maximum pressure over time

We can use the “pressure” field to extract the heart beat signal. We want to find the maximum pressure value across the mesh elements at each time step of our dataset. *VisIt* provides a Query over time mechanism that allows us to extract this data.

First, we need to set our query options to use timestep as the independent variable for our query.

1. Go to Controls->Query over time options.
2. Select Timestep.
3. Click Apply and Dismiss.

Now we can execute the Max query on all of our time steps and collect the results into a curve.

1. Click on the Pseudocolor plot to make sure it is active.
Fig. 3.32: The pseudocolor plot of the pressure.
2. Go to Controls->Query to bring up the Query window.
3. Select Max.
4. Check Do Time Query.
5. Click Query.

This will process the simulation output files and create a new window with a curve that contains the maximum pressure value at each time.

Additional exercises

- How many heart beats does this dataset cover?
- Estimate the number of beats per minute of the simulated heart (each cycle is 0.005 seconds).

### 3.4.4 Contours and sub-volumes of high velocity

Examining the velocity magnitude

Next we create a Pseudocolor plot to look at the magnitude of the “velocity” vector field.

1. Delete all your existing plots by selecting them all and clicking Delete.
2. Go to Add->Pseudocolor->velocity_magnitude.
3. Open the Pseudocolor plot attributes window and set the color table options as before.
4. Click Draw.

Notice that the velocity at the surface of the mesh is zero. To get a better understanding of the flow inside the mesh, we will use operators to extract regions of high blood flow.
Fig. 3.34: The query over time plot.
Fig. 3.35: The pseudocolor plot of the velocity magnitude.
Creating a semi-transparent exterior mesh plot

When looking at features inside the mesh, it helps to have a partially transparent view of the whole mesh boundary for reference. We will add a Subset plot to create this view of the mesh boundary.

1. Uncheck *Apply operators to all plots*.

   ![Apply operators to all plots](image)

   Fig. 3.36: The apply operators to all plots setting.

2. Go to *Add->Subset->Mesh*.
3. Open the Subset plot attributes window.
4. Change the color to *Light Blue*.
5. Set the *Opacity* slider to 25%.
6. Click *Apply*.
7. Click *Draw*.

Contours of high velocity

Now we will extract contour surfaces at high velocity values using the Isosurface operator.

1. Select the Pseudocolor plot in the plot list.
2. Go to *Operators->Slicing->Isosurface*.
3. Open the Isosurface operator attributes window.
4. Set *Select by* to *Value*, and use “10 15 20”.
5. Click *Apply* and *Dismiss*.
6. Click *Draw* and press the *Play* button to animate the plot over time.

You will see the contour surfaces extracted from the “velocity_magnitude” field animate as the simulation evolves.

Sub-volumes of high velocity

As an alternative to contours, we can also extract the sub-volume between two scalar values using the Isovolume operator.

1. Click *Stop* to stop the animation.
2. Remove the Isosurface operator.
3. Go to *Operators->Selection->Isovolume*.
4. Open the Isovolume operator attributes window.
5. Set the *Lower bound* to “10” and the *Upper Bound* to “20”.
6. Click *Apply* and *Dismiss*.
7. Click *Draw* and press the *Play* button to animate the plot over time.
Fig. 3.37: The transparent subset plot.
Fig. 3.38: The transparent subset plot with iso surfaces of velocity magnitude.
Fig. 3.39: The transparent subset plot with an iso volume of velocity magnitude.
3.4.5 Visualizing the velocity vector field

This section of the tutorial outlines using glyphs, streamlines, and pathlines to visualize the velocity vector field from the simulation.

**Plotting the vector field directly with glyphs**

VisIt’s Vector plot renders a vector field at each time step as a collection of arrow glyphs. This allows us to see the direction of the vectors as well as their magnitude. We will create a vector plot to directly view the simulated “velocity” vector field.

1. Go to Add->Vector->velocity.
2. Open the Vector plot attributes window.
3. Go to the Vectors tab.
4. Set Stride to “5”.
5. Go to the Color section on the Data tab.
6. Change the Magnitude to Spectral, and check the Invert option.
7. Go to the Glyphs tab.
8. In the Scale section, set the Scale to “0.5”.
9. In the Style section, set Arrow body to Cylinder.
10. In the Rendering section, set Geometry Quality to High.
11. Click Apply and Dismiss.
12. Click Draw.
13. Click Play.

**Examining features of the flow field with streamlines**

To explore the flow field further we will seed and advect a set of streamlines near the inflow of the artery. Streamlines show the path massless tracer particles would take if advected by a static vector field. To construct Streamlines, the first step is selecting a set of spatial locations that can serve as the initial seed points.

We want to center our seed points around the peak velocity value on a slice near the inflow of the artery. To find this location, we query a sliced pseudocolor plot of the “velocity_magnitude”.

1. Go to Add->Pseudocolor->velocity_magnitude.
2. Open the Pseudocolor plot attributes window and set the color table options as before.
3. Go to Operators->Slicing->Slice.
4. Open the Slice operator attributes window.
5. In the Normal section set Orthogonal to Y Axis.
6. In the Origin section select Point and set the value to “3 3 3”.
7. In the Up Axis section uncheck Project to 2D.
8. Click Apply and Dismiss.
9. Click Draw.
Fig. 3.40: The vector plot of velocity.

3.4. Aneurysm
Fig. 3.41: The velocity magnitude on a slice.
Query to find the maximum velocity on the slice

1. Click to make sure the Pseudocolor plot of your “velocity_magnitude” slice is active.
2. Go to Controls->Query.
3. Select Max.
4. Select Actual Data.
5. Click Query.

This will give you the maximum scalar value on the slice and the x,y,z coordinates of the node associated with this value. We will use the x,y,z coordinates of this node to seed a set of streamlines.

Plotting streamlines of velocity

1. Go to Add->Pseudocolor->operators->IntegralCurve->velocity.
2. Open the IntegralCurve operator attributes window.

3.4. Aneurysm
Fig. 3.43: Creating a streamline plot with the IntegralCurve operator.
3. Go to the Source section on the Integration tab.
4. Set the Source type to Circle.
5. Set the Origin to the value returned from the max query: “3.45115 3 5.54927”, excluding any commas in the input text box.
6. Set the Normal to the y-axis: “0 1 0”.
7. Set the Up axis to the z-axis: “0 0 1”.
8. Set the Radius to “0.12”.
9. Go to the Sampling section.
10. Set Sampling along: to Boundary.
11. Set Samples in Theta: to “12”.
12. Go to the Advanced tab.
13. In the Warnings section, uncheck all of the warning checkboxes.
14. Click Apply and Dismiss.

Fig. 3.44: The IntegralCurve operator attributes.

15. Open the Pseudocolor plot attributes window.
16. Go to the Data tab.
17. In the Color section set the Color table to Reds.
18. Go to the Line section on the Geometry tab.
19. Set Line type to Tubes.
20. Set Tail to Sphere.
21. Set Head to Cone.
22. Set the head and tail Radius to “0.03”.

3.4. Aneurysm
Fig. 3.45: The Pseudocolor attributes for the streamline data.
Fig. 3.46: The Pseudocolor attributes for the streamline geometry.
23. Click *Apply* and *Dismiss*.

24. Click *Draw*.

25. Use the time slider controls to view a few time steps.

![Window 1](image)

Fig. 3.47: The streamlines of velocity.

Finally, to explore the time varying behavior of the flow field we will use pathlines. Pathlines show the path massless tracer particles would take if advected by the vector field at each timestep of the simulation.

We will modify our previous *IntegralCurve* options to create pathlines.

1. Set the time slider controls to the first timestep.
2. Open the *IntegralCurve* attributes window.
3. Go to the Appearance tab.
4. In the Streamlines vs Pathlines section select Pathline.
5. In the Pathlines Options section set How to perform interpolation over time to Mesh is static over time.

![Streamlines vs Pathlines](image)

Fig. 3.48: The IntegralCurve operator pathline attributes.

6. Click Apply and Dismiss.

This will process all 200 files in the dataset and construct the pathlines that originate at our seed points.

### 3.4.6 Publishing to SeedMe.org

**Required setup**

- Sign-in or Sign-up at SeedMe.org.
- Download your “API Key file”, then move it to your Home directory.

**Sharing automation script**

In this section we will render and save pathline trace in 20 steps. Then upload and share the rendered 20 images as a sequence and instruct SeedMe to encode a video from these set of images at 2 frames per second. A sample video can be seen [here](#).

A detailed example with a brief explanation in the comments can be seen [here](#).

1. Go to Controls->Command.
2. Find an empty tab.
3. Paste the following Python snippet into this tab.

---

3.4. Aneurysm 735
Fig. 3.49: The pathlines of velocity.
# file: aneurysm_seedme.py
# info:
# Example showing how to use SeedMe to publish a pathline animation
#
##############################################################################
# To do : Set the following four variables
##############################################################################
seedme_apikey_path = '/absolute/path/to/seedme.txt'
my_rendered_image_path = '/absolute/path/for/images/'  # does not traverse recursively
my_content_privacy = "public"  # private (default), group, public
my_share_list = "one@example.com, two@example.com"  # comma delimited emails

# Set save window attributes including path where the rendered images will be saved
sa = SaveWindowAttributes()
sa.outputToCurrentDirectory = 0
sa.outputDirectory = my_rendered_image_path
sa.fileName = "pathline"
sa.family = 1
sa.format = sa.PNG
sa.width = 512
sa.height = 512
sa.screenCapture = 0
sa.saveTiled = 0
sa.quality = 80
sa.progressive = 0
sa.binary = 0
sa.stereo = 0
sa.compression = sa.PackBits  # None, PackBits, Jpeg, Deflate
sa.forceMerge = 0
sa.resConstraint = sa.ScreenProportions  # NoConstraint, EqualWidthHeight, ScreenProportions
sa.advancedMultiWindowSave = 0
SetSaveWindowAttributes(sa)

# Now save this pathline visualization in 20 frames (images)
# Animate our pathlines by cropping based on time
iatts = IntegralCurveAttributes()
iatts.cropValue = iatts.Time
iatts.cropEndFlag = 1
nsteps = 20  # Number of steps
final_time = .995
for i in range(nsteps+1):
    iatts.cropEnd = final_time * i / nsteps
SetOperatorOptions(iatts)
SaveWindow()  # will save images at the sa.outputDirectory provided above

# Upload and share content at SeedMe.org
# Instruct the seedme module to upload 20 images then encode a video from it
# seedme module ships with VisIt 2.9.x +
# --------------------------------------------------------------------------------

```python
import seedme

# Set path to the APIKey file
obj.set_auth_via_file(seedme_apikey_path)

# Create a dictionary for rendered image sequence
my_seq = {
    "filepath": my_rendered_image_path,
    "title": "Pathline",
    "description": "Pathlines show the path massless tracer particles would take if advected by the vector field at each timestep of the simulation.",
    "fps": 2,
    "encode": True,
}

# Create seedme object
obj = seedme.SeedMe()

# Create a new collection using create_collection method
# composed with title and sequence with public access, shared with two people
result = obj.create_collection(title="Aneurysm vis",
                               privacy=my_content_privacy,  # string = One of
                               sharing=my_share_list,  # string = Comma delimited
                               notify=True,  # Boolean = False(default) send email
                               sequences=my_seq,  # upload sequence
                               )

# create_collection returns the result as a string in json format
print(result)

url = obj.get_url(result)
# Visit this url on your web browser
print("\n\nThe url for this collection is: " + url)
```

4. Click Execute.

To view your shared content login to SeedMe.org then navigate to My collections.

To learn more about the SeedMe Python API review the example demo.py.

### 3.4.7 Calculating the flux of a velocity field through a surface

To calculate a flux, we will need the original velocity vector, the normal vector of the surface, and VisIt’s Flux Operator. We will calculate the flux through a cross-slice located at Y=3, at the beginning of the artery.
Creating the slice and showing velocity glyphs

First we will directly plot the velocity vectors that exist on the slice through the 3D mesh.

1. Delete any existing plots.
2. Go to Add->Vector->velocity.
3. Open the Vector plot attributes window.
4. Go to the Vectors tab and set the Fixed number to “40”.
5. Go to the Glyphs tab.
6. Set Arrow body to Cylinder.
7. Set Geometry Quality to High.

8. Click Apply and Dismiss.
9. Go to Operators->Slicing->Slice.
10. Open the Slice operator attributes window.
11. Set Normal to Arbitrary and to “0 1 0”.
12. Set Origin to Intercept and to “3”.
13. Uncheck Project to 2D.
14. Click Make default, Apply and Dismiss.

Fig. 3.50: The Vector plot attributes.

3.4. Aneurysm
15. Click *Draw*.

16. Go to *Add*->*Pseudocolor*->*velocity_magnitude*.

17. Open the Pseudocolor plot attributes window.

18. Set *Limits* to *Use Current Plot*.

19. Click *Apply* and *Dismiss*.

20. Go to *Operators*->*Slicing*->*Slice*.

21. Click *Draw*.

22. Go to *Add*->*Mesh*->*Mesh*.

23. Open the Mesh plot attributes window.

24. Set *Mesh color* to *Custom* and select a medium grey color.

25. Click *Apply* and *Dismiss*.

26. Click *Draw*.

27. Zoom in to explore the plot results.

Fig. 3.51: The Slice operator attributes.

In order to give some context to the Vector plot of velocity on the slice let's add a Pseudocolor plot of velocity_magnitude on the same slice and a Mesh plot.
Fig. 3.52: The velocity on the slice.
The Vector plot uses glyphs to draw portions of the instantaneous vector field. The arrows are colored according to the speed at each point (the magnitude of the velocity vector). Next we create an expression to evaluate the vectors normal to the Slice. These normals should all point in the Y direction.

**Creating a vector expression and using the DeferExpression operator**

We will use VisIt’s pre-defined expression to evaluate the normals on a cell-by-cell basis.

1. Go to Controls->Expressions.
2. Click New.
3. Change the Name to “normals” and the Type to Vector mesh variable.
4. Go to Insert function->Miscellaneous->cell_surface_normal in the Standard editor tab.
5. Go to Insert variable->Mesh->Mesh in the Standard editor tab.

![Fig. 3.53: The Expressions window.](image)

6. Click Apply and Dismiss.
7. Return to the Vector plot and change its variable to “normals”.

You will then get the error message saying: *The ‘normals’ expression failed because The Surface normal expression can only be calculated on surfaces. Use the ExternalSurface operator to generate the external surface of this object. You must also use the DeferExpression operator to defer the evaluation of this expression until after the external surface operator. In fact, VisIt cannot use the name Mesh which refers to the original 3D mesh. It needs to defer the evaluation until after the Slice operator is applied. Thus, we need to add the Defer Expression operator.*
8. Go to Operators->Analysis->DeferExpression.
9. Open the DeferExpression operator attributes window.
10. Go to Variables->Vector->normals.

![DeferExpression operator attributes window](image)

Fig. 3.54: The DeferExpression window.

11. Click Apply and Dismiss.
12. Click Draw.
13. Verify that all your normals point in the up (Y) direction.

Calculating the flux on the slice

We are now ready for the final draw.

1. Go to Add->Pseudocolor->operators->Flux->Mesh.
2. Go to Operators->Slicing->Slice.
3. Open the Slice operator attributes window.
4. Verify that the default values previously saved are used.
5. Move the Slice operator above the Flux operator.
6. Go to Operators->Analysis->DeferExpression.
7. Move the DeferExpression operator above the Flux operator just below the Slice operator.
8. Open the Flux operator attributes window.
9. Set the Flow field to “velocity”.
10. Click Apply and Dismiss.
11. Click Draw.

Verify that you have a display that is cell-centered, and that will vary with the Time slider
12. Get the numerical value of the flux by query-ing for the Weighted Variable Sum.

### 3.5 Potential Flow

This tutorial demonstrates VisIt’s features while exploring results from simple simulations of potential flow around an obstruction, specifically an airfoil. Potential flow assumes irrotational flow. That is, there is no rotational motion in the flow, no vortices or eddies. This assumption is valid for low velocities and certain types of gases/fluids and
Fig. 3.55: The Vector plot of the normals.

Fig. 3.56: The Flux operator attributes window.
Fig. 3.57: The Vector plot of the flux.
Fig. 3.58: The result of the Weighted Variable Sum query.
obstructions. When the flow does involve rotation, a more complex solution involving Navier-Stokes equations is required.

The potential flow solver is a mini-app developed using the MFEM finite element library. The example is available for this tutorial thanks to Aaron Fischer and Mark Miller of LLNL. The data set includes VTK output files for a set of solutions where the angle of attack of the velocity varies from -5 degrees to 25 degrees.

3.5.1 Open the dataset

1. Click on the Open icon to bring up the File open window.
2. Navigate your file system to the folder containing “potential_flow_ang_sweep.visit”.
3. Highlight the file “potential_flow_ang_sweep.visit” and then click OK.

3.5.2 Plotting the mesh topology

First we will examine the mesh used by the solver.

Create a Mesh plot

1. Go to Add->Mesh->main.
2. Click Draw.

After this, the mesh plot is rendered in VisIt’s Viewer window. This is a 2D mesh, modify the view by planning and zooming in the viewer window. Zoom in near the airfoil and look at the mesh structure.

Modify the Mesh plot settings

1. Double click on the Mesh plot to open the Mesh plot attributes window.
2. Experiment with settings for:
   - Mesh color
   - Opaque color
   - Opaque mode

You will need to click Apply to commit the settings to your plot.

3.5.3 Examining the velocity magnitude

In addition to the mesh topology, this dataset provides a vector field “v”, representing the velocity, associated with the mesh vertices.

VisIt automatically defines an expression that allows us to use the magnitude of the “v” vector field as a scalar field on the mesh. The result of the expression is a new field named “v_magnitude”.

We will use Pseudocolor plots to examine the “pressure” and “velocity_magnitude” fields.

1. Go to Add->Pseudocolor->v_magnitude.
2. Click Draw.
3. Double click on the Pseudocolor plot to bring up the Pseudocolor plot attributes window.
Fig. 3.59: Example mesh plot settings for the Potential Flow data.
Fig. 3.60: Example mesh plot result the Potential Flow data.
4. In the **Limits** section, enable the **Maximum** checkbox and set the limit to 1.

5. In the **Color** section, change the color table to **Spectral** and check the **Invert** button.

6. Click **Apply**.

7. Click **Draw**.

8. Drag the **Time** animation controls above the plot list on the main **GUI** window.

   You will see the velocity magnitude solutions for the different angles of attack.

   Experiment with the **Color for values > max** option to see where the range is being clipped.

<table>
<thead>
<tr>
<th>Contours</th>
<th>of</th>
<th>velocity</th>
<th>magnitude</th>
</tr>
</thead>
</table>

Now we will add an additional plot to view velocity magnitude contours

1. Go to **Add->Contour->v_magnitude**.

2. Double click on the Contour plot to bring up the Contour plot attributes window.

3. In the **Contour Levels** section, enable the **Maximum** checkbox and set the limit to 1.

4. In the **Lines** section, set the **Line width** to 2.

5. Click **Apply**.

6. Click **Draw**.

7. Drag the **Time** animation controls above the plot list on the main **GUI** window.

   You will see the contours of the velocity magnitude solutions for the different angles of attack.

   Delete the contour plot when you are finished exploring, but keep the pseudocolor plot.

### 3.5.4 Visualizing the velocity vector field

This section of the tutorial outlines using glyphs and streamlines to visualize the velocity vector field from the simulation.

<table>
<thead>
<tr>
<th>Plotting</th>
<th>the</th>
<th>velocity</th>
<th>vector</th>
<th>field</th>
</tr>
</thead>
</table>

**VisIt**’s Vector plot renders a vector field at each time step as a collection of arrow glyphs. This allows us to see the direction of the vectors as well as their magnitude. We will create a vector plot to directly view the simulated “v” vector field.

1. Go to **Add->Vector->v**.

2. Open the Vector plot attributes window.

3. Go to the **Vectors** tab.

4. Set **Stride** to “17”.

5. Go the the **Data** tab.

6. In the **Limits** section, enable the **Maximum** checkbox and set the value to “1”.

7. In the **Color** section, change the **Magnitude** to **viridis**

8. Go to the **Glyphs** tab.

9. In the **Scale** section, uncheck **Scale by magnitude** and **Auto scale**.
Fig. 3.61: The pseudocolor plot attributes for the velocity magnitude example.
Fig. 3.62: The pseudocolor plot of the velocity magnitude.
Fig. 3.63: Example contour plot settings for Potential Flow velocity magnitude.
Fig. 3.64: A contour plot of the velocity magnitude.
Fig. 3.65: *Vectors* tab settings for example vector plot of velocity
Fig. 3.66: *Data* tab settings for example vector plot of velocity
Fig. 3.67: *Glyphs* tab settings for example vector plot of velocity
10. Click Apply and Dismiss.
11. Click Draw.
12. Zoom in near the airfoil.
13. Drag the Time animation controls above the plot list on the main GUI window.

You will see glyphs of velocity solutions for the different angles of attack.

Delete the vector plot when you are finished exploring, but keep the pseudocolor plot.

Examining features of the flow field with streamlines

To explore the flow field further we will seed and advect a set of streamlines on the left side of the mesh. Streamlines show the path massless tracer particles would take if advected by a static vector field. To construct Streamlines, the first step is selecting a set of spatial locations that can serve as the initial seed points.

The flow moves left to right, we will use a vertical line of seed points on the left side of the mesh.

Plotting streamlines of velocity

1. Go to Add->Pseudocolor->operators->IntegralCurve->v.
2. Open the IntegralCurve operator attributes window.
Fig. 3.69: Creating a streamline plot with the IntegralCurve operator.
3. Go to the Source section on the Integration tab.
4. Set the Source type to Line.
5. Set the Start to “-2 -2 0”, excluding any commas in the input text box.
6. Set the Stop to “-2 2 0”.
7. Set Samples along line to “10”.
8. Click Apply and Dismiss.
9. Click Draw on the Main GUI
10. In IntegralCurve operator attributes, click Apply again (for good measure) and then Dismiss.
11. Open the Pseudocolor plot attributes window.
12. Go to the Data tab.
13. In the Color section set the Color table to viridis.
14. Go to the Line section on the Geometry tab.
15. Set Line type to Ribbons.
16. Set Tail to Sphere.
17. Set the tail Radius to “0.025”.
18. Click Apply and Dismiss.
19. Click Draw.
20. Use the time slider controls to view a few different angles of attack solutions.
Fig. 3.70: The IntegralCurve operator attributes.
Fig. 3.71: The Pseudocolor attributes for the streamline data.
Fig. 3.72: The Pseudocolor attributes for the streamline geometry.
Fig. 3.73: The streamlines of velocity at 0 degree angle of attack.
Fig. 3.74: The streamlines of velocity at 20.5 degree angle of attack.
4.1 Creating a Pull Request

4.1.1 Overview

Pull Requests (PR (Pull Request)s) allow developers to review work before merging it into the develop branch. PRs are extremely useful for preventing bugs, enforcing coding practices, and ensuring changes are consistent with VisIt’s overall architecture. Because PR reviews can take time, we have adopted policies to help tailor the review effort and balance the load among developers. We hope these policies will help ensure PR reviews are completed in a timely manner. The benefits of reviews outweigh the added time.

4.1.2 Forking the repo

Developers who do not have write access to the primary VisIt repo may make contributions by forking the repo and submitting pull requests. GitHub provides excellent informational articles about forking a repo and creating pull requests from a fork.

4.1.3 Working with the Template

PR submissions are populated with a template to help guide the content. Developers do not have to use this template. Keep in mind, however, that reviewers need structured context in order to accurately and quickly review a PR. So, it is best to use the template or something very similar to it. The text sections in the template are designed to be replaced by information relevant to the work involved. For example, replace a line that says Please include a summary of the change with an actual summary of the change.

In general, if part of the template is not relevant, please delete it before submitting the PR. For example, delete any items in the checklist that are not relevant.

If additional structured sections in the PR submission are needed, please use GitHub markdown styling.

In the sections below, we describe each of the sections of the PR template in more detail.
Description

GitHub supports a number of idioms and keywords in PR submissions to help automatically link related items. Please use them.

For example, when typing a hashtag (#) followed by a number or text, a search menu will appear providing potential matches based on issue or PR numbers or headlines. Sometimes no matches will be produced even if the number being entered is correct, but the link will still occur when the PR is submitted. By placing the keyword “Resolves” in front of a link to an issue, the issue will automatically close when the PR is merged.

If a PR is unrelated to a ticket, please delete the “Resolves #…” line for clarity.

Type of Change

Bug fixes, features, and documentation improvements are among the most common types of PRs. You may select from the menu by replacing the space between the square brackets ([ ] ) with an uppercase X, so that it looks exactly like [X]. You can also make this selection after submitting the PR by checking the box that appears on the submitted PR page.

If “Other” is checked, please describe the type of change in the space below.

Testing

Replace the content of this section with a description of how the change was tested.

The Checklist

The Checklist serves as a list of suggested tasks to be performed before submitting the PR. Those that have been completed should be checked off. Any items that do not relate to the PR should be deleted. For example, if the PR is not for a bugfix or feature, adding a test may not be required and this checklist item should be deleted.

4.1.4 Reviewers

GitHub will not allow non-owners to merge PRs into develop without a reviewer’s approval. Non-owners will need at least one reviewer. Owners may merge a PR into develop without review. But, that does not necessarily mean they should. Follow the guidelines below to determine the need for and number of reviewers. Note, these guidelines serve as a “lower bound”; you may always add more reviewers to your PR if you feel that is necessary.

No Reviewers (owners only)

If your changes are localized, you have satisfied all the testing requirements and you are confident in the correctness of your changes (where correctness is measured by both the correctness of your code for accomplishing the desired task and the correctness of how you implemented the code according to VisIt’s standard practices) then you may merge the PR without a reviewer after the CI tests pass.

One reviewer

If the changes have a broader impact or involve an unfamiliar area of VisIt or existing behavior is being changed, then a reviewer should be added.

Non-owners must always have at least one reviewer even if you satisfy all other guidelines for the No Reviewers case.
If your changes substantially modify existing behavior or you are updating significant amounts of the code or you are designing new architectures or interfaces, then you should have at least two reviewers.

Choosing Reviewers

GitHub automatically suggests reviewers based on the blame data for the files you have modified. You should choose the GitHub suggested reviewer unless you have a specific need for a specific reviewer.

4.1.5 Iteration Process

Review processes are iterative by nature, and PR reviews are no exception. A typical review process looks like this:

1. The developer submits a PR and selects a reviewer.
2. The reviewer reviews the PR and writes comments, suggestions, and tasks.
3. The developer gets clarification for anything that is unclear and updates the PR according to the suggestions.
4. Repeat steps 2 and 3 until the reviewer is satisfied with the PR.
5. The reviewer approves the PR.

The actual amount of time it takes to perform a review or update the PR is relatively small compared to the amount of time the PR waits for the next step in the iteration. The wait time can be exacerbated in two ways: (1) The reviewer or developer is unaware that the PR is ready for the next step in the iteration process, and (2) the reviewer or developer is too busy with other work. To help alleviate the situation, we recommend the following guidelines for the developer (guidelines for the reviewer can be found here).

- Make sure the code is clear and well commented and that the PR is descriptive. This helps the reviewers quickly familiarize themselves with the context of the changes. If the code is unclear, the reviewers may spend a lot of time trying to grasp the purpose and effects of the PR.
- Immediately answer any questions the reviewers ask about the PR. Enabling notifications will help speed this along.
- When the reviewers have finished reviewing (step 2), quickly update the PR according to the requested changes. Use the @username idiom to notify the reviewers for any clarification.
- When you have finished updating your PR (step 3), write a comment on the PR using @username to let the reviewers know that the PR is ready to be looked at again.

4.2 Reviewing a Pull Request

4.2.1 Overview

Pull Requests (PRs) allow developers to review work before merging it into the develop branch. PRs are extremely useful for preventing bugs, enforcing coding practices, and ensuring changes are consistent with VisIt’s overall architecture. Because PR reviews can take time, we have adopted policies to help tailor the review effort and balance the load among developers. We hope these policies will help ensure PR reviews are completed in a timely manner. The benefits of reviews outweigh the added time.
4.2.2 Checklist

In the course of reviewing a PR, the reviewer should use the following as a checklist. The reviewer should verify that any deleted items are rightfully so.

- The developer followed Visit’s style guidelines
- The developer commented the code, particularly in hard-to-understand areas
- The developer updated the release notes
- The developer made corresponding changes to the documentation
- The developer added debugging support
- The developer added tests that prove the fix is effective or that the feature works
- New and existing unit tests pass
- If necessary, the developer added any new baselines to the repository

4.2.3 Comments and Tasks

GitHub provides two ways to add comments to the PR.

**Generic Comments**

The first type of comment is a generic PR comment for communicating about general things related to the changes or the PR process. This comment box is found at the bottom of the “Conversation” tab, which is the main tab on the PR page. The reviewer should use this when pinging the developer to update changes (see Iteration Process below).

**Code Related Comments**

The “Files changed” tab in the PR will show a diff of all the changes. Hover the mouse over the white space to the right of the line number and a blue plus sign will appear. Click this and a comment box will pop up. Type any comments and click either “Add single comment” or “Start a review” (see Review Changes for more information).

This type of comment can be used to ask specific questions or suggest specific changes to the PR.

4.2.4 Review

In addition to comments, the reviewer should also explicitly mark the state of the PR. There are two ways to do this.

Upon writing a code related comment, select the “Start a review” button. This will initiate a review. Click “Add review comment” for each new comment. When you are done, navigate to the top-right of the page and click “Finish your review”.

Alternately, the reviewer can first write all the comments and then submit a review. Use the “Add single comment” button for each code related comment. Then, once you have finished commenting, navigate to the top-right of the page and click “Finish your review”.

Upon clicking the green “Finish your review”, GitHub will present the ability to add additional generic comments and to update the state of the PR. If you left comments via the “Add single comment” button, then you must add an additional comment here to be able to submit a review. These are the three options for updating the PR:

1. Comment - Submit general feedback without explicit approval. This is ambiguous and should not be used because the developer does not always know if the reviewer think changes should be made. It does not update the state of the PR.
2. Approve - Submit feedback and approve merging these changes. Use this when the PR is ready to be merged into develop.

3. Request changes - Submit feedback that must be addressed before merging. Use this when the developer should make additional changes to the PR.

### 4.2.5 Iteration Process

Review processes are iterative by nature, and PR reviews are no exception. A typical review process looks like this:

1. The developer submits a pull request and selects a reviewer.
2. The reviewer writes comments and submit a “Request change” review or an “Approve” review.
3. The developer updates the PR according to the suggestions.
4. Repeat steps 2 and 3 until the PR is ready.
5. The reviewer approves the PR.

The actual amount of time it takes to perform a review or update the PR is relatively small compared to the amount of time the PR waits for the next step in the iteration. The wait time can be exacerbated in two ways: (1) The reviewer or developer is unaware that the PR is ready for the next step in the iteration process, and (2) the reviewer or developer is too busy with other work. To help alleviate the situation, we recommend the following guidelines for the reviewer (guidelines for the developer can be found here):

- Immediately address the PR. Enabling notifications will help speed this along.
- If anything in the PR is unclear, ask specific questions using generic or code related comments. Make use of the `@username` idiom to directly ping the developer.
- Clearly mark the review as “Approved” or “Request changes”.
- Notify the developer with the `@username` idiom that the PR is ready for updates.
- When the developer has updated the PR, make it a top priority to review it again.
- When the PR is ready to be merged into develop, approve the PR and squash-merge the PR into develop with a succinct description of the changes.

If you are chosen as a reviewer and you know that you will not be able to review the PR in a timely manner, please let the developer know and provide suggestions for who to choose instead. Once you start a PR review, you should make it a priority and stick with it until the end.

### 4.3 Release Candidate (RC) Development

#### 4.3.1 Overview

VisIt normally has two active branches for doing development. The first is develop and the second is the current release candidate. Work performed on the develop branch will go into the next major release, such as 3.1. Work performed on the current release candidate will go into the next minor release, such as 3.0.2. When doing work on the release candidate the normal sequence of operations is as follows:

- A branch is created off the current release candidate.
- Changes are made on the branch.
- A pull request is generated to merge the changes to the current release candidate.
- The changes are then merged into the release candidate.
A branch is created off of develop.
- The changes from the branch off the release candidate are applied to the branch.
- A pull request is generated to merge the changes to develop.
- The changes are then merged into develop.

In some instances the changes made to the release candidate are not applied to develop, in many instances the exact same changes can be applied to both the release candidate and develop, and in some instances the changes applied to the two branches are slightly or significantly different.

The rest of the section will go through the steps of the most common case of making the exact same changes to both branches using an example of updating the 3.0.2 release notes on the 3.0RC and develop.

### 4.3.2 Creating the RC branch

First you checkout the 3.0RC and then create your branch.

```
git checkout 3.0RC
```
```
git checkout -b task/brugger1/2019_09_05_update_release_notes
```

### 4.3.3 Making the changes

At this point you would modify your branch as you normally do, modifying, adding or deleting files, and then committing the changes to the branch.

### 4.3.4 Creating the pull request on the release candidate

Once you have committed all your changes to the branch you are ready to create the pull request. You will start out by pushing your changes to GitHub as normal.

```
git push --set-upstream origin task/brugger1/2019_09_05_update_release_notes
```

Now you go over to GitHub and create your pull request. When creating your pull request, make sure that you are merging it into the release candidate.

Now you go through the normal pull request process. Once you have merged your changes into the release candidate you can delete the branch at GitHub and locally.

```
git remote prune origin
```
```
git remote update
```
```
git checkout 3.0RC
```
```
git pull
```
```
git branch -D task/brugger1/2019_09_05_update_release_notes
```

### 4.3.5 Apply the same changes to develop

You will apply your changes from the 3.0RC to develop by creating a patch of your changes to the 3.0RC and applying them to a branch created off of develop. The easiest way to create the patch is immediately after you have merged your changes into the release candidate before anyone else makes any changes. In this case you can get the last set of changes from the head. If someone else has made changes in the mean time you will need to use the SHA of your merge to the release candidate. When we create the branch to make the changes on develop, you can use the
Fig. 4.1: Merging into the release candidate.
same name as you used on the release candidate branch and add _develop. Normally, you can omit the first two steps below since you presumably just did that a moment ago.

```bash
git checkout 3.0RC
git pull
rm -f patch.txt
git format-patch -1 HEAD --stdout > patch.txt
git checkout develop
git pull
git checkout -b task/brugger1/2019_09_05_update_release_notes_develop
git am -3 < patch.txt
```

In the case where you need to use the SHA to create the patch, you can get it from the code tab at GitHub for the release candidate branch.

![GitHub screenshot](image.png)

Fig. 4.2: Getting the SHA for the merge into the release candidate.

The command to create the patch would then look like:

```bash
git format-patch -1 69b0561 --stdout > patch.txt
```

Sometimes conflicts occur when applying the patch. This may happen with frequently updated files such as the release notes. If that happens you will get a message similar to the one below indicating which files had conflicts.
In our case it was the release notes. The file will be modified with the conflicts highlighted in the normal \texttt{>>>>>>>}, \texttt{========}, and \texttt{<<<<<<<<<} notation. You can go in and edit the files and then do a \texttt{git add} for each file that was in conflict. After that point you can do a \texttt{git am --continue}.  

```bash
vi src/resources/help/en_US/relnotes3.0.2.html
git add src/resources/help/en_US/relnotes3.0.2.html
git am --continue
```

Now you changes will have been committed to the branch with the appropriate commit message. You are now ready to push the change to GitHub and create a new pull request.

### 4.3.6 Creating the pull request for develop

You first push your changes to GitHub as normal.

```bash
git push --set-upstream origin task/brugger1/2019_09_05_update_release_notes_develop
```

Now you go over to GitHub and create your pull request. When creating your pull request, make sure that you are merging it into develop.

In the description you can simply say that you are merging from the release candidate into develop rather than providing all the normal pull request information. If you are resolving an issue, you will want to mention that, since the automatic closing of issues only happens when you merge into develop.

Now you go through the normal pull request process. Once you have merged your changes into develop you can delete the branch at GitHub and locally.

```bash
git remote prune origin
git remote update
git checkout 3.0RC
git pull
git branch -D task/brugger1/2019_09_05_update_release_notes_develop
```

That’s it. You have now made the exact same change to both the 3.0RC and develop.

### 4.4 Regression Testing

#### 4.4.1 Overview

VisIt has a large and continually growing test suite. VisIt’s test suite involves a combination python scripts in src/test, raw data and data generation sources in src/testdata and of course the VisIt sources themselves. Regression tests are
Fig. 4.3: Merging into develop.
Fig. 4.4: The pull request with the abbreviated description.
run on a nightly basis. Testing exercises VisIt’s viewer, mdserver, engine and cli but not the GUI.

4.4.2 Running regression tests

Where nightly regression tests are run

The regression suite is run on LLNL’s Pascal Cluster. Pascal runs the TOSS3 operating system, which is a flavor of Linux. If you are going to run the regression suite yourself you should run on a similar system or there will be differences due to numeric precision issues.

The regression suite is run on Pascal using a cron job that checks out VisIt source code, builds it, and then runs the tests.

How to run the regression tests manually

The regression suite relies on having a working VisIt build and test data available on your local computer. Our test data and baselines are stored using git lfs, so you need to setup git lfs and pull to have all the necessary files.

The test suite is written in python and to source is in src/test. When you configure VisIt, a bash script is generated in the build directory that you can use to run the test suite out of source with all the proper data and baseline directory arguments.

```
cd visit-build/test/
./run_visit_test_suite.sh
```

Here is an example of the contents of the generated run_visit_test_suite.sh script

```
/Users/harrison37/Work/github/visit-dav/visit/build-mb-develop-darwin-10.13-x86_64/
→thirdparty_shared/third_party/python/2.7.14/darwin-x86_64/bin/python2.7
/Users/harrison37/Work/github/visit-dav/visit/src/test/visit_test_suite.py \
   -d /Users/harrison37/Work/github/visit-dav/visit/build-mb-develop-darwin-10.13-x86_\n   →64/build-debug/testdata/ \ 
   -b /Users/harrison37/Work/github/visit-dav/visit/src/test/../../test/baseline/ \ 
   -o output \ 
   -e /Users/harrison37/Work/github/visit-dav/visit/build-mb-develop-darwin-10.13-x86_\n   →64/build-debug/bin/visit "$@"
```

Once the test suite has run, the results can be found in the output/html directory. Open output/html/index.html in a web browser to view the test suite results.

Accessing regression test results

The nightly test suite results are posted to: http://portal.nersc.gov/project/visit/.

In the event of failure on the nightly run

If any tests fail, “all” developers who updated the code from the last time all tests successfully passed will receive an email indicating what failed. In addition, failed results should be available on the web.

If the results fail to post, the visit group on NERSC’s systems may be over quota. If you have a NERSC account you can check usage by sshing to NERSC and running the following command:

```
prjquota visit
```
How regression testing works

The workhorse script that manages the testing is `visit_test_suite.py` in `src/test`. Tests can be run in a variety of ways called `modes`. For example, VisIt’s nightly testing is run in `serial`, `parallel` and `scalable,parallel` modes. Each of these modes represents a fundamental and relatively global change in the way VisIt is doing business under the covers during its testing. For example, the difference between `parallel` and `scalable,parallel` modes is whether the scalable renderer is being used to render images. In the `parallel` mode, rendering is done in the viewer. In `scalable,parallel` mode, it is done, in parallel, on the engine and images from each processor are composited. Typically, the entire test suite is run in each mode specified by the regression test policy.

There are a number of command-line options to the test suite. `./run_visit_test_suite.sh -help` will give you details about these options. Until we are able to get re-baselined on the systems available outside of LLNL firewalls, options enabling some filtering of image differences will be very useful. Use of these options on platforms other than the currently adopted testing platform (pascal.llnl.gov) will facilitate filtering big differences (and probably real bugs that have been introduced) from differences due to platform where tests are run. See the section on filtering image differences.

There are a number of different categories of tests. The test categories are the names of all the directories under `src/test/tests`. The .py files in this directory tree are all the actual test driver files that drive VisIt’s CLI and generate images and text to compare with baselines. In addition, the `src/test/visit_test_main.py` file defines a number of helper Python functions that facilitate testing including two key functions; Test() for testing image outputs and TestText() for testing text outputs. Of course, all the .py files in `src/test/tests` subtree are excellent examples of test scripts.

When the test suite finishes, it will have created a web-browsable HTML tree in the html directory. The actual image and text raw results will be in the current directory and difference images will be in the diff directory. The difference images are essentially binary bitmaps of the pixels that are different and not the actual pixel differences themselves. This is to facilitate identifying the location and cause of the differences.

Adding a test involves a) adding a .py file to the appropriate subdirectory in `src/test/tests`, b) adding the expected baselines to `test/baselines` and, depending on the test, c) adding any necessary input data files to `src/testdata`. The test suite will find your added .py files the next time it runs. So, you don’t have to do anything special other than adding the .py file.

One subtlety about the current test modality is what we call `mode specific baselines`. In theory, it should not matter what mode VisIt is run in to produce an image. The image should be identical across modes. In practice there is a long list of things that can contribute to a handful of pixel differences in the same test images run in different modes. This has lead to mode specific baselines. In the baseline directory, there are subdirectories with names corresponding to modes we currently run. When it becomes necessary to add a mode specific baseline, the baseline file should be added to the appropriate baseline subdirectory.

In some cases, we skip a test in one mode but not in others. Or, we temporarily disable a test by skipping it until a given problem in the code is resolved. This is handled by the `--skiplist` argument to the test suite. We maintained list of the tests we currently skip and update it as necessary. The default skip list file is `src/test/skip.json`.

Filtering Image Differences

There are many ways of both compiling and running VisIt to produce image and textual outputs. In many cases, we expect the image or textual outputs to be about the same (though not always bit-wise exact matches) even if the manner in which they are generated varies dramatically. For example, we expect VisIt running on two different implementations of the GL library to produce by and large the same images. Or, we expect VisIt running in serial or parallel to produce the same images. Or we expect VisIt running on Ubuntu Linux to produce the same images as it would running on Mac OSX. We expect and therefore wish to ignore ‘minor variations’. But, we want to be alerted to ‘major variations’. So when any developer runs a test, we require some means of filtering out image differences we expect from those we are not expecting.

On the other hand, as we make changes to VisIt source code, we may either expect or not expect image outputs for specific testing scenarios to change in either minor or dramatic ways. For example, if we fix a bug leading to a
serious image artifact that just happened to be overlooked when the original baseline image was committed, we could
improve the image dramatically implying a large image difference and still expect such a difference. For example,
maybe the Mesh plot had a bug where it doesn’t obey the Mesh line color setting. If we fix that bug, the mesh line
color will likely change dramatically. But, the resultant image is expected to change too. Therefore, have a set of
baselines from which we compute exact differences is also important in tracking impact of code changes on VisIt
behavior.

These two goals, running VisIt tests to confirm correct behavior in a wide variety of conditions where we expect
minor but not major variations in outputs and running VisIt tests to confirm behavior as code is changed where we
may or may not expect minor or major variations are somewhat complimentary.

It may make sense for developers to generate (though not ever commit) a complete and valid set of baselines on their
target development platform and then use those (uncommitted) baselines to enable them to run tests and track code
changes using an exact match methodology.

Metrics:

- **Total pixels** - count of all pixels in the test image
- **Plot pixels** - count of all pixels touched by plot(s) in the test image
- **Coverage** - percent of all pixels that are plot pixels (plot pixels / total pixels). Test images in which plots
occupy a small portion of the total image are fraught with peril and should be avoided to begin with. Images
with poor coverage are more likely to produce false positives (e.g. passes that should have failed) or to exhibit
somewhat random differences as test scenario is varied.
- **DMAX / DMAXP** - maximum raw numerical / human perceptual difference in any color (R, G or B) channel or
intensity (average of R, G, B colors). A good first try in filtering image differences is a dmax setting of 1. That
will admit variations of 1 in any R, G or B channel or in intensity. However, for line-based plots like the mesh
plot, due to differences in the way lines of the plot get scanned into pixels, this metric can fail miserably.
- **DMED / DMEDP** - median value of raw numerical / human perceptual differences over all color channels and
intensity

When running the test suite on platforms other than the currently adopted baseline platform or when running tests in
modes other than the standard modes, a couple of options will be very useful: **pixdiff** and **avgdiff**. The pixdiff option
allows one to specify a tolerance on the percentage of non-background pixels that are different. The avgdiff option
allows one to specify a second tolerance for the case when the pixdiff tolerance is exceeded. The avgdiff option
specifies the maximum average (intensity) difference difference allowed averaged over all pixels that are different.

**Tips on writing** regression **tests**

- Except in cases where annotations are being specifically tested, remember to call TurnOffAllAnnotations() as
one of the first actions in your test script. Otherwise, you can wind up producing images containing machine-
specific annotations which will produce differences on other platforms.

- When writing tests involving text differences and file pathnames, be sure that all pathnames in the text strings
passed to TestText() are absolute. Internally, VisIt testing system will filter these out and replace the machine-
specific part of the path with VISIT_TOP_DIR to facilitate comparison with baseline text. In fact, the .txt files
that get generated in the current dir will have been filtered and all pathnames modified to have VISIT_TOP_DIR
in them.

- Here is a table of python tests scripts which serve as examples of some interesting and lesser known VisIt/Python
scripting practices:
### Script

<table>
<thead>
<tr>
<th>Script</th>
<th>What it demonstrates</th>
</tr>
</thead>
<tbody>
<tr>
<td>tests/faulttolerant/savewindow.py</td>
<td>• uses python exceptions</td>
</tr>
<tr>
<td>tests/databases/ltaps.py</td>
<td>• uses OpenDatabase with specific plugin</td>
</tr>
<tr>
<td></td>
<td>• uses SIFL restriction via names of sets</td>
</tr>
<tr>
<td>tests/databases/silo.py</td>
<td>• uses OpenDatabase with virtual database and a specific timestep</td>
</tr>
<tr>
<td>tests/rendering/scalable.py</td>
<td>• uses OpenComputeEngine to launch a parallel engine</td>
</tr>
<tr>
<td>tests/rendering/offscreensave.py</td>
<td>• uses Test() with alternate save window options</td>
</tr>
<tr>
<td>tests/databases/xform_precision.py</td>
<td>• uses test-specific environment variable settings</td>
</tr>
</tbody>
</table>

### Rebaselining

A python script, `rebase.py`, at `src/tests` dir can be used to rebaseline large numbers of results. In particular, this script enables a developer to rebase test results without requiring access to the test platform where testing is performed. This is because the PNG files uploaded (e.g. posted) to VisIt’s test results dashboard are suitable for using as baseline results. To use this script, run `./rebase.py -help`. Once you’ve completed using rebase.py to update image baselines, don’t forget to commit your changes back to the repository.

### 4.4.3 Using VisIt Test Suite for Sim Code Testing

VisIt’s testing infrastructure can also be used from a VisIt install by simulation codes how want to write their own Visit-based tests. For more details about this, see: Leveraging VisIt in Sim Code Regression Testing

### 4.5 XML

#### 4.5.1 Overview

VisIt developers use several xml-based code generation tools to implement VisIt’s features. The source core for these tools is kept in `src/tools/dev/xml/` and `src/tools/dev/xmledit/`.

#### 4.5.2 CMake

We rely on xml code generation to keep our State object, Attribute, and Plugin APIs up-to-date. To automate the process we provide CMake targets that call our xml code generation tools for each object or plugin registered. Individual code gen targets are all wired into top level targets that allow you to apply the code gen tools to categories of code gen tasks. These targets replace older tools such as regen-ajp and various regenerateatts.py scripts.
CMake Target | Target Action
--- | ---
gen_cpp_all | Run xml2atts on all identified objects
gen_python_all | Run xml2python on all identified objects
gen_java_all | Run xml2java on all identified objects
gen_info_all | Run xml2info on all plugins
gen_cmake_all | Run xml2cmake on all plugins
gen_plugin_all | Run all applicable xml tools for all plugins

CMake Code Gen Functions

These are the helper functions we use to create targets that call xml tools in our CMake build system.

XML Tools Helper functions in src/CMake/VisItMacros.cmake:

<table>
<thead>
<tr>
<th>CMake Function</th>
<th>Target Action</th>
</tr>
</thead>
<tbody>
<tr>
<td>ADD_CPP_GEN_TARGET</td>
<td>Calls xml2atts</td>
</tr>
<tr>
<td>ADD_PYTHON_GEN_TARGET</td>
<td>Calls xml2python</td>
</tr>
<tr>
<td>ADD_JAVA_GEN_TARGET</td>
<td>Calls xml2java</td>
</tr>
<tr>
<td>ADD_INFO_GEN_TARGET</td>
<td>Calls xml2info</td>
</tr>
<tr>
<td>ADD_CMAKE_GEN_TARGET</td>
<td>Calls xml2cmake</td>
</tr>
</tbody>
</table>

The xml2plugin and xml2avt tools are only called when you first create a new plugin or object, they are not exposed here.

Plugin Tools Helper functions in src/CMake/PluginMacros.cmake:

- ADD_PLUGIN_CODE_GEN_TARGETS - wires up:
  - ADD_CPP_GEN_TARGET
  - ADD_PYTHON_GEN_TARGET
  - ADD_JAVA_GEN_TARGET
  - ADD_INFO_GEN_TARGET
  - ADD_CMAKE_GEN_TARGET
  - ADD_DATABASE_CODE_GEN_TARGETS - wires up:
    - ADD_INFO_GEN_TARGET
    - ADD_CMAKE_GEN_TARGET
  - ADD_OPERATOR_CODE_GEN_TARGETS - alias for ADD_PLUGIN_CODE_GEN_TARGETS
  - ADD_PLOT_CODE_GEN_TARGETS - alias for ADD_PLUGIN_CODE_GEN_TARGETS

CMake Options

VISIT_CMAKE_VERBOSE_GEN_TARGET_MESSAGES (default: OFF) When enabled display messages about code gen targets during CMake configure time.
4.6 Preparing for a Release

4.6.1 Overview

VisIt supports three types of releases, major, minor and patch, where the version number consists of “major.minor.patch”. Patch releases are the most common type of release and typically occur three to four times a year. Minor releases are the next most common type of release and may occur once or twice a year. Major releases happen very infrequently, with as much as 10 years passing between major releases.

4.6.2 Preparing for a Patch Release

Preparing for a minor release is pretty straightforward and consists of updating a few files. These consist of

VERSION
INSTALL_NOTES
gui/Splashscreen.C

4.6.3 Preparing for a Minor Release

Preparing for a minor release consists of performing all the steps involved in preparing for a patch release, along with some additional ones, such as creating the release candidate branch and updating the splash screen.

Creating the Release Candidate Branch

Creating a release candidate branch is just like creating a normal branch. Here are the steps used to create the 3.1RC.

git checkout develop
git pull
git checkout -b 3.1RC
git push --set-upstream origin 3.1RC

Updating the Splashscreen

The splashscreen is the first thing the user sees when running VisIt so the version number included in the splashscreen image should be up to date. Updating the splashscreen usually means just updating the version number in the current splashscreen images but in the event of a major or minor release (when the first or second digit in the version changes), the splashscreen images should be redesigned to showcase new features.

There are two image files associated with the splashscreen, both of which are XCF. XCF files are the native image format of the GIMP image-editing program. One is for the splashscreen and the second is for the icon on MacOS X. They are both used as the first step in the process to create the splashscreen and icon.

The rest of this section will be focused on updating the version number. Changing the images would be the same in terms of the mechanics involved except that it would involve more editing of the image files.

Changing the version on the splashscreen

Follow these steps to update the version on the splashscreen.

1. Go to the src/common/icons directory.
2. The splashscreen image’s XCF files are named VisIt3.0.xcf, VisIt3.1.xcf, etc.
3. Copy the file from the last version to the new name for the current version.
4. Open the file in GIMP.

![GIMP interface showing VisIt splashscreen](image)

Fig. 4.5: The splashscreen in GIMP.

You’ll see that the file has several layers to it. There are four layers for each of the four splash screen images that get randomly chosen from when starting VisIt or are cycled through when you select About in the Help menu.

4. Select the text layer containing the version number and change it.
5. Save the file.

Now you are ready to create the png images that are actually read in by Qt. When you open the XCF file all the layers corresponding to the four different splashscreen images will be enabled. When you save the first image you will have them all shown. To save the second image you will hide the layer corresponding to the first splashscreen image. You will successively hide one additional layer until you have saved all four of the png images.

6. Go to File->Export As and change Name to VisIt1.png.
7. Click on Export.
8. Click on Export on the window that pops up to allow you to set the save options.
9. Hide Background1.
10. Repeat steps 6 - 9, saving images VisIt2.png, VisIt3.png and VisIt4.png.

The images saved by GIMP result in warning messages when read by Qt. To modify the images so that the warning message disappears do the following.

```
convert VisIt1.png VisIt1a.png
convert VisIt2.png VisIt2a.png
convert VisIt3.png VisIt3a.png
```
Fig. 4.6: Hiding the Background1 layer in GIMP.
11. Copy the files to src/resources/images.

Changing the version on the MacOS X icon

When VisIt starts on MacOS X systems, it adds an icon into the Mac application dock. The icon that we use is based on the splashscreen but is stored in MacOS X icon format.

Follow these steps to update the version on the MacOS X icon.

1. Go to the src/common/icons directory.
2. Create the directory VisItIcon.iconset.
3. Open the file VisIt3.x-square.xcf in GIMP.
4. Select the text layer containing the version number and change it.
5. Go to Image->Scale Image.
6. Change the Image Size Width and Height to 1024.
7. Click on Scale.
8. Go to File->Export As and change Name to VisItIcon.iconset/icon_512x512@2x.png.
9. Click on Export.
10. Click on Export on the window that pops up to allow you to set the save options.

Now you need to create several sizes of the file. You will use ImageMagick for this.

Now you will use iconutil to create the icns file. Note that iconutil is only available on the Mac.

4.6.4 Preparing for a Major Release

Preparing for a major release is the same as preparing for a minor release with the addition of putting VisIt through the Information Management software release process.
4.7 Creating a Release

4.7.1 Overview

When we put out a new release we should tag the repository and create a release. We will describe creating a release by way of example using the steps used to create the 3.0.1 release.

4.7.2 Tagging the release

To create a release you will first create a tag using git commands. You should get the short SHA for the release that can be found on the splash screen of any of the binaries built from the source tar file. The Linux distributions are all built with the source tar file. The Windows distribution is typically not. To bring up the splash screen go to Help->About.

Now you can issue the git commands to create the tag and push it to GitHub.

```
git checkout 3.0RC
git checkout 2f38385
git tag v3.0.1
git push origin v3.0.1
```

If you go to GitHub and go to the Releases tab you will see the newly created tag. Now you are ready to create the release. Click on Draft a new release to bring up the form to create a new release.

Now you can enter information about the release. Set the Tag version to v3.0.1, the Release title to v3.0.1 and copy and paste the description from the 3.0.0 release into the description, changing the link to the release notes appropriately. At this point you can go to the bottom of the window and click on Publish release.
Fig. 4.8: Creating a new release.
Fig. 4.9: Entering information about the release.
Your newly created release will now appear.

Fig. 4.10: The newly created release.

### 4.7.3 Deleting a release

If you mess up the tag or the release you can delete the tag using git commands.

```bash
git tag -d v3.0.1
git push origin :refs/tags/v3.0.1
```

You can then remove the release at GitHub. The release will change to a draft release because the tag no longer exists. Go ahead and click on the release to bring up the draft release.

Click on *Delete* to delete the release.
Fig. 4.11: Selecting the draft release corresponding to the deleted tag.
Fig. 4.12: Deleting the draft release corresponding to the deleted tag.
4.8 Finding Memory Leaks

4.8.1 Overview

We support several mechanisms to find memory leaks. The two best mechanisms are using Valgrind and vtkDebugLeaks. Valgrind is used to detect memory leaks in a specific component. You run VisIt with the appropriate options and then when VisIt is finished running you will get logs with a report of memory leaks and usage. The log files are quite large. The top of the log file contains stack traces of where the memory was allocated for each chunk of memory allocated. It is sorted from the smallest leaks to the largest leaks. At the bottom of the log is a summary of memory leaks and usage. vtkDebugLeaks is specifically used to find VTK memory leaks. It provides a list of all the VTK objects that are still in use when VisIt terminates to the terminal. There will be one list for the viewer and one for the engine (multiple lists if running in parallel).

4.8.2 Building VisIt for Valgrind and vtkDebugLeaks

The following steps were from building and running VisIt 3.1 on Quartz, a Linux cluster.

**Building** **the** **Third** **party** **Libraries**

In order to use vtkDebugLeaks you will need to enable it when you build VTK. Edit the build_visit script and find the line:

```
vtk_debug_leaks="false"
```

and change it to:

```
vtk_debug_leaks="true"
```

In order for the stack traces from Valgrind to be the most useful, you should build the third party libraries with debug support. In our case we are going to do a minimal build with just the Silo and HDF5 I/O libraries.

```
./build_visit3_1_0 --required --mesagl --llvm --silo --hdf5 --debug --no-visit \
--thirdparty-path /usr/workspace/wsa/visit/visit/thirdparty_shared/3.1.0/toss3_debug \
--makeflags -j16
```

**Building VisIt**

Just like the third party libraries, VisIt needs to be built with debug support in order for Valgrind to produce useful stack traces. Furthermore, VisIt contains conditional code that does additional cleanup at exit to eliminate spurious memory leaks that is enabled with VISIT_DEBUG_LEAKS. The following steps were used to build VisIt as described.

```
cd visit3.1.0
mkdir build
cd build
/usr/workspace/wsa/visit/visit/thirdparty_shared/3.1.0/toss3_debug/cmake/3.9.3/linux-\ ˓
-x86_64_gcc-4.9/bin/cmake -DCMAKE_BUILD_TYPE=Debug -DVISIT_CONFIG_SITE=/usr/ \ ˓
-workspace/wsa/brugger/visit_memory/quartz2498.cmake -DCMAKE_CXX_FLAGS:STRING="- \ ˓
-DDDEBUG_MEMORY_LEAKS" ..\src
make -j 36
```

You are now ready to start looking for memory leaks.
Follow these steps to run Valgrind on the viewer to find memory leaks from a basic use case.

1. Run VisIt with Valgrind on the viewer in nowin mode.

   cd ..
   mkdir run1
   cd run1
   ../visit3.1.0/build/bin/visit -valgrind viewer -nowin

2. Open `wave.visit`.

3. Create a Pseudocolor plot of `pressure`.

4. Save the window.

5. Delete the plot.

6. Close the database.

7. Exit

There are several things to note. We ran in nowin mode to eliminate leaks from Qt, which are difficult to address. Since we ran in nowin mode we had to save an image in order to do some rendering. After we saved the image we deleted the plot and closed the database to clean up as much memory as possible.

After VisIt exits, you will get vtkDebugLeaks output sent to the terminal as well as log files created by Valgrind. Let’s look at the Valgrind output first.

Looking at the Valgrind Output

Valgrind creates several log files with the output. You are interested in the largest one. Here are the files generated from the run described above.

```
ls -l
total 48792
-rw------- 1 brugger brugger 49715025 Dec 4 11:44 vg_viewer_8870.log
-rw------- 1 brugger brugger 449 Dec 4 11:43 vg_viewer_8915.log
-rw------- 1 brugger brugger 449 Dec 4 11:44 vg_viewer_9281.log
-rw------- 1 brugger brugger 32373 Dec 4 11:44 visit0000.png
```

Here is the output from the end of the largest log file.

```
tail -11 vg_viewer_8870.log
==8870== LEAK SUMMARY:
==8870== definitely lost: 0 bytes in 0 blocks
==8870== indirectly lost: 0 bytes in 0 blocks
==8870== possibly lost: 33,097 bytes in 245 blocks
==8870== still reachable: 30,386,572 bytes in 24,057 blocks
==8870== of which reachable via heuristic:
==8870== stdstring : 25,296 bytes in 649 blocks
==8870== suppressed: 0 bytes in 0 blocks
==8870== For counts of detected and suppressed errors, rerun with: -v
==8870== ERROR SUMMARY: 202 errors from 202 contexts (suppressed: 0 from 0)
```

This is actually pretty good. There is still more work to be done to address the possibly lost memory and then there may be issues with the still reachable. This is probably primarily from a lack of cleanup before exiting.
Looking at the vtkDebugLeaks Output

You will get vtkDebugLeaks output from both the viewer and engine since both have VTK code and both were linked against VTK built with vtkDebugLeaks. Here is the engine output, which came out first.

```
vtkDebugLeaks has detected LEAKS!
Class "9vtkBufferIxE" has 1 instance still around.
Class "vtkDataSetAttributes" has 2 instances still around.
Class "vtkGraphInternals" has 1 instance still around.
Class "vtkOutputWindow" has 1 instance still around.
Class "vtkInformation" has 1 instance still around.
Class "vtkInformationIntegerValue" has 4 instances still around.
Class "vtkGraphEdge" has 1 instance still around.
Class "vtkIdTypeArray" has 1 instance still around.
Class "vtkTypeUInt32Array" has 1 instance still around.
Class "vtkFieldData" has 1 instance still around.
Class "vtkMergeTree" has 1 instance still around.
Class "vtkCommand or subclass" has 1 instance still around.
Class "9vtkBufferIxE" has 1 instance still around.
```

As you can see, it had relatively few leaks associated with VTK. These may not even be leaks, they are probably from a lack of cleaning up before exiting. More work needs to be done here.

Here is the viewer output.

```
vtkDebugLeaks has detected LEAKS!
Class "vtkOpenGLRenderTimerLog" has 1 instance still around.
Class "vtkOpenGLTextActor" has 33 instances still around.
Class "vtkOpenGLTextMapper" has 1 instance still around.
Class "vtkTextureObject" has 5 instances still around.
Class "9vtkBufferIxE" has 146 instances still around.
Class "vtkTexturedActor2D" has 1 instance still around.
Class "vtkBackgroundActor" has 1 instance still around.
Class "vtkCell1Data" has 195 instances still around.
Class "vtkVisItTextActor" has 1 instance still around.
Class "vtkInformationIntegerVectorValue" has 17 instances still around.
Class "vtkInformationVector" has 1454 instances still around.
Class "vtkVisItCubeAxesActor" has 1 instance still around.
Class "vtkPerspectiveTransform" has 6 instances still around.
Class "vtkPointData" has 195 instances still around.
Class "vtkProperty2D" has 16 instances still around.
Class "vtkCompositeDataPipeline" has 290 instances still around.
Class "vtkMatrix3x3" has 236 instances still around.
Class "vtkTrivialProducer" has 100 instances still around.
Class "vtkAxesActor2D" has 1 instance still around.
Class "vtkOpenGLIndexBufferObject" has 1006 instances still around.
Class "vtkMatrix4x4" has 1331 instances still around.
Class "vtkPickingManager" has 1 instance still around.
Class "QVTKInteractor" has 1 instance still around.
Class "vtkCoordinate" has 160 instances still around.
Class "vtkSimpleTransform" has 35 instances still around.
Class "vtkFollower" has 88 instances still around.
Class "vtkOutputWindow" has 1 instance still around.
Class "vtkPoints" has 192 instances still around.
Class "vtkInformation" has 3503 instances still around.
Class "vtkActorCollection" has 3 instances still around.
Class "vtkLine" has 5 instances still around.
Class "vtkGenericOpenGLRenderWindow" has 1 instance still around.
```

(continues on next page)
Class "vtkVolumeCollection" has 3 instances still around.
Class "vtkPropCollection" has 5 instances still around.
Class "vtkInformationIntegerPointerValue" has 36 instances still around.
Class "vtkTriad2D" has 1 instance still around.
Class "vtkPolyData" has 159 instances still around.
Class "vtkLookupTable" has 34 instances still around.
Class "vtkPixel" has 5 instances still around.
Class "vtkAppendPolyData" has 1 instance still around.
Class "vtkOpenGLImageMapper" has 1 instance still around.
Class "vtkPropPicker" has 1 instance still around.
Class "vtkActor2D" has 11 instances still around.
Class "vtkOpenGLCamera" has 3 instances still around.
Class "vtkPolyData" has 159 instances still around.
Class "vtkLookupTable" has 34 instances still around.
Class "vtkPixel" has 5 instances still around.
Class "vtkAppendPolyData" has 1 instance still around.
Class "vtkOpenGLImageMapper" has 1 instance still around.
Class "vtkPropPicker" has 1 instance still around.
Class "vtkActor2D" has 11 instances still around.
Class "vtkOpenGLCamera" has 3 instances still around.
Class "vtkOpenGLVertexArrayObject" has 1006 instances still around.
Class "vtkOpenGLActor" has 115 instances still around.
Class "vtkOpenGLPolyDataMapper" has 118 instances still around.
Class "vtkKidList" has 10 instances still around.
Class "vtkWorldPointPicker" has 1 instance still around.
Class "vtkDoubleArray" has 10 instances still around.
Class "vtkMatrixToLinearTransform" has 26 instances still around.
Class "vtkAlgorithmOutput" has 194 instances still around.
Class "vtkCullerCollection" has 3 instances still around.
Class "vtkOpenGLRenderer" has 3 instances still around.
Class "vtkPolyDataAlgorithm" has 89 instances still around.
Class "vtkDepthSortPolyData2" has 1 instance still around.
Class "vtkInformationIntegerValue" has 3756 instances still around.
Class "vtkOpenGLLight" has 10 instances still around.
Class "vtkOpenGLPolyDataMapper2D" has 45 instances still around.
Class "vtkTextProperty" has 93 instances still around.
Class "vtkCell1Array" has 146 instances still around.
Class "vtkRendererCollection" has 1 instance still around.
Class "vtkOpenGLVertexBufferObjectCache" has 1 instance still around.
Class "vtkActor2DCollection" has 3 instances still around.
Class "vtkImageData" has 36 instances still around.
Class "vtkFloatArray" has 222 instances still around.
Class "vtkInformationStringValue" has 108 instances still around.
Class "vtkInformationExecutivePortVectorValue" has 194 instances still around.
Class "vtkOpenGLVertexBufferObject" has 9 instances still around.
Class "vtkIdTypeArray" has 146 instances still around.
Class "vtkTransform" has 541 instances still around.
Class "vtkOutlineSource" has 5 instances still around.
Class "vtkOpenGLVertexBufferObjectGroup" has 163 instances still around.
Class "vtkFieldData" has 195 instances still around.
Class "vtkVisItAxisActor2D" has 9 instances still around.
Class "vtkOpenGLShaderCache" has 1 instance still around.
Class "vtkTDxInteractorStyleCamera" has 3 instances still around.
Class "vtkImageData" has 36 instances still around.
Class "vtkFloatArray" has 222 instances still around.
Class "vtkInformationStringValue" has 108 instances still around.
Class "vtkInformationExecutivePortVectorValue" has 194 instances still around.
Class "vtkOpenGLVertexBufferObject" has 9 instances still around.
Class "vtkIdTypeArray" has 146 instances still around.
Class "vtkTransform" has 541 instances still around.
Class "vtkOutlineSource" has 5 instances still around.
Class "vtkTimerLog" has 166 instances still around.
Class "vtkBufferIfE" has 222 instances still around.
Class "vtkBufferIdE" has 10 instances still around.
Class "vtkCommand or subclass" has 208 instances still around.
Class "vtkBufferIhE" has 41 instances still around.
Class "vtkInformationExecutivePortValue" has 237 instances still around.
Class "vtkFXAAOptions" has 3 instances still around.

As you can see, the viewer has considerably more leaks associated with VTK. Again, these may not be leaks but merely a lack of cleanup before exiting. More work needs to be done here as well.

That’s it. Happy hunting!

4.9 Using Docker

4.9.1 Overview

Docker is a platform for building containers. Containers can run either Windows or Linux operating systems. Docker is available on the Mac, Windows and Linux. The rest of this tutorial will primarily be focused on running Docker on Windows. The content on installing and setting up Docker is Windows specific but the remainder of the content on creating and using containers is operating system independent.

4.9.2 Installing Docker on Windows

Install Docker on your system. It is free to download and install. You will need to be running Windows Professional and you will need administrator priviledges. The following link will get you started.

https://docs.docker.com/docker-for-windows/

You will need to enable experimental features to be able to use the --squash option when building your container. You can enable experimental features with the Settings window. Go to the Daemon tab and check the Experimental features checkbox and press Apply. Note that this will restart the Docker daemon, which will kill container builds or running containers.

If you run into problems running out of disk space, you can increase the amount of disk space allocated to Docker with the Settings window. Go to the Advanced tab and move the Disk image max size to the right to increase the amount of disk space and press Apply. Note that this will restart the Docker daemon, which will kill container builds or running containers.

4.9.3 Creating a Docker Container

First you will want to bring up a Command window and use that to run Docker commands. Next we’ll create a folder to hold all our Docker files. We are assuming that you are at the root of the C: drive.

C:\>cd \Users\brugger1
C:\Users\brugger1>mkdir docker
C: cd docker

Now you need to copy all the relevant files to your docker folder. You must have the following files in your folder.
Fig. 4.13: Enabling experimental features.
Fig. 4.14: Increasing the disk space allocated to Docker.
VisIt User Manual Documentation, Release 3.1

C:\Users\brugger1\docker>dir
Volume in drive C is Windows
Volume Serial Number is A8F6-9F9C

Directory of C:\Users\brugger1\docker
09/13/2019 02:46 PM <DIR> .
09/13/2019 02:46 PM <DIR> ..
09/13/2019 02:39 PM 737,636 build_visit3_0_2
08/28/2019 01:45 PM 1,173 build_visit_docker_cleanup.py
09/12/2019 12:24 PM 1,322 Dockerfile-debian9
09/13/2019 07:27 AM 1,176 Dockerfile-fedora27
09/12/2019 03:49 PM 1,337 Dockerfile-ubuntu16
09/12/2019 12:36 PM 1,321 Dockerfile-ubuntu18
09/12/2019 12:23 PM 216 run_build_visit.sh
09/13/2019 02:39 PM 121,776,180 visit3.0.2.tar.gz
8 File(s) 122,520,361 bytes
2 Dir(s) 814,047,002,624 bytes free

These files can be found in the VisIt repository at GitHub in the following location.
https://github.com/visit-dav/visit/tree/develop/scripts/docker

The Dockerfile determines the type of operating system you will build your container with. The first line in the Dockerfile contains information about the operating system. Here is a link to a reference on Dockerfile.
https://docs.docker.com/engine/reference/builder/

The Dockerfile will need to be specific to the operating system since the way you install packages and do other administrative tasks will vary among different Linux operating systems, although there are only a few unique variants that the rest are built on. You can go to the Docker Hub to find Linux distributions to start with.
https://hub.docker.com/_/centos
https://hub.docker.com/_/debian
https://hub.docker.com/_/fedora
https://hub.docker.com/_/ubuntu

In this example the Dockerfile is set up to use Ubuntu 16. The Dockerfile installs all the packages needed to build VisIt and then uses build_visit to create all the third party libraries as well as the config site file. The build will take several hours. Sometimes I have had it stop sending text to the Command window, so if it looks like it is hung, it may actually be happily progressing along.

C:\Users\brugger1\docker>docker build -f Dockerfile-ubuntu16 -t visitdev:3.0.2-ubuntu16 . --squash

Start up the container and run it interactively.

C:\Users\brugger1\docker>docker run -t -i visitdev:3.0.2-ubuntu16 /bin/bash
visit@bea87fee3276:$

Now the container is ready for you to build VisIt. First, you need to copy the tar file with the source code. To do this, you will need to go to another Command window and use the container id shown in the prompt.

C:\Users\brugger1\docker>docker cp visit3.0.2.tar.gz bea87fee3276:/home/visit

Now go back to the first Command window and create your distribution.
Now let’s test it to make sure we can create an image.

```
visit@bea87fee3276:~/visit3.0.2/build$ cd ../..
visit@bea87fee3276:~$ cp visit3.0.2/src/tools/dev/scripts/visit-install .
visit@bea87fee3276:~$ ./visit-install 3.0.2 linux-x86_64 visit
visit@bea87fee3276:~$ visit@bin/visit -cli -nowin
>>> OpenDatabase("visit/data/curv2d.silo")
>>> AddPlot("Pseudocolor", "d")
>>> DrawPlots()
>>> SaveWindow()
>>> quit()
visit@:~$
```

Now let’s go back to the second Command window and copy the binary distribution back out of the container and the image we created.

```
C:\Users\brugger\docker>docker cp bea87fee3276:/home/visit/visit3_0_2.linux-x86_64.tar.gz .
C:\Users\brugger\docker>docker cp bea87fee3276:/home/visit/visit0000.png .
```

At this point you can exit your container.

```
visit@bea87fee3276:~$ exit
C:\Users\brugger\docker>
```

You should view the image to verify that it was produced correctly. You now have the binary distribution for VisIt 3.0.2 for Ubuntu 16.

4.9.4 Creating a Dockerfile From Scratch

To create a Dockerfile from scratch it is best to do so running interactively as root with the base operating system image. You can start by installing packages that you are certain you will need. At that point you can run build_visit until it fails, determining what missing package caused the failure, installing the missing package and repeating until you have gotten build_visit to complete with the third party libraries you want to build. From that experience you can create your Dockerfile.

4.9.5 Useful Docker Commands

Here are some useful Docker commands.

```
docker image ls
docker container ls --all
```

(continues on next page)
Docker will create a “checkpoint” after each command it executes. Everytime you partially create an image or execute a container it is saving those checkpoints. This can quickly start to consume a lot of disk space, so you should frequently list your images and containers and remove those that you no longer need.

### 4.10 Customer Response

We deliberately use the term **Customer Response** as an amalgamation of the terms **Customer Support** and **Incident Response** to represent the fact that both activities are involved here. **Customer Support** involves responding to inquiries about the use of VisIt. *How do I do X? Y is not working for me, can you help? I am having trouble getting VisIt to work in client/server.* These are examples of typical **Customer Support** inquiries VisIt developers are routinely enlisted to answer.

In some cases, user’s inquiries reveal a deeper, systemic, recently occurring **operational** issue which may involve either an issue with the VisIt software itself (e.g. a botched VisIt installation, an incompatibility in client/server protocol, a missing database reader plugin, etc.) or an issue with underlying computational infrastructure upon which VisIt depends (incompatible graphics drivers, a downed file system or network, a security environment trigger). These latter issues are typical of **Incident Response** services and require rapid response in diagnosing and enlisting resources to correct.

As is typical of most DOE software project teams, VisIt developers wind up having to serve in both the roles of **Customer Support** and **Incident Response**. So, we call this activity **Customer Response**. While some aspects of **Customer Response** are under the direct control of VisIt developers, many aspects are not and involve collaborations with other teams in resolving. In most cases the extent of the VisIt team’s involvement in operations is confined primarily to the VisIt software itself; its development, testing, release and deployment which includes installations we directly support on systems where we have accounts, hosted binary downloads for common platforms and the tools to build from sources.

The practice of having software development staff **integrated** with operations is more commonly referred to as DevOps. There is a pretty good video that introduces these concepts.

This document explains our procedures for how VisIt software developers handle active customer response incidents. Some of the goals of this process are . . .

- To try to **load balance** the work of customer response in an equitable way across the development team.
- To achieve a one-hour response time.
- To reduce incident response interruptions for the team as a whole.
- To identify and document escalation paths for major incidents.

### 4.10.1 Meeting the one-hour response time goal

This is a desirable but challenging goal. We agree as a team to try to meet it. But, we recognize that our schedules might not always permit strict adherence to this goal. So, as an absolute, last resort, **bare minimum**, we expect issues occurring 8am-3pm to be responded to by COB the same day. Issues occurring after 3pm but prior to start of next business day shall be responded to by noon of that next day.

Note that responding to a customer inquiry does not necessarily mean **resolving** it. Sometimes, the only response possible is to acknowledge the customer’s inquiry and let them know we intend to allocate resources to address it as soon as practical. In many cases, an immediate response to acknowledge even just the receipt of a customer’s inquiry .
with no progress towards actual resolution goes a long way towards creating the goodwill necessary to negotiate a
day or more of time to respond more fully.

4.10.2 One Week Cycles

For the VisIt project, normal business hours are 8am-12pm and 1-5pm, Monday through Friday excluding LLNL
holidays. The work of live customer response is allocated in one-week intervals. During a week, one developer is
assigned as the Primary customer response contact and a second developer is assigned as a Backup. Live customer
response incidents will be logged and tracked in a separate GitHub repository within the visit-dav GitHub
organization.

A schedule, showing the primary and backup assignments going out several months will be posted on a wiki page.
Ideally, on a team of 8, each developer will serve as primary only one week out of every two months. This leads to
fair load by head-count but isn’t weighted by project assignments. From a project development perspective, it would
be more appropriate for a developer that is only 50% time on VisIt to serve as the primary only half as often as a
100% time developer. On the other hand, few developers are 100% time on VisIt. The majority of VisIt developers
divide their time across multiple projects. Using 50% as the sort of nominal VisIt allocation, we expect to load
balance weeks of customer response work equally across all developers but infrequently have those with more than
50% time on VisIt doing an extra week.

In the short term, it will help to pair veteran backups with a primary who may have less experience with live
customer response.

4.10.3 The Primary’s Role

The primary’s role is to respond, within one hour’s time, to each incident that occurs during that week including those
that came in during the preceding weekend/holiday. Other developers who may be contacted directly by customers
should redirect such inquires to the primary. The primary’s goal is to wrap-up all incidents by the end of their week.
If an issue cannot be wrapped-up, it gets handed-off to next week’s primary. Handoffs are managed formally with an
email to customer and the next week’s primary and backup contact information.

For each new incident, the primary should file an issue ticket in the live-customer-response repository and assign
themselves. When the incident is resolved, the associated issue should be closed. Primary should endeavor to capture
all relevant information and communications involving the incident in this issue. For users who do not have GitHub
accounts, this will require the primary to do a modest amount of cutting and pasting between email and GitHub
issues.

The goal of the primary is to reach a resolution of each incident. Resolution often involves one or more of the
following activities.

• Answering a question or referring user to documentation.
• Determining if user’s issue is known (e.g. an issue ticket already exists).
• Updating a known issue with additional information from this user, perhaps reprioritizing the issue or putting
  the issue ticket back into the UN-reviewed state for further discussion at a VisIt project meeting.
• Diagnosing the incident.
• Identifying and subsequently fixing any low-hanging fruit.
• Developing a reproducer for developers.
• Developing a work-around for users.
• Identifying and filing a new coding issue in the main source repository.
In particular, please take note that resolution does not always mean a customer’s incident is able to be addressed favorably within the constraints of our customer response processes. Sometimes, the best that can be achieved is the filing of a highly informative issue ticket to be prioritized, scheduled and ultimately resolved as part of normal weekly development meetings.

Another role of the primary is to use any time not working active incidents to fix low-hanging fruit issues; either those the primary is currently managing or those left undone from previous incidents. For example, if an incident reveals incorrect or missing documentation, the primary should work to fix such issues within his/her week. As a rule of thumb, low-hanging fruit is considered to be anything that primary believes is fixable within a half-day’s (4 hours) worth of effort.

When primary has challenges with any of the paths to resolution above, s/he should feel free to engage other developers with help. But primary should first enlist the backup. When developer expertise other than backup is needed, primary should arrange mutually agreeable times to engage those other developers.

There shall be no expectation that a developer serving as primary can get any other work done beyond their customer response obligations. In slow weeks, its conceivable they can. But, there can be no implied assumption or expectation that this will be the case. Furthermore, even in a slow week, the primary may likely be able to find plenty of low-hanging-fruit-type tasks in the main repository issues to tackle and fix.

During their communication’s with the customer, the primary should avoid reply-all-type communications.

For applicable incidents, the primary should prepare a set of bullets to form an outline of a post-mortem for the incident to be reviewed at VisIt project meetings.

### 4.10.4 The Backup’s Role

Ideally, like the rest of the development team, the backup is never called into action for customer response. However, there are cases where the backup may be needed. For this reason, the backup is asked to at least track and maintain knowledge and awareness of the issues the primary is supporting. A good way to achieve this is to selectively watch the visit-live-incidents repository. In cases where primary needs help in resolving an incident, the backup shall serve as the first line of defense. In addition, the backup should be ready to step up to the primary role in the (hopefully rare) occurrence that the primary is unable to maintain the one-hour response time goal. To the extent possible, the transition from backup to primary should be managed formally. Otherwise, the backup should simply listen in but otherwise not engage.

### 4.10.5 Other Developer’s Role

In the ideal, the primary is able to handle all customer response activity and no other developers are interrupted. Thus, other developers should feel free to ignore customer inquiries and/or redirect customers who may contact them directly. One option to reduce visit-user traffic is to switch to receiving a daily digest. Other options might be email client side filters.

### 4.10.6 Escalation

Customer response incidents may escalate for a variety of reasons. The technical expertise or authority required may be beyond the primary’s abilities. Other difficulties may arise in responding to a given customer’s needs. For issues that the primary does not know how to resolve, the backup should be enlisted. It may just be a quick response that the backup can answer. If the backup cannot help, the primary should enlist other developers using the @ mention feature of the associated GitHub issue. However, where a primary is responsible for maintaining a one-hour response time, other VisIt developers so enlisted are free to either delay or even decline to respond (but nonetheless inform the primary of this need) if their current schedules do not permit timely response.
If the work required to resolve a customer response incident is either not known or not believed to be a low-hanging-fruit type task, the primary should search the issue system to see if this is a known issue and, if so, add additional information to that known issue about this new customer response incident (and perhaps remove the reviewed tag from the issue to cause the issue to be re-reviewed at the next VisIt project meeting) or submit a new issue to the main repository issues.

4.10.7 Special Considerations for SCF

Occasionally, incidents arise that may be handled only on the SCF. This is not too common but does happen and it presents problems for a geographically distributed team.

On the one hand, customers on SCF are accustomed to longer response times. On the other hand, often work on the SCF is a high priority and requires rapid response from a developer that is on site with access to SCF.

In many ways, an SCF-only incident is just a different form of escalation.

Our current plan is to handle this on a case-by-case basis. If neither the primary nor backup are able to handle a customer response incident requiring the SCF, the primary should

- First determine the customer’s required response time. It may be hours or it may be days. If it is days. Its conceivable the issue could be handled in the following week by a new primary/backup pair.
- If customer indicates immediate response required, primary should inquire the whole team to arrange another developer who can handle it.

4.10.8 Relation to Programmatic Work

When developers are faced with many development tasks, its easy to think of customer response work as an interruption to those tasks. However, customer response is indeed a critical aspect of programmatic work. It is part of what is involved in keeping VisIt a working and useful tool in our customer’s workflows not only here at LLNL, likely VisIt’s biggest customer, but wherever in DOE/DOD and elsewhere in the world VisIt is used. Often the issues users at large encounter and report are relevant to the use of VisIt here at LLNL.

4.10.9 To Do

- Decide when to switch to this new process
- Where do we keep weekly “calendar”? * maybe a wiki page or issue on live issues repo?
- Mark to get telephone forwarding procedure

4.11 OpenGL in VisIt

VisIt requires an OpenGL 3.2 context to work properly. Mesa provides a 3.3 context. Most desktop computers or laptops with graphics cards provide an OpenGL 4.6 or 4.7 context. For some unknown reason most (if not all) Linux HPC systems only provide a 3.0 context.

When using the QVTKOpenGLWidget with Qt, the following code snippet needs to be executed before creating the QApplication to tell Qt that it needs an OpenGL 3.2 context.

```cpp
// Setting default QSurfaceFormat required with QVTKOpenGLWidget.
```

(continues on next page)
auto surfaceFormat = QVTKOpenGLWidget::defaultFormat();
surfaceFormat.setSamples(0);
surfaceFormat.setAlphaBufferSize(0);
QSurfaceFormat::setDefaultFormat(surfaceFormat);

4.11.1 OpenGL in Qt

The sections of Qt that deal with OpenGL are

qtbase/src/opengl
qtbase/src/openglextensions
plugins/platforms/xcb/gl_integrations/xcb_glx
platformsupport/glxconvenience

The context creation is performed in

plugins/platforms/xcb/gl_integrations/xcb_glx/qglxintegration.cpp
void QGLXContext::init(QXcbScreen *screen, QPlatformOpenGLContext *share)

4.11.2 OpenGL in VTK

The sections of VTK that deal with OpenGL are

GUISupport/Qt
Rendering/OpenGL2

The context creation is performed in

GUISupport/Qt/QVTKOpenGLWidget.cxx

Other stuff is done in

Rendering/OpenGL2/vtkOpenGLRenderWindow.cxx

4.11.3 OpenGL documentation

GLX is the OpenGL extension to the X Window System. In the X Window System, OpenGL rendering is made available as an extension to X in the formal X sense: connection and authentication are accomplished with the normal X mechanisms. As with other X extensions, there is a defined network protocol for the OpenGL rendering commands encapsulated within the X byte stream.

Since performance is critical in 3D rendering, there is a way for OpenGL rendering to bypass the data encoding step, the data copying, and interpretation of that data by the X server. This direct rendering is possible only when a process has direct access to the graphics pipeline.

Good documentation on GLX can be found at:

GLX functions all start with “glX” and GLX constants all start with “GLX”.
Here is a link to some documentation about creating an OpenGL 3.0 context. It is the source of the test in build_visit to determine if the OpenGL on a system supports creating a 3.2 context.

https://www.khronos.org/opengl/wiki/Tutorial:_OpenGL_3.0_Context_Creation_(GLX)

Here is a link to some documentation about the history of the changes to OpenGL by OpenGL version.

Index

A
  AAN, 408
  Always, Auto, Never, 408

C
  Cell, 409
  Cell-centered, 409

N
  Node, 408
  Node-centered, 408

P
  Parallel task, 409
  Point, 408
  Point-centered, 409

S
  SIL, 409
  SR, 409
  SR mode, 409
  Subset Inclusion Lattice, 409

V
  Vertex, 408

Z
  Zone, 409
  Zone-centered, 409