Getting Started with EnSight 10.0

Computational Engineering International, Inc.
2166 N. Salem Street, Suite 101, Apex, NC 27523
USA • 919-363-0883 • 919-363-0833 FAX
http://www.ceisoftware.com

Copyright © 1994 - 2012, Computational Engineering International, Inc. All rights reserved.
Printed in the United States of America.

EN-GS Revision History

EN-GS:6.0-1	June 1997
EN-GS:6.0-2	August 1997
EN-GS:6.0-3	October 1997
EN-GS:6.0-4	October 1997
EN-GS:6.1-1	March 1998
EN-GS:6.2-1	September 1998
EN-GS:6.2-2	December 1998
EN-GS:7.0-1	December 1999
EN-GS:7.1-1	April 2000
EN-GS:7.3-1	March 2001
EN-GS:7.4-1	March 2002
EN-GS:7.4-2	October 2002
EN-GS:7.6-1	May 2003
EN-GS:8.0-1	December 2004
EN-GS:8.2-1	August 2006
EN-GS:9.0-0	September 2008
EN-GS:9.1-0	January 2010
EN-GS:9.2-0	November 2010
EN-GS:10.0-0	January 2012

This document has been reviewed and approved in accordance with Computational Engineering International, Inc. Documentation Review and Approval Procedures.

This document should be used only for Version 10.0 and greater of the EnSight program.

Information in this document is subject to change without notice. This document contains proprietary information of Computational Engineering International, Inc. The contents of this document may not be disclosed to third parties, copied, or duplicated in any form, in whole or in part, unless permitted by contract or by written permission of Computational Engineering International, Inc. Computational Engineering International, Inc. does not warranty the content or accuracy of any foreign translations of this document not made by itself. The Computational Engineering International, Inc. Software License Agreement and Contract for Support and Maintenance Service supersede and take precedence over any information in this document.

EnSight® is a registered trademark of Computational Engineering International, Inc. All registered trademarks used in this document remain the property of their respective owners.

CEI's World Wide Web addresses:
http://www.ceisoftware.com

Restricted Rights Legend

Use, duplication, or disclosure of the technical data contained in this document by the Government is subject to restrictions as set forth in subparagraph (c)(1)(ii) of the Rights in Technical Data and Computer Software clause at DFARS 252.227-7013. Unpublished rights reserved under the Copyright Laws of the United States.
Contractor/Manufacturer is Computational Engineering International, Inc., 2166 N. Salem Street, Suite 101, Apex, NC 27523.

Table of Contents

Introduction

What's in *Getting Started*?

This *Getting Started* manual contains the following information:

- An introduction to the EnSight user interface (Chapter 1).
- How to Start EnSight, stand-alone or distributed (Chapter 2).
- Some simple, step-by-step demonstrations of basic EnSight functionality (Chapters 3–6).

Conventions Used in *Getting Started*

The following typographic conventions are used in the *Getting Started* manual:

A numbered step tells you exactly what to do:

1. Change the value to "0.0" and press Enter.

UNIX and DOS level commands are denoted in a fixed-width font. Never type the leading "`%`" – it indicates that the command is to be issued at a shell prompt.

```
% ensight100.client -cm
```

Menu selections use ">" to indicate the selection hierarchy. For example, "Tools > Plane > Line" means to select Plane from the Tools menu and then select Line from the Plane cascade menu.

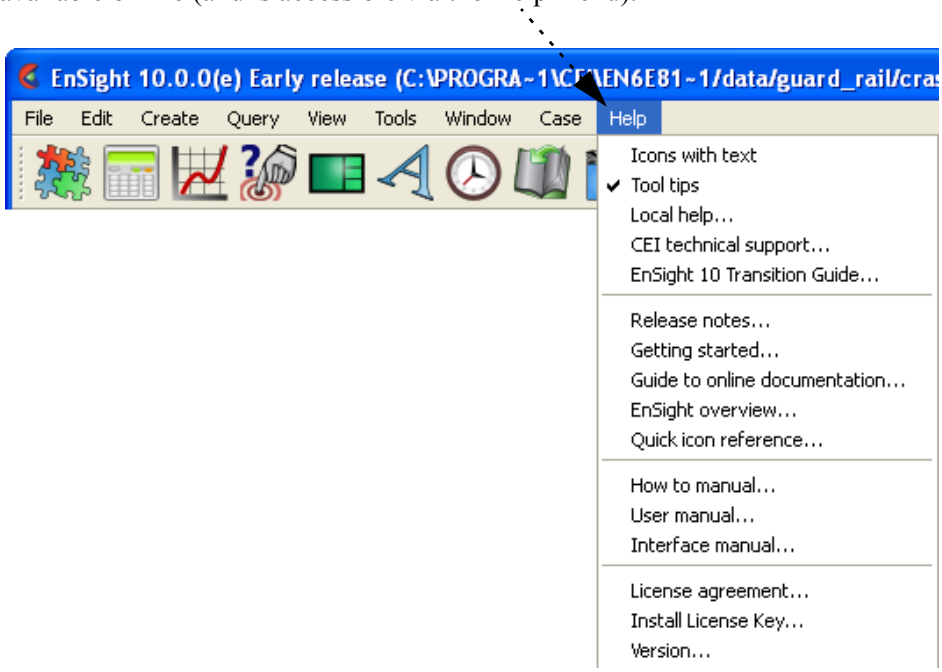
Notes and warnings provide particularly important information:

Note: Text emphasized in this fashion is a note.

Warning: Text emphasized in this fashion is a warning. Warnings typically indicate that your actions may have unintended consequences.

Where's the Rest?

The remainder of the EnSight documentation (as well as a version of this manual) is available online (and is accessible via the Help menu).



The other documentation is divided into three manuals:

How To

The How To documentation consists of relatively short articles that describe how to perform a specific operation in EnSight, such as change the color of an object or create an isosurface. Step-by-step instructions and pictures of relevant dialogs are included. In addition, each How To article typically contains numerous hyperlinks (colored blue) to other related articles (and relevant sections of the User Manual).

To access a list of the online How To articles, select Help > How To Manual.... From this location you can easily navigate to any of the articles or to the table of contents.

User Manual

The User Manual provides a detailed reference for EnSight.

To access the User Manual, select Help > User Manual.... From this location you can easily navigate to any of the chapters or the table of contents.

Note: When navigating in the online User Manual, you can easily return to the User Manual Table Title Page by clicking on the footer text: EnSight User Manual.

Interface Manual

The Interface Manual provides details on interface API's, as well as customization available through issuing commands to EnSight via a socket interface or through Python.

Note: The manuals contain useful navigation bookmarks which have been set up automatically for your PDF viewer. Also we encourage the use of the search function of your PDF viewer.

1 EnSight Graphical User Interface

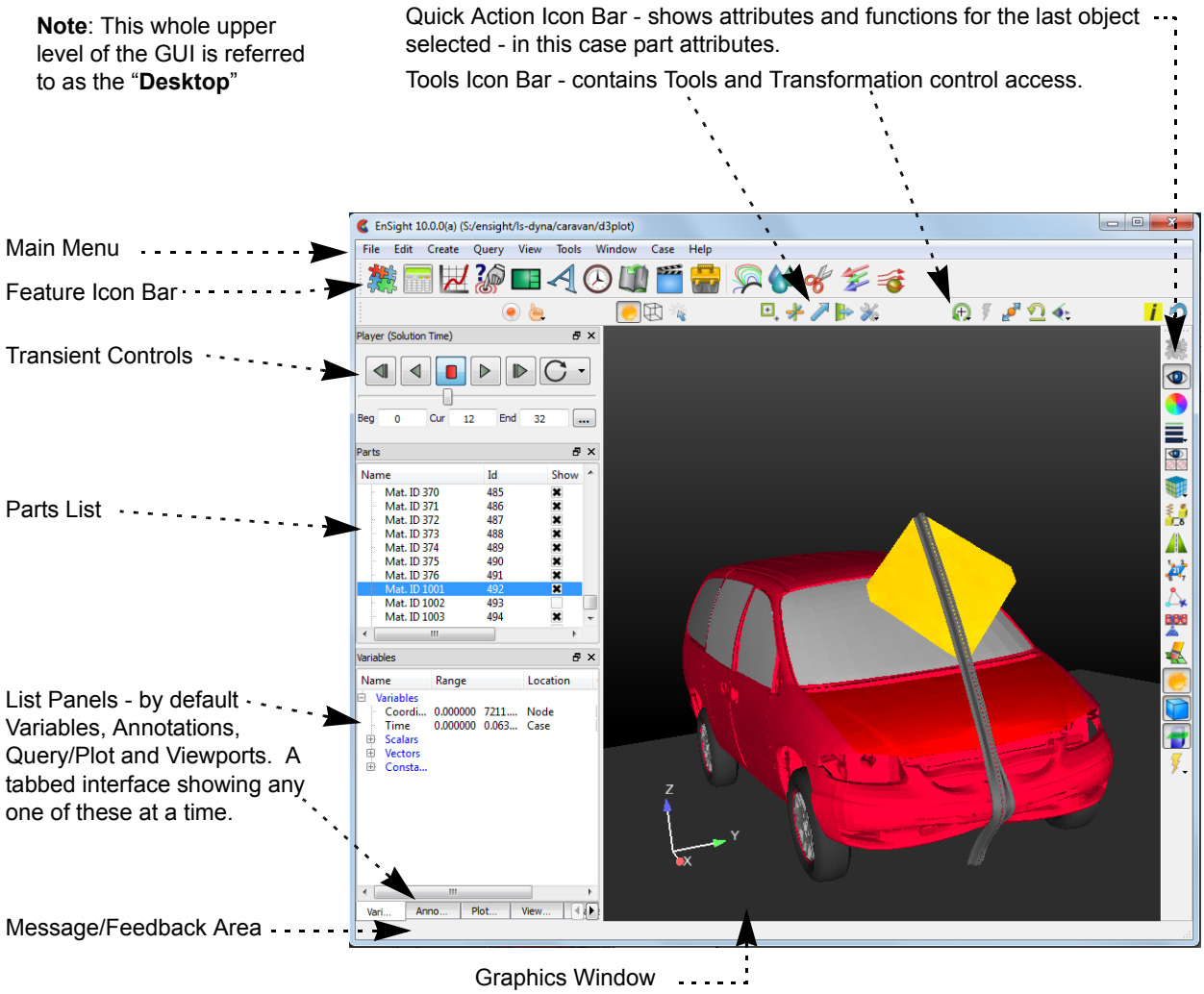
This chapter provides a quick introduction to the EnSight Graphical User Interface. In this chapter you will explore the layout of the user interface. Conventions used in the interface (such as how to select multiple items in a list) will also be discussed.

Note: If you are using EnSight Lite, this chapter will explain some icons/features which are NOT available to you.

1.1 Tour of the GUI

The major components of the EnSight user interface are shown below.

Note: This whole upper level of the GUI is referred to as the "**Desktop**"



Main Menu

The Main menu provides access to basic EnSight functionality. The Help menu (at the far right end of the menu bar) contains items for accessing online help.

Message/Feedback Area

The Message area displays brief messages during various operations.

Information Button

Clicking the Information button  in the Tools Icon Bar will bring up the EnSight Message Window where additional information about EnSight operations is displayed.

The color of the Information button will indicate when content is present  .

Parts List

The Parts List displays all *parts* associated with the current session. A part is a named collection of elements (or cells) and associated nodes. All components of a part share the same set of attributes (such as color or line width).

Parts are accessed via the Parts list. Items in the list are selected by placing the mouse pointer over the item and clicking the left mouse button. You can extend a selection by

pressing the shift key as you click an item. Additional techniques for selecting parts are discussed on page 1-10.

Understanding part concepts is crucial for productive use of EnSight. See section 3.3 *Parts and Part Attributes* for more information on parts.

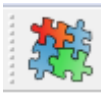
Feature Icon Bar

The Feature Icon Bar contains icons associated with the major "features" of EnSight (not all of which will be showing by default). Clicking the left mouse button on an icon selects the feature and opens the associated interface in the Feature Panel.

Attached to the Feature Icon bar are the secondary features. By default contours, isosurfaces, clips, vector arrows, and particle traces. These icons are simply fast access to the Features, i.e., if you click on the Clip icon it sets the Part feature into the clip setting.

All of the features (and secondary features) on the Feature Icon Bar can be customized. Simply right click on the Feature Icon Bar and select Customize Feature Toolbar...

Features include the following:

Part Icon		Opens the Parts Feature Panel which allows editing and creation of parts.
Variable Calculator		Opens the variable calculator, for creating computed variables.
Query/Plot		Opens the Query/plot Feature Panel allowing editing and creation of queries and plotters.
Interactive Query		Provides controls for specifying interactive queries, which display variable values as the mouse is moved over objects in the Graphics Window, as the cursor tool is moved within a volume, or at specific node, element, ijk, or xyz locations.
Viewports		Opens the Viewports Feature Panel and allows the editing and creation of viewports.
Annotations		Opens the Annotation Feature Panel and allows the editing and creation of various annotations such as text, lines, dials, etc.
Solution Time		Provides controls for managing time for transient datasets. This icon is only present for transient data.
Flipbook Animation		Provides controls for specifying Flipbook animations. Flipbook animations are on-screen animations that permit graphic transformations during playback. Flipbooks can be used to animate clipping planes and isosurfaces and are also useful for visualizing transient data
Keyframe Animation		Provides controls for specifying keyframe animations. Keyframe animation provides sophisticated motion control and output options for generating animations for either online presentation (*e.g.* MPEG) or video.

User Defined Tools		Provides a user interface to plug-in extensions from CEI and site or user customization.
Contour		Create or modify a new contour (isoline) part using the part(s) selected in the Parts list as parents and the specified variable.
Isosurface		Create or modify a new isosurface part using the part(s) selected in the Parts list as parents and based on an isovalue of the variable specified.
Clip		Create or modify a new clip part using the part(s) selected in the Parts list as parents. EnSight can create several types of clips including 1D line clips, planar clips, and quadric clips.
Vector Arrows		Create or modify a new vector arrow part using the part(s) selected in the Parts list as parents and the variable specified. Vector arrows display direction and magnitude of a vector variable.
Particle Trace		Create or modify a new particle trace part using the part(s) selected in the Parts list as parents and the variable specified as the velocity variable.

Other Secondary Features not shown in the typical start GUI (Use Right click to Customize the Feature Icon bar to add these)

Subset Parts		Create or modify a new subset part from node and/or element label ranges of model parts.
Profile Plot		Create or modify a new profile plot part using the part(s) selected in the Parts list as parents and the variable specified. A profile plot is the 1D counterpart of an elevated surface.
Elevated Surface		Create or modify a new elevated surface part using the part(s) selected in the Parts list as parents and the variable specified. An elevated surface is a surface projected away from another surface with scaling based on the value of a variable.
Vortex Cores		Create or modify a vortex core part using the part(s) selected in the Parts list as parents and the variable specified.
Shock Surfaces/ Regions		Create or modify a shock surface or region part using the part(s) selected in the Parts list as parents and the variable specified.
Separation/ Attachment Lines		Create or modify separation or attachment line parts using the part(s) selected in the Parts list as parents and the variable specified.

Material Part		Create or modify parts which are based on the intersection or domain of elements with mixed material values.
Tensor Glyph		Create or modify a tensor glyph part using the part(s) selected in the Parts list (as parents) and the tensor specified. This icon is invisible by default.
Developed Surface		Create or modify a new developed surface part using the part(s) selected in the Parts list as parents. A developed surface is constructed by unrolling a quadric clip about its axis of revolution.
Point Part		Create or modify a point part from specified locations in a parent part selected in the Parts list. A point part consists only of points and their variable values from the parent part. Think of a point part as a point clip.
Extrusion Part		Create or modify a new extrusion part using the part(s) selected in the Parts list as parents. An extrusion part is the 3D part created from rotating or translating a 2D parent part and incrementally creating 3D elements from the 2D parent.

Feature Panel

The Feature Panel provides the interface controls associated with the current feature selected from the Feature Icon bar.

The Feature Panels provide “Create” buttons. Clicking the Create button will build a new object (such as an isosurface, clipping plane, or text annotation) based on current settings. Once built, the new object will appear in one of the object list panels (part, annotation, viewports, etc.).

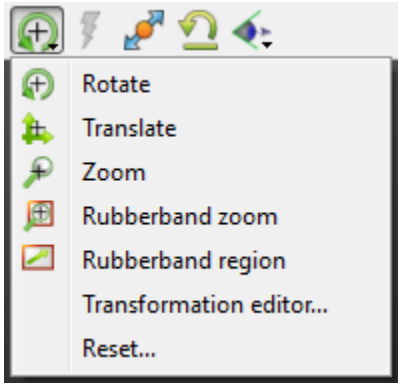
Quick Action Icon Bar

The Quick Action Icon Bar displays the icons associated with the most recently selected object (such as part, annotation, viewport, etc.). The icons displayed will depend on what object is selected. By default the Quick Action Icon Bar is displayed on the same level as the Feature Icon Bar but you are free to move it wherever you want by grabbing it by the tear off marker and dragging it to a new location. The Quick Icon Reference in the How To Manual online documentation provides a quick reference for all EnSight icons as well as hotlinks from the icons to relevant online articles.

Transformation Control Area of Tools Icon Bar

The Transformation Control Area provides icons that control various aspects of object transformations. Transformations are accomplished by selecting the action (such as rotate), moving the mouse into the Graphics Window, clicking and holding the left mouse button, and dragging the mouse to achieve the desired transformation. Also keep in mind that rotate, translate and zoom are by default attached to the left, middle, and right mouse buttons.

Several transforms are found in the transformations pulldown (which effectively maps the transform to the mouse key (by default left) attached to the "current' transform.



The possible actions are:

Rotate

Rotate: click and drag
- left-right to rotate about the vertical axis
- up-down to rotate about the horizontal axis
- left-right or up-down with the control key pressed to rotate about the screen Z axis.

Translate

Translate: click and drag
- left-right to translate in the horizontal direction
- up-down to translate in the vertical direction
- left-right or up-down with the control key pressed to translate in Z.

Zoom

Zoom: click and drag
- up/right to zoom out or down/left to zoom in
- with control key pressed to pan

Zoom is implemented by moving the look from point.

Band Zoom

Rubber-band zoom: click and hold the left mouse button on one corner of the desired viewing region, drag to opposite corner. An outline of the region will appear as you drag. Release the mouse button to zoom to the outlined region.

Rubberband region

Selection tool: click and hold the left mouse button on one corner of the desired viewing region, drag to opposite corner. An outline of the region will appear as you drag. Release the mouse button to see the selection tool. Adjust the center or corner locations of the tool further if needed. Then click the magnifying glass symbol near upper left corner to zoom. Or click the eraser symbol to blank out elements under the tool.

Transformation Editor...		Opens the Transformation Editor dialog. From within this editor, all global transforms (including scaling where appropriate) as well as frame and tool transforms can be controlled. There is also control of the center of transform, look-at/look-from locations, and z-clipping.
Reset...		Open the Reset Tools and Viewports dialog that permits easy resetting of all or some transformation operations.
Reset Transforms		Resets the global transforms for the current viewport.
Fit		Fits the currently visible model in the currently selected viewport and resets the center of transform to center of visible parts.
Standard or Saved Views		A pulldown to set +/- X/Y/Z views or to bring up the views manager to store/recall saved views.
Undo		Allows the user to undo/redo the last transformation.
information		Read informational feedback for EnSight status

Graphics Window

All 3D objects, as well as annotation entities, are displayed in the Graphics Window. The Graphics Window can contain additional (up to fifteen) user-defined viewports as well as X-Y plots.

1.2 User Interface Conventions

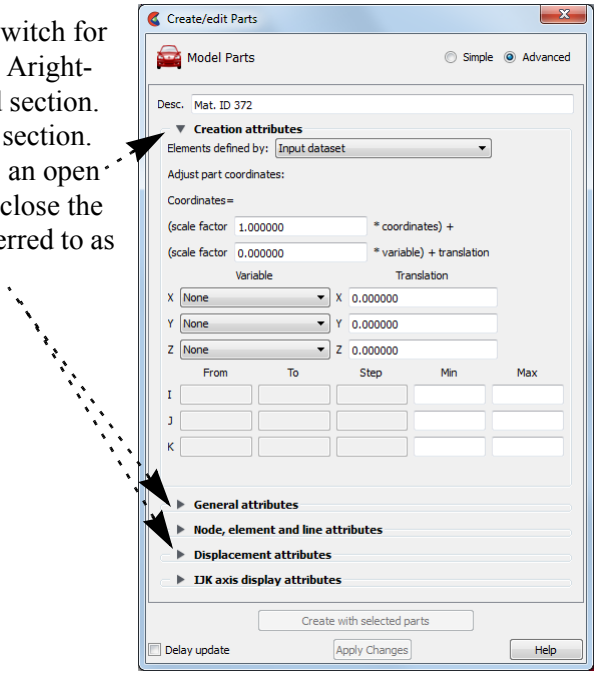
The EnSight user interface uses standard menus, dialogs, buttons, and other interface components. This section provides information on these components as well as instructions for interaction.

Dialog Windows

A dialog is a window that groups interface components based on function. Dialogs are typically opened by making selections from a menu or clicking an icon. Menu selections and icons that open dialogs usually end with "...". Most EnSight dialogs can be opened and closed independently. In order to optimize scarce workstation screen real estate, you should close dialogs that are not in use. The default position of each dialog was chosen to best use the space available. You can, however, move the dialogs using your window manager - the next time you restart EnSight the dialogs should be the same size and location.

Dialogs typically consist of buttons, menus, lists, and areas to type in. Some EnSight dialogs also have expandable sections that let you hide parts of the interface that you use infrequently. Each expandable section consists of an indicator button, a section title, and the contents of the section. The indicator button and the section title are always visible. If the section is open, the contents are visible as well.

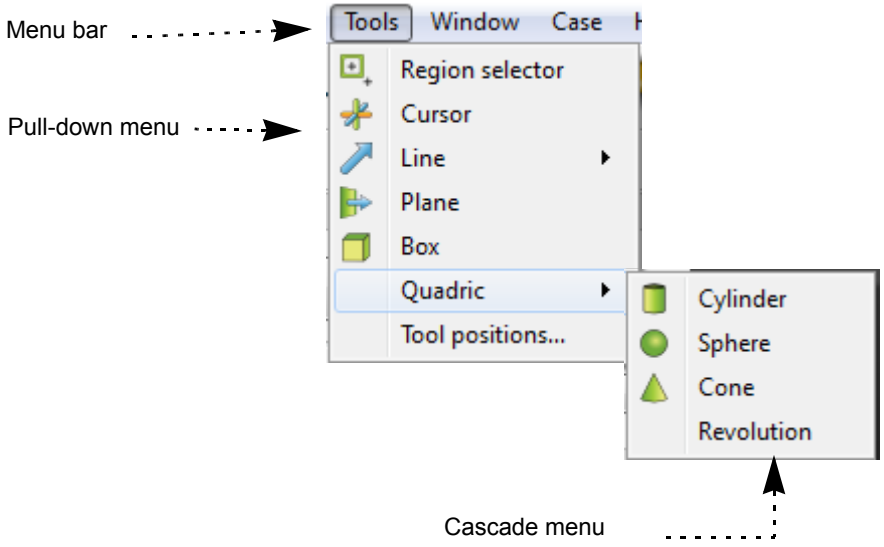
The indicator button is a toggle switch for opening and closing the section. A right-pointing arrow indicates a closed section. Clicking the arrow will open the section. A down pointing arrow indicates an open section. Clicking the arrow will close the section. These indicators are referred to as *turndown buttons*.



Menus

The EnSight documentation uses the following terms to describe various types of menus:

Menu bar	A horizontal strip across the top of a dialog listing menu titles.
Pull-down menu	A pull-down menu is one accessed directly from a menu bar.
Cascade menu or submenu	A submenu is accessed from another menu selection. Submenu selections are indicated by a right-pointing arrow.
Options menu or pop-up menu	An options menu is accessed by pressing the associated button. The current selection always appears as the button title.



Lists

Lists (such as Variable lists and the Parts list) are presented in panels or dialogs as scrollable sections. Various mechanisms are used to select items from a list for further action:

To...	Do This...	Details...
Select an item	Single-click	Place the mouse pointer over the item and click the left mouse button. The item is highlighted to reflect the "selected" state.
Extend a (possibly long) contiguous selection	Shift-click	Place the mouse pointer over the item. Press the shift key and click the left mouse button. This action will extend a selection to include all those items sequentially listed between the previous selection and this one.
Extend a non-contiguous selection	Control-click	Place the mouse pointer over the item. Press the control key and click the left mouse button. This action will extend a selection by adding the new item, but not those in-between any previously selected items.
De-select an item	Control-click	Place the mouse pointer over the selected item. Press the control key and click the left mouse button. This action will de-select the item.
Shortcuts	Right-click	Place the mouse pointer over the selected item in a list or in the graphics window and click the right mouse button. This action will pop up a context-sensitive pulldown list if available.

You can also double-click list items which is the same action as right clicking and selecting Edit... To double-click, place the mouse pointer over the item and click the left mouse button twice in rapid succession.

Buttons

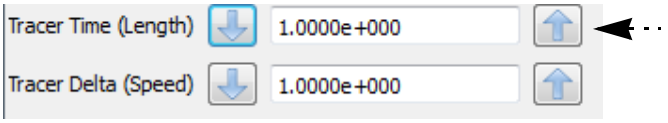
EnSight uses the following kinds of buttons:

Rectangular

Place the mouse cursor in the button area and click the left mouse button. Rectangular buttons typically access the function described in the label. If the label is followed by "..." then the button opens another dialog.

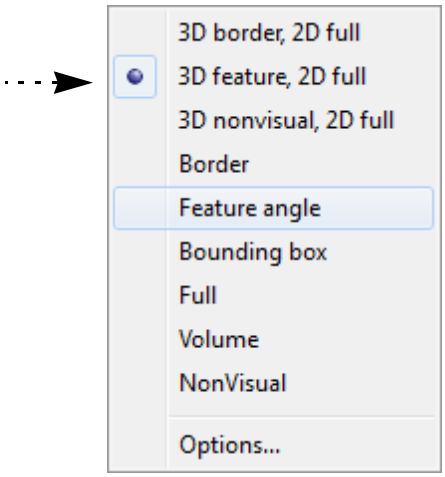
Arrow

Place the mouse cursor in the button area and click the left mouse button. Arrow buttons typically have an associated text field or scroll bar. Clicking the button increments or decrements the text field or scroll bar value.



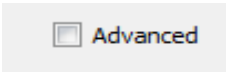
Diamond

Place the mouse cursor in the button area and click the left mouse button. Diamond buttons (also called radio buttons) are toggles that select an item from a mutually exclusive list. Exactly one diamond button of a group can be on at any given time.



Square

Place the mouse cursor in the button area and click the left mouse button. Square buttons are toggles that access the function indicated by the label.



Text Fields

EnSight utilizes two types of text fields:

Information Text Fields — These text fields are used to report information and cannot be edited by the user.

Editable Text Fields — Place the mouse cursor in the text field and click to insert a blinking insertion cursor. Several techniques are available to accelerate text editing. Select a single word by double-clicking or the entire string by triple-clicking. Selected text is replaced by subsequent typing. The left and right arrow keys (on most systems) will move the insertion cursor.

Where appropriate (*i.e.* in File Selection dialogs), EnSight recognizes the following shortcut specifications for directories:

`~/`	Expands to your home directory.
`~username/`	Expands to the home directory of `username`.
`./`	Expands to the current working directory.
`../`	Expands to the parent directory of the current working directory

Note that standard wildcard characters (*e.g.* `*` to represent a series of zero or more arbitrary characters) can also be used in File Selection dialogs.

1.3 Where's the Rest?

See Starting EnSight in Chapter 2 and familiarize yourself with the EnSight user interface, then proceed to Chapter 3, Simple Demonstration.

Several online articles provide overview and reference information. See the *EnSight Overview* (Help > EnSight Overview...) and the Quick Icon Reference (Help > Quick Icon Reference...).

For additional overview information, see Chapter 1 of the User Manual.

Finally, the table of contents as well as internal links, are easily navigated. For questions related to a specific task, see the How To Manual (Help > How To Manual...). For detailed information on a feature or concept, see the User Manual (Help > User Manual...).

2 Starting EnSight

EnSight can be started by double clicking icons or by typing commands in a shell window. EnSight can run in a stand-alone manner (client and server process both on the same machine, started simultaneously) or as separate processes (client and server started separately on the same or separate machines and provided connection parameters to link the two processes). This chapter provides a quick introduction to various ways of starting and connecting EnSight.

2.1 Necessary Prerequisites

EnSight must have been installed and the `CEI_HOME` and `PATH` environment variables setup properly. If you successfully performed the installation verification as described in the Installation Guide, you have verified that these things are correct. Note that the different versions of EnSight shown below can all be simultaneously installed to the same $CEI_HOME directory and work fine, using the same license manager.
(See `$CEI_HOME/ensight100/doc/Manuals/Installation.pdf`)

Note:
`ensight100` - *will start version 10.0*
`ensight92` - *will start version 9.2*
`ensight91` - *will start version 9.1*
`ensight90` - *will start version 9.0*
`ensight82` - *will start version 8.2*

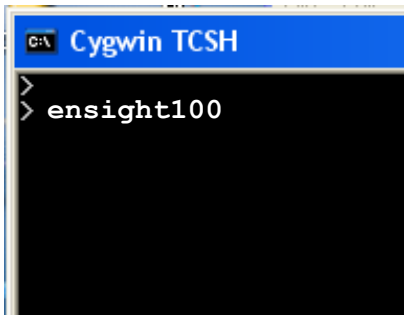
2.2 Starting EnSight for Stand-alone Use

If you want to run EnSight in a stand-alone manner (*i.e.* the Client and Server are both running on the same workstation), you can use these simplified steps to start EnSight and *auto-connect* the Client and Server processes.

1. Log into the machine on which the EnSight Client and Server have been installed.

For Linux Systems:

2. Open a shell window and start EnSight using the `ensight100` shell script:



Note:
`ensight100` - *will start version 10.0*
`ensight92` - *will start version 9.2*
`ensight91` - *will start version 9.1*
`ensight90` - *will start version 9.0*
`ensight82` - *will start version 8.2*

For Windows Systems:

2. Start EnSight by double clicking on the EnSight 10.0 icon on the desktop:

Or by choosing EnSight 10.0 from the Start > Programs > CEI menu.

For Mac Systems:

2. Start the EnSight100 app in the Applications folder.

This will automatically start both the Client and the Server and make the connection. To see if the connection is successful, you can click on the Information button on the EnSight Desktop. You should see “Connection accepted” in the EnSight Message Window which comes up. You should now close the EnSight Message Window and proceed to the next section.

If the connection failed, please consult Troubleshooting the Connection in the Installation Guide before contacting CEI support.

(See `$CEI_HOME/ensight100/doc/Manuals/Installation.pdf`)

2.3 Starting EnSight for Distributed Use - Manual Connection

You can perform a *manual connection* where the Client is started in manual connection mode waiting for a server, and the server is started separately and told to connect to the waiting Client.

See *How To Connect EnSight Client & Server* (Help > How To Manual ...) in the online documentation for details.

(For information on the online help facility, see *Using Online Help* on page 3-14.)

2.4 Starting EnSight for Distributed Use - Automatic Connection

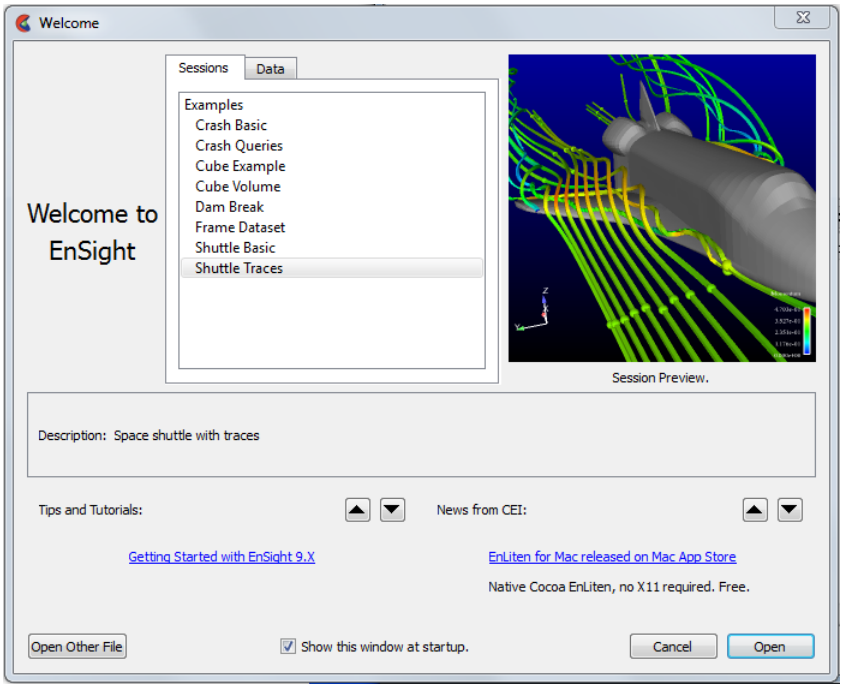
You can perform an *automatic connection* where the Server starts automatically and connects to the Client, even though the two processes are on different host systems. This type of connection requires some initial setup and is not discussed in the *Getting Started* manual. However, once configured, the automatic connection lets you start a session in a single step.

See *How To Connect EnSight Client & Server* (Help > How To Manual ...) in the online documentation for details.

(For information on the online help facility, see *Using Online Help* on page 3-14.)

2.5 Welcome Screen

The first thing you will see when you start EnSight 10 is the Welcome screen.

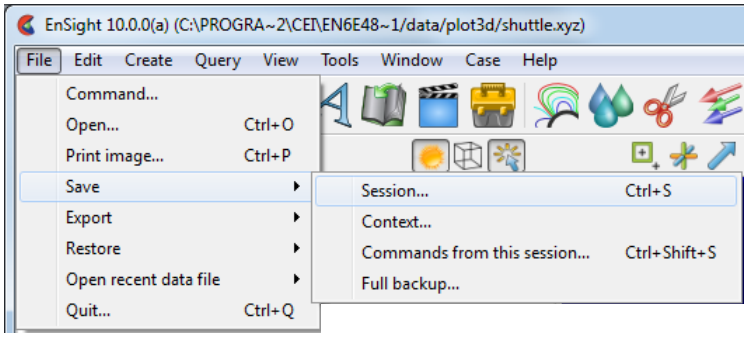


From the Welcome screen you can load a session file, most recently used data, view screencast tutorials, or jump to a file dialog to load new data.

Sessions Tab

A list of example session files are provided with the 10.0 release and show up in the Welcome Screen under the Sessions tab. As you select each one in the list, a preview image will appear together with a brief description. To load a session file simply select it and hit the Open button.

A session file is a binary file containing state information and may (optionally) contain the data. As shown below, a session file can be saved by the user by selecting the Save Session.... option from the File pulldown. Notice also that the version of EnSight as well as the dataset name is shown in the title bar at the top.



Data Tab

From the Welcome Screen Data tab you are presented with a list of the most recently used datasets that you have loaded. This is functionally equivalent to accessing this list via the File->Open recent data file.

To use simply select the dataset you wish to load and hit the Open button.

Tips and Tutorials

This area of the Welcome Screen dialog will show you a link to a screencast containing the tip described. The up/down arrow buttons allow you to browse the available screencasts. The screencasts are streamed and require an internet connection.

News from CEI

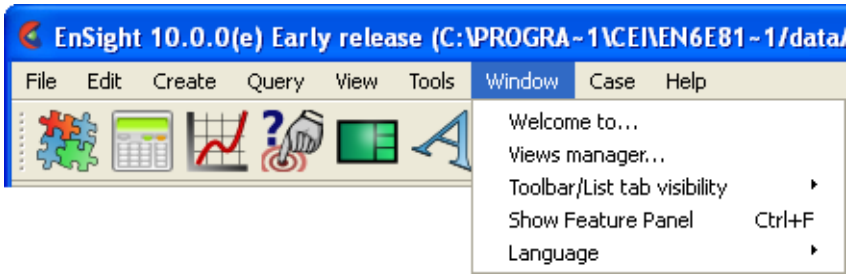
This area of the Welcome Screen dialog will show you a link to the latest communications from CEI. The up/down arrow buttons will allow you to browse the available news communications. This requires an internet connection.

Cancel

Select this button to dismiss this dialog. Next step for you will likely be an option under the File pulldown.

Welcome Screen Visibility

Toggle off the “Show this window at startup” if you do not wish to see this dialog at startup. You will still have the option to later see the Welcome Screen from EnSight’s Window menu as shown below.



3 Simple Demonstration

This chapter provides a step-by-step demonstration of basic EnSight operation. After successfully completing this chapter, you should be able to:

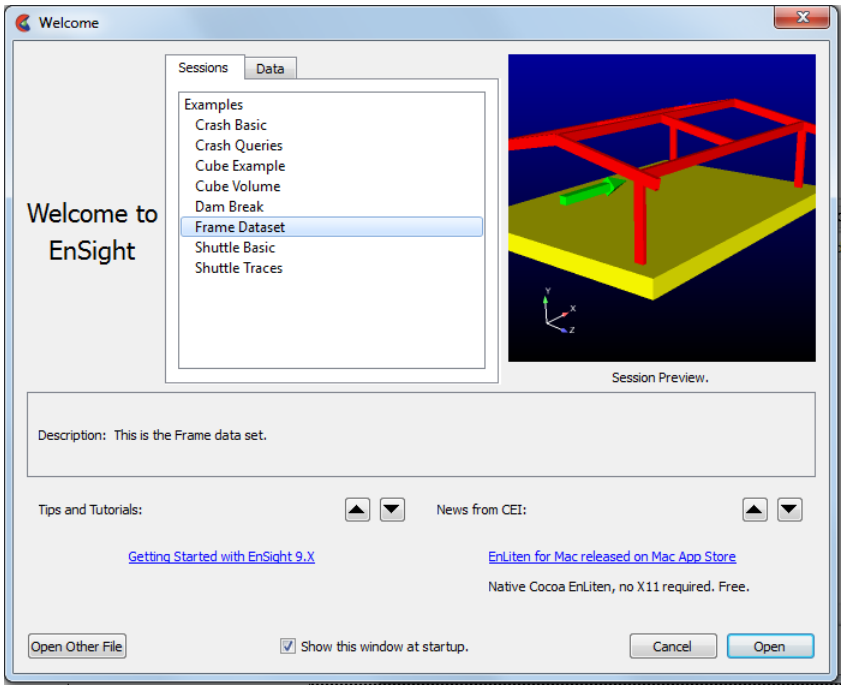
- read a dataset and load a model,
- transform objects in the Graphics Window: rotate, translate, and zoom,
- reset transformations,
- work with parts and change part attributes,
- save an image of the graphics window to a file,
- access the online documentation,
- exit EnSight.

3.1 Reading a Dataset

After starting EnSight, the next step in any session is to read a dataset and load the parts. To read a dataset, the relevant files and data format must be specified. EnSight supports several common computational analysis formats. In addition, EnSight also supports native formats suitable for storing both block structured and unstructured (*i.e.* finite-element) geometry. In this example, we will load an EnSight native format file.

Welcome Screen

After you start EnSight you will see the Welcome screen



From this window you can read sessions (data plus commands) or the most recently used data sets. In fact, for this particular example you can read the data by simply choosing Frame Dataset from the Sessions tab followed by a "Open". Your starting view will be slightly different than if you read the data with the instructions below but no harm done. Or select the Cancel button to load data from the File menu as described in the next few pages.

Drag and Drop

For many file formats, drag-n-drop will load your initial data. For recognized formats, you can start EnSight and drag the data file onto EnSight's window, or you can drag the dataset onto the EnSight executable icon on your desktop.

For more control over the data loading process, or to load additional data:.

1. Select File > Open... from the EnSight Main menu.

For EnSight to properly read data, it must know both the file name and format. When File > Open... is used, EnSight attempts to obtain this information from the suffix of the chosen file. If the file suffix is known to EnSight (as contained in an association file), it can proceed to read the data and load all of the files into EnSight. If an association is not known, EnSight will ask for more information. In this example, we will load a simple geometry in a known format.

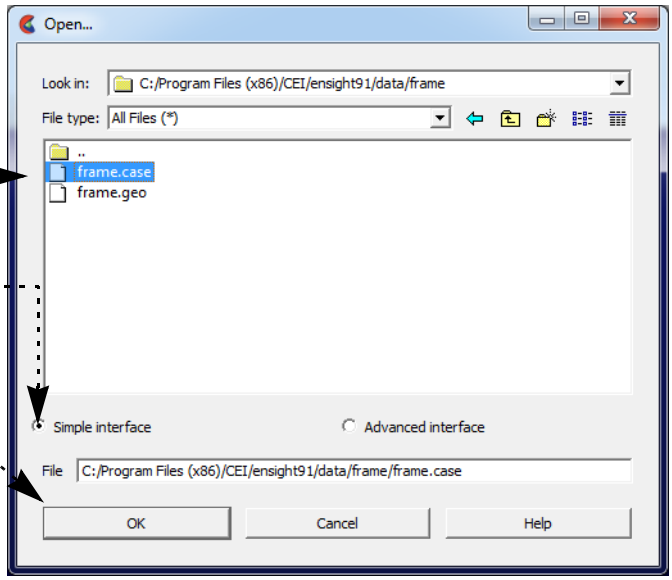
2. Navigate to the `$CEI_HOME/ensight100/data/frame` directory.

   Note: On Windows you can type "/" in the Look in: field to display the mounted drives.*

3. Click `frame.case`

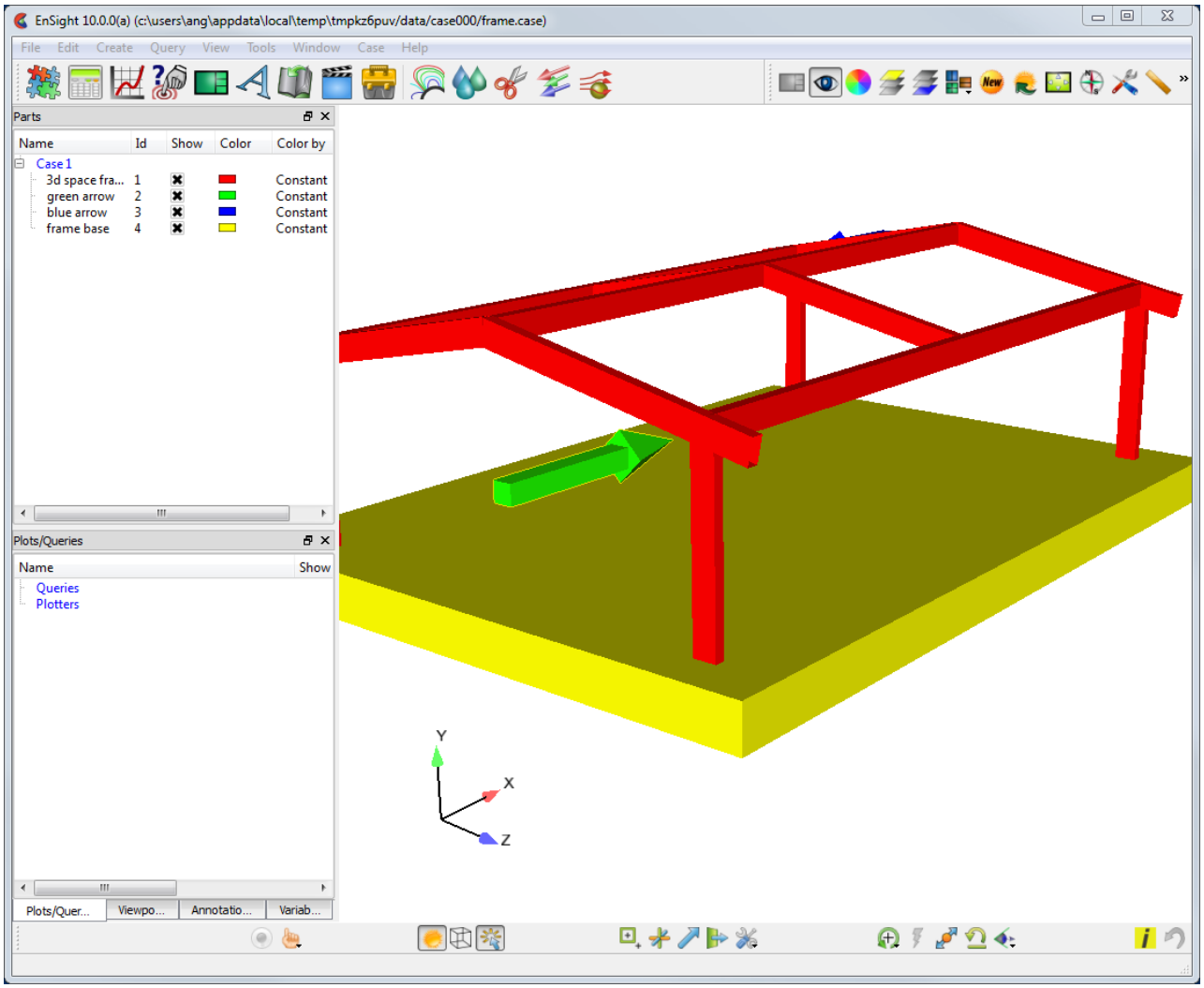
4. Make sure you are using the Simple Interface

5. Click OK to accept the selections and close the dialog window.



See Help > How To Manual ... (and select *How To Read Data)* for more information on data reading. In this case, you can also get there by clicking the Help button.

When the File Selection dialog is closed, EnSight reads the data and loads all four model parts (they will be listed in the Parts list and displayed in the Graphics Window). They will not be oriented as below, but that is okay.



Note: *To improve the legibility of the* Getting Started *manual, the images in the Graphics Window are shown with a white background rather than the default dark background that EnSight uses.*

If you desire to only load some of the parts, or to use different representations as they are loaded, this is easily done. See *How To Read Data* (Help > How To Manual ...) for information on the two-step method of reading and loading data.

*Getting **Your** Data Into EnSight*

EnSight supports a number of common data formats as well as interfaces to various simulation packages. In order to keep the list of readers and translators as current as possible, tables are maintained on our website. Please go to the following location to see the latest (**http://www.ceisoftware.com/data-interfaces/**).

3.2 Performing Transformations

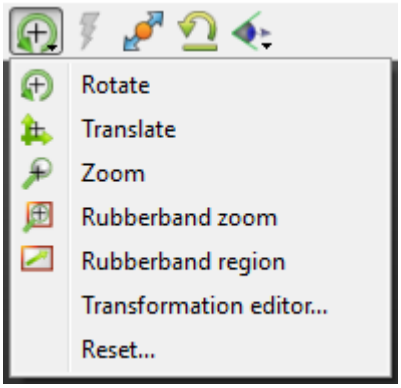
It's easy to rotate, translate, and zoom the geometry displayed in the Graphics Window.

Note: *By default EnSight ships with the left mouse button defined to do Transformation Actions (by default set to Rotate), the middle button click and move defined to do Translations directly, the right mouse button (or rolling the mouse wheel) defined to do Zoom operations directly.*

The assignment of operations to mouse buttons can be customized by the user.

Rotate using the mouse:

1. Move the mouse pointer into the Graphics Window.
2. Click and hold the ***left*** mouse button and drag the mouse left and right. The model rotates about the screen vertical axis.
3. Drag the mouse up and down. The model rotates about the screen horizontal axis.
4. Hold down the control key and move the mouse left and right. The model rotates about the screen perpendicular axis.
5. **Or**, to redefine the left mouse button (if it has been changed to something else), select the Graphics Window Transforms icon and choose Rotate from the pulldown.



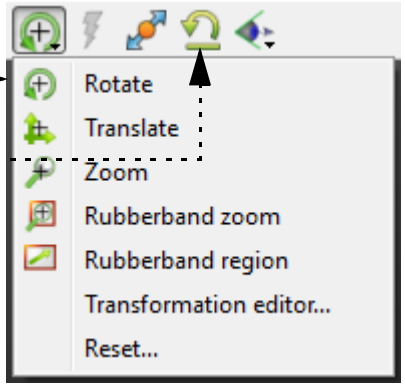
Translate using the mouse:	**or:**
1. Move the mouse pointer into the Graphics Window. 2. Click and hold the ***center*** mouse button and drag the mouse left and right. The model translates horizontally. 3. Drag the mouse up and down. The model translates vertically. 4. Hold down the control key and move the mouse left and right. The model translates in and out along the axis perpendicular to the screen.	5. To redefine the left mouse button, select the Graphic Window Transform icon and choose Translate from the pulldown. 6. Move the mouse pointer into the Graphics Window. 7. Click and hold the ***left*** mouse button and drag the mouse left and right. The model translates horizontally. 8. Drag the mouse up and down. The model translates vertically. 9. Hold down the control key and move the mouse left and right. The model translates in and out along the axis perpendicular to the screen.

Zoom using the mouse:	**or:**
1. Move the mouse pointer into the Graphics Window. 2. Click and hold the ***right*** mouse button and drag the mouse up or to the right. The “camera” zooms out. 3. Drag the mouse down or to the left. The “camera” zooms in. 4. Hold down the control key and move the mouse. The “camera” pans.	5. To redefine the left mouse button, select the Graphic Window Transform icon and choose from the pulldown Zoom. 6. Move the mouse pointer into the Graphics Window. 7. Click and hold the ***left*** mouse button and drag the mouse up or to the right. The “camera” zooms out. 8. Drag the mouse down or to the left. The “camera” zooms in. 9. Hold down the control key and move the mouse. The “camera” pans.
Note that the zoom operation actually moves the virtual camera rather than moving the geometry.	

See *How To Rotate, Zoom, Translate, Scale* (Help > How To Manual ...) for more information on model transformations.

At this point, you have probably transformed the model into a strange orientation. Before going on set the transformation mode back to Rotate

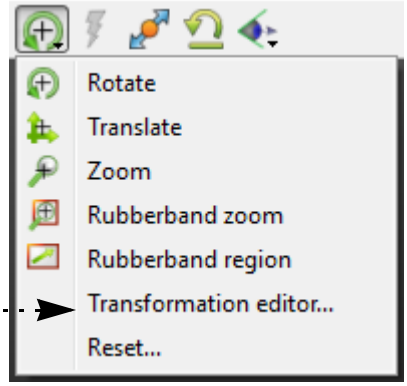
Then click on the Reinitialize View icon.



See *How To Reset Tools and Viewports* (Help > How To Manual ...) for more information on resetting transformations.

You can also perform precise transformations (such as rotating 22.5 degrees about the X axis) using the Transformation Editor

1. Click Transformation editor... in the Transformation pulldown.



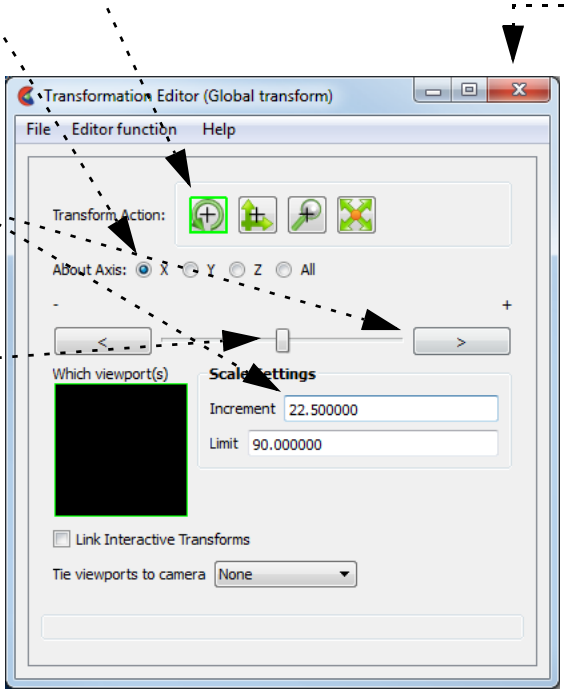
This opens the Transformation Editor dialog in Global Transform mode.

2. Select Rotate icon in the Transformation Editor dialog.
3. Select X Axis.
4. Place the mouse pointer in the Increment field and triple-click the left mouse button to select the entire value “1.000000”.
5. Enter the value “22.5”, then slick the right arrow button.

The model will be rotated precisely 22.5 degrees about the X axis (in either the negative or positive direction) when you click the arrow buttons.

6. Grab the slider (click and hold the left mouse button) and drag left and right. Note that the actual number of degrees rotated is printed above the slider thumb switch as it is moved.

Experiment with the Transformation Editor – set Axis to other values or change the current action in the Transformation Control area to edit translation or scale. (If the model doesn’t seem to move in the translate mode, increase the increment and limit, press Enter, and move the slider again.) When you are done:



7. Click Close.
8. Reinitialize all transformations by clicking on the Reinitialize View icon as described above.

3.3 Parts and Part Attributes

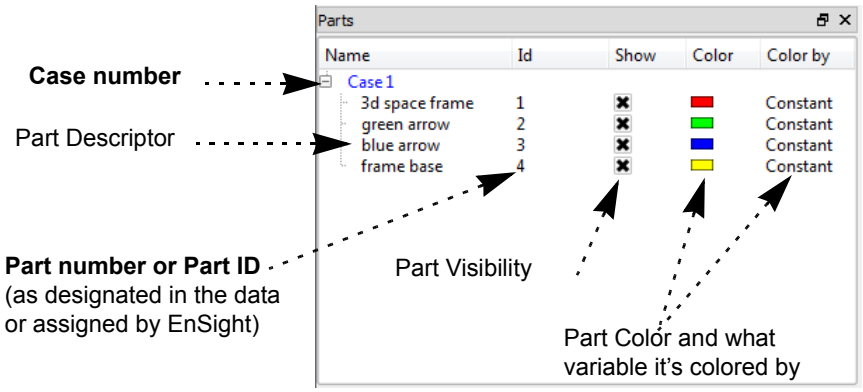
Since virtually every task you perform in EnSight involves some form of part manipulation, it is vital to understand how parts work.

Parts are either built during the reading/loading process (such as File > Open...) or created during an EnSight session. Parts created during reading/loading are called *model parts* and are based on your computational mesh and associated surfaces as defined in your data files. Model parts can also be created during an EnSight session by performing geometric operations (such as a copy) on other model parts.

All other parts are created during an EnSight session and are called *created* or *derived* parts. Created parts are built using one or more other parts as the *parent parts*. The created parts are said to *depend on* the parent parts. If one or more of the parent parts change, all parts depending on those parent parts are automatically recalculated and redisplayed to reflect the change. Examples of created parts include clipping planes, isosurfaces, isocontours, and particle traces.

Only model parts will be used in this section. The next two chapters will work with created parts.

The Parts list provides access to all parts. Each part is listed individually in a scrollable list. By default, each entry provides a part descriptor (name) and additional pieces of information:



EnSight provides a large number of *attributes* that can be edited on a per-part basis. Attributes control the appearance or behavior of parts. Examples include visibility, color, line width, and transparency. Part attributes for selected parts can be changed via the Quick Action Icon bar. Some global part attributes can be changed in the View Menu. For example, Main Menu > View > Shaded (or the shaded toggle on the desktop) enables/disables shaded surface display for all parts. For example, there is a shaded surface icon in the Quick Acton Icon Bar for parts that enables/disables shaded surface display on a per-part basis, and a global toggle on the Tools Icon Bar that turns it on and off on a global basis.

Many operations in EnSight (such as setting attributes) require that the parts to be acted upon are selected (highlighted) prior to the operation. Items in the Parts list are selected by placing the mouse pointer over the item and clicking the left mouse button. You can extend a selection by pressing the shift or control key as you click an item. Additional techniques for selecting items in lists are discussed on page 1-10.

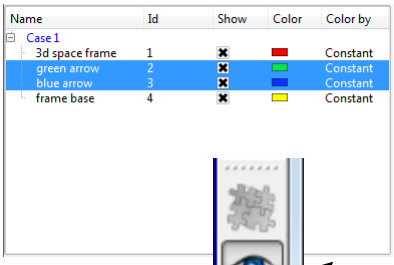
In the remainder of this section, we will explore parts and changing part attributes.

The first task is to enable shaded (hidden surface) display. By default, the part-specific shaded attribute is on. By toggling on the corresponding Shaded toggle on the Desktop, we enable hidden surface display for all parts.

1. If the Shaded Toggle is not ON, click it to toggle shaded surface display for all parts (icon becomes brighter). Leave shaded ON.

All objects in the Graphics Window are now displayed with shaded surfaces. Turning this toggle off will result in a wireframe display. We can now selectively disable shaded display for individual parts.

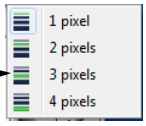
2. Select the "green arrow" and "blue arrow" parts in the Parts list: place the mouse pointer over the green arrow part, click the left mouse button, followed by a Ctrl left click (push and hold the Ctrl button on the keyboard and press the left mouse button) on the blue arrow part. When completed both parts should be highlighted in the Part list. Note that the selected parts are highlighted in the graphics area. To toggle this off, click on the Highlight Selected parts icon.Toggle it back ON.



3. Click the Part Shaded Toggle to disable shaded surface display for the selected parts.

Note that the Part Shaded Toggle icon changes (the icon will be dimmed) to reflect the fact that the selected parts have shaded disabled.

Let's change the line width of the selected parts:

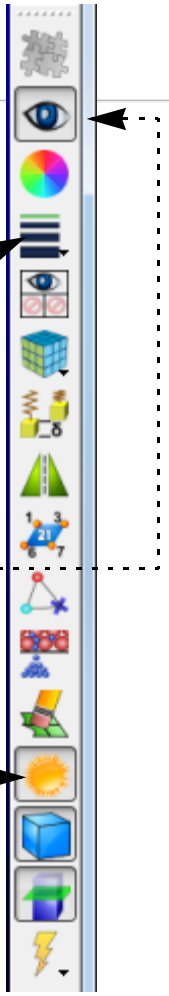


4. Click the Part Line Width pulldown and select 3 Pixels from the menu.

Note that the line width of the selected parts (the arrows) has increased.

5. Select the "frame base" part in the Parts list.
6. Click the Part Visibility Toggle to disable display of the part.

After these operations, the geometry in your Graphics Display should look like the following.



Parts are assigned a default color when loaded into EnSight. These colors are *constant*, meaning that every portion of the part is colored the same. Parts can also be colored by a variable value. Since the value associated with a variable typically varies from node to node, the displayed color will vary across the surface of the part.

To change part colors:

7. Select all the parts in the Parts list by using the Left mouse button to click on the Case1 line in the Part list.
8. Click the Color/transparency icon in the Quick Action icon bar.

The Part color, lighting, & transparency dialog opens. (Note that the variables list is set to Constant Color.)

9. Pick a shade of gray from the bottom row of the color palette.

Note that all parts in the Graphics Window are now colored by the same shade of gray.

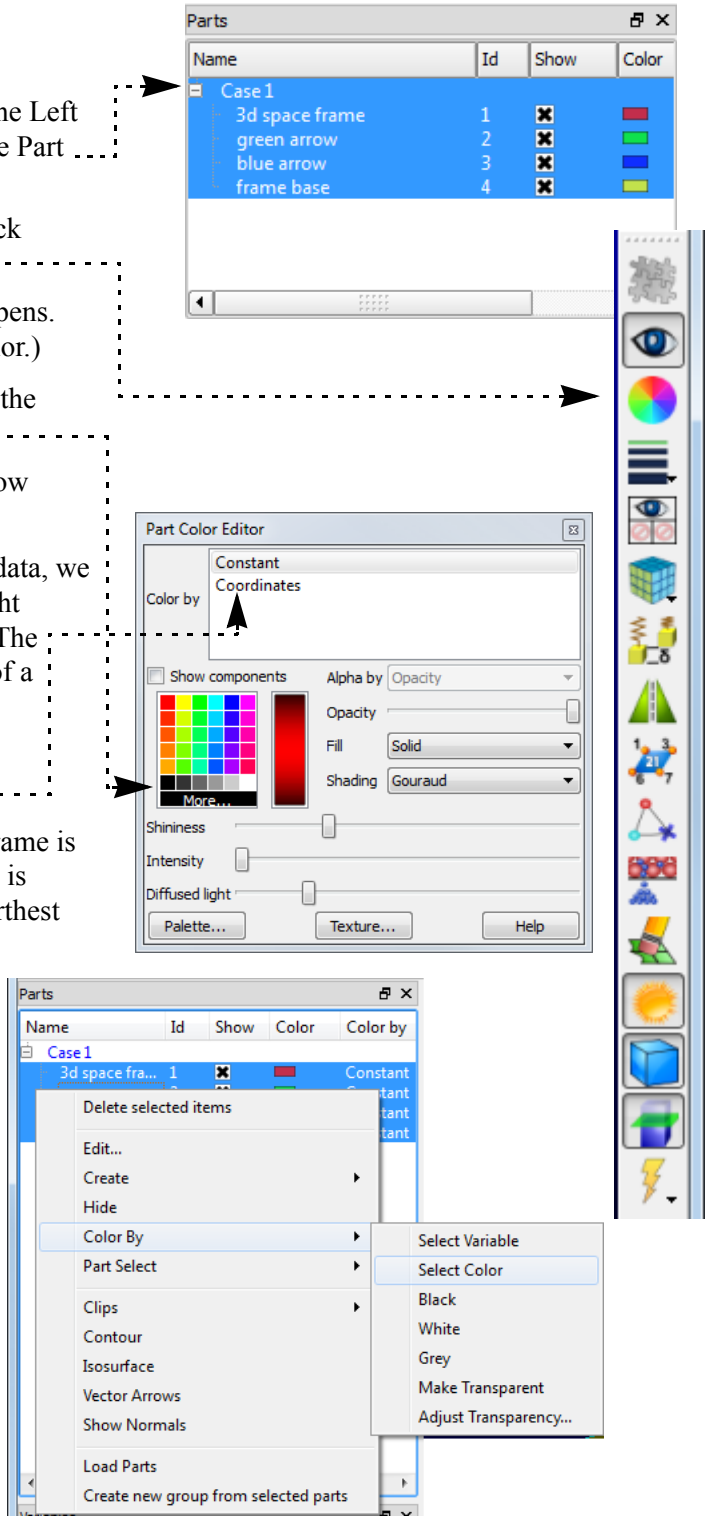
Although this dataset has no associated variable data, we can still color the parts by a variable since EnSight provides some defaults: Coordinates and Time. The Coordinates variable treats the XYZ coordinate of a node as a vector; color is assigned based on the magnitude of that vector.

10. Select Coordinates from the Color by list.

Note the color change. The left rear post of the frame is closest to the origin of the coordinate system and is colored blue. The right front post of the frame farthest from the origin is colored red.

Another way to color a part is to utilize the extensive right click capability. To test this reselect all of the parts using Step 7

11. Right click on any of the selected parts in the list which brings up a right click menu.
12. Select “Color By” then either select “Select Variable” or “Select Color”

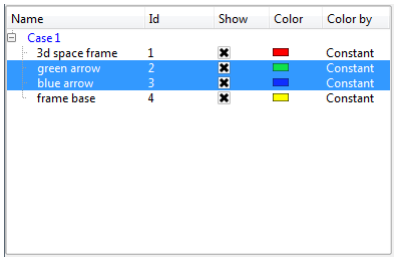


See *How To Change Color* (Help > How To Manual ...) for more information.

Up to this point, all parts have been selected by clicking in the Parts list. You can also select parts by picking them in the Graphics Window. Although this dataset contains only four parts, most contain many more and selecting parts via the list can become tedious. To select parts by picking:

13. Move the mouse into the Graphics Window and place the pointer over the arrow on the left side of the model.
14. Left click on this part.

Look at the Parts list: the "green arrow" part (number 2) should now be the only part selected. (If this is not the case, be sure the mouse pointer is directly over one of the lines of the part left-click again.)



15. Move the mouse pointer over the arrow on the right side of the model.
16. Press and hold the control key. With the control key still down, left-click the mouse button while it is over the right arrow part.

Holding down the control key during the pick *extends* the current selection: both the "green arrow" and the "blue arrow" parts should now be selected in the Parts list. See *How To Select Parts* (Help > How To Manual ...) for more information.

Note that the selected parts are highlighted in the graphics window to assist you in identifying selected parts.

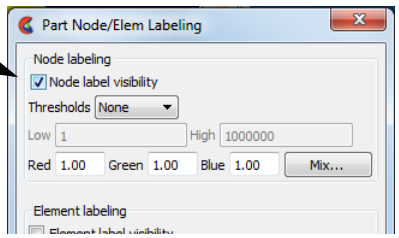
17. Toggle the highlight parts toggle OFF.

EnSight indicates in the graphics window which parts are selected as long as the global toggle is ON.

18. Now toggle the highlight parts toggle ON.
19. Click the Part Shaded Toggle to enable shaded surface display for the selected parts.
20. Select the "frame base" part in the Parts list.
21. Click the Part Visibility Toggle to enable display of the part.

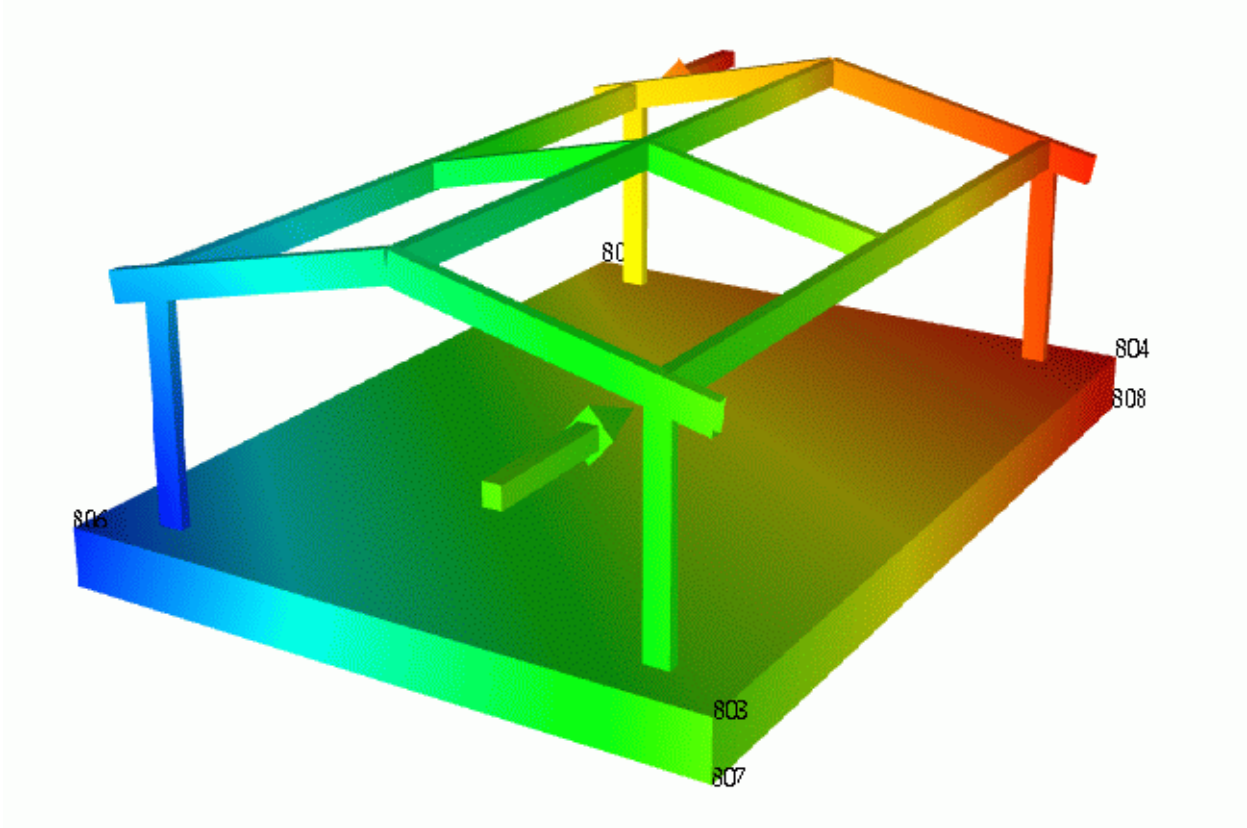
EnSight can display node and element labels on selected parts. The label values are either provided explicitly from the dataset or are provided by EnSight. To display node labels:

22. Click the Node/Element Labeling icon, and toggle on Node label visibility in the dialog that comes up, then close the dialog.



See *How To Display Labels* (Help > How To Manual ...) for more information.

After these operations (and some rotate and zoom transformations) your Graphics Window should look something like the following. (Note that the node labels have been colored black here to contrast with the white background – your labels will be white on a darkened background.)



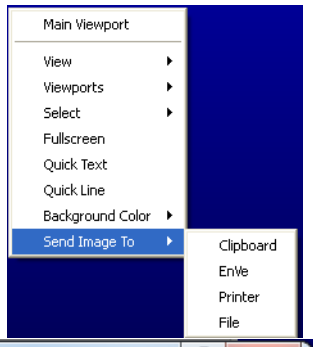
3.4 Saving Files

Saving Images

EnSight supports several formats for image output. In this example, both PNG and JPEG files will be saved.

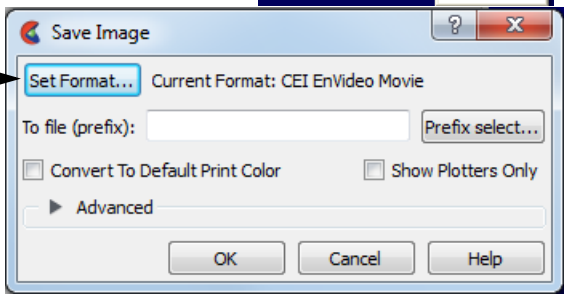
For Quick output:

1. Right click on the graphics window background and select Send Image To and choose File. Name the file and pick a location. The image is saved in PNG format, the size of the graphics window, with all the default image settings.

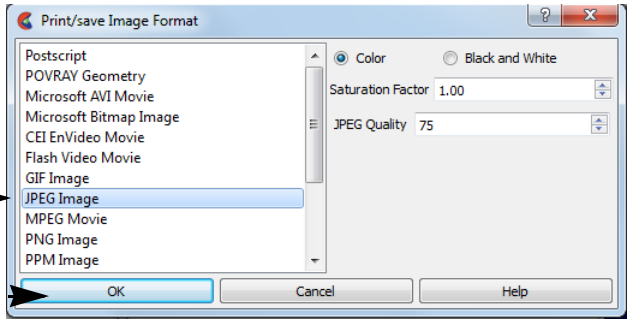


Detailed output options:

2. Select File > Export > Image... to open the Save Image dialog. By default, EnVideo is selected.

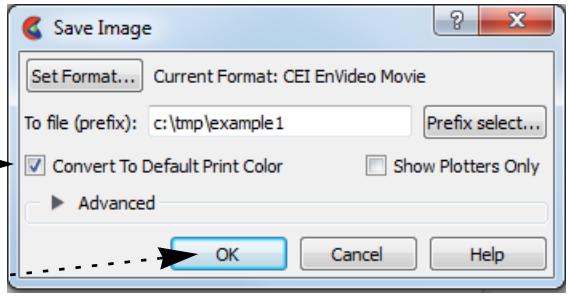


3. Click on Set Format... and select JPEG image file.



4. Click OK to close the format dialog.
5. Enter a file prefix in the To File field, and hit Return.

Note: *By default, EnSight will save images in the directory from which the Client is started. Since this directory is part of the EnSight distribution, it is probably write protected. To save the image in your home directory instead, on a Linux system prefix the filename with "~/". On a Windows system, use "~\". Note on Windows, ~ = C:\Documents and Settings\username*



6. Click on Convert to Default Print Colors (changes background to white and white objects to black).
7. Click OK

See *How To Print/Save an Image* (Help > How To Manual ...) for more information on image formats and options.

3.5 Using Online Help

EnSight 10 provides several manuals online. The two you will use most often are:

How To — The How To documentation consists of relatively short articles that describe how to perform a specific operation in EnSight, such as change the color of an object or create an isosurface. Step-by-step instructions and pictures of relevant dialogs are included. In addition, each How To article typically contains numerous hyperlinks (colored blue) to other related articles (and relevant sections of the User Manual).

User Manual — The User Manual provides a detailed reference for EnSight.

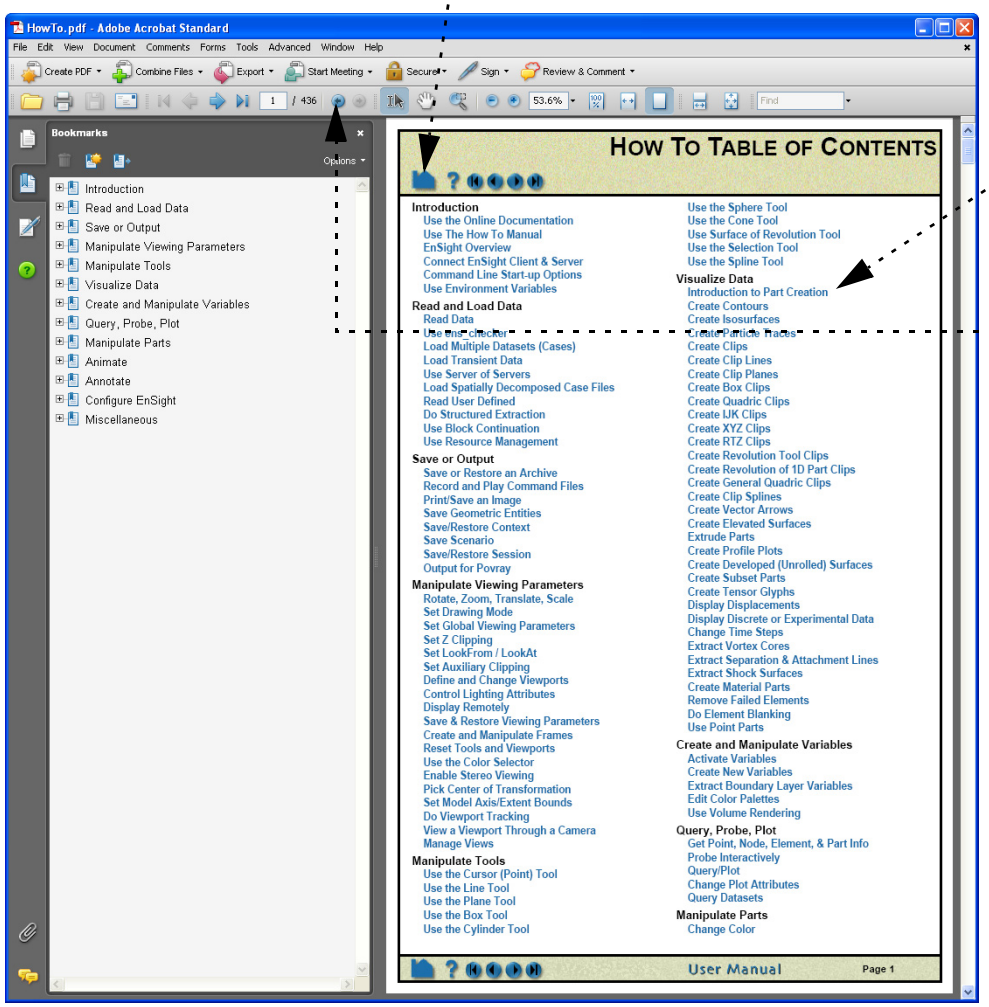
Several documents are directly accessible from the main Help menu. The remainder can be accessed through hyperlinks or a table of contents. Most of the complex dialogs within EnSight have help buttons that will open a corresponding *How To* article.

To open the *How To Table of Contents*:

1. Select Help > How To Manual...

The EnSight online documentation uses a PDF viewer. The PDF viewer provides much the same functionality as a World Wide Web browser while providing greater control over document content quality. The user interface is very simple and provides intuitive navigation controls.

Besides the use of bookmarks for navigation, The following can be used:



- Clicking on the house icon will return you to the How To table of contents.
- Blue text is a link: clicking on it will jump to a new location.
- You can go back to the previously viewed page by clicking the back button (very similar to a Web browser).

The *How To* articles are heavily cross-linked and also provide links to relevant sections of the *User Manual*.

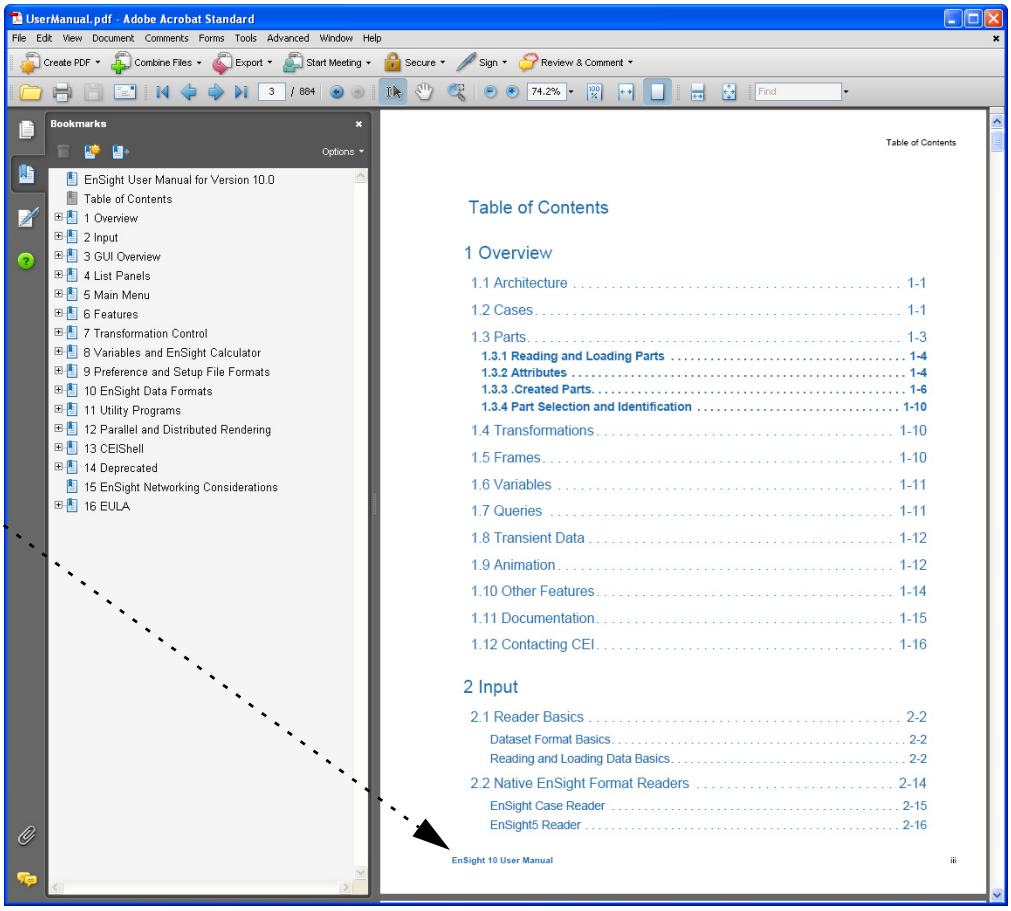
For more information on using Acrobat Reader (including printing) and the *How To* documentation, click on the question mark beside the house icon.

To access the *User Manual* table of contents:

1. Select Help > User Manual... from the EnSight Main menu, then click "Table of Contents" on the title page itself or in the Bookmark list.
2. The *User Manual* contains blue hypertext links just like the *How To* articles: you can click on items and jump to a new location:

- chapter and section entries in the table of contents
- cross references in the text that begin "See ..."
- the footer of every page will jump back to the title page of the User Manual (from which you can jump to any chapter).

3. Close the reader when done.



Printing the documentation

Adobe Acrobat .pdf files for all documentation are included on the EnSight CD distribution (and were placed in the $CEI_HOME/ensight100/doc/Manuals directory during the installation process). These documents (GettingStarted.pdf, HowTo.pdf, UserManual.pdf, and others for advanced users) have been optimized for printing and formatted for letter-size paper.

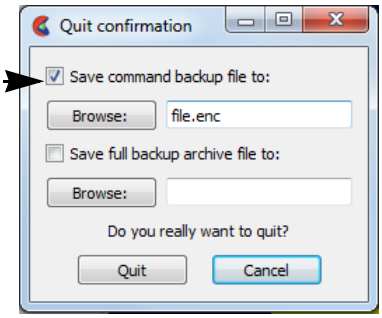
You can open these files and print any or all pages from within a pdf reader, or you can order printed copies from our website.

3.6 Exiting EnSight

All EnSight actions have a counterpart in the EnSight command language. During a session, all actions are being recorded to a default command file. When you exit EnSight, you have the option of saving this command file.

To quit EnSight:

1. Select File > Quit... from EnSight's Main menu. The Quit Confirmation dialog opens.



If you toggle on Save Command Backup File To and save the commands (for example, to a file called `file.enc`), you could then start a subsequent Linux version EnSight session (from the same starting directory) with the command:

```
% ensight100 -p ~/file.enc
```

(assuming you are running stand-alone) to duplicate the entire session. You could accomplish the same thing by starting EnSight normally and then playing the command file from the command dialog (this would be the normal way to do it in the Windows version: File>Command and load the command file). See *How To Record and Play Command Files* (Help > How To Manual ...) for more information. In Windows, you can start EnSight and just drag the .enc command file from the desktop to the EnSight window and EnSight will recognize it as a command file and run it.

Note that the Quit Confirmation dialog also provides an option to save a "Full Backup Archive File". An archive (which is saved for both the Client and the Server), contains a binary memory dump of the complete state of each system. Unlike replaying a command file (which has to re-execute every action), the restoration of an archive occurs very fast since only the saved memory state for each system is restored. See *How To Save or Restore an Archive* (Help > How To Manual ...) for more information.

2. Click OK to exit EnSight.

3.7 Where's the Rest?

After successfully completing this tutorial, you should proceed to the next demonstration. Although the next tutorial contains some material applicable to all users, it is intended primarily for analysts doing computational fluid dynamics (CFD) work. If your application area is not CFD, you may wish to consider skipping the next chapter and proceeding directly to Chapter 6.

The How To Manual contains details on the operations covered in this chapter. The following online articles are relevant to the topics covered here.

Note: you get to these by selecting Help from the main menu, then How To Manual... (Help > How To Manual...), then the topic indicated below:

Consult...	For More Information On...
How To Read Data	specifying data to read into EnSight
How To Rotate, Zoom, Translate, Scale	performing transformations in the Graphics Window as well as performing precise transformations using the Transformation Editor
How To Reset Tools and Viewports	resetting transformations back to the default settings
How To Select Parts	selecting parts
How To Set Attributes	setting part attributes

4 Flow Visualization Example: Unstructured Mesh

This chapter provides step-by-step instructions for performing many basic postprocessing operations – especially those relevant to computational fluid dynamics analysis. After successfully completing this chapter, you should be able to:

- create a clipping plane and display contours on the plane,
- move the clipping plane with the mouse (interactive clipping),
- create an isosurface and change the isovalue interactively,
- create a single particle trace and a rake of traces,
- move the rake of traces with the mouse (interactive particle tracing),
- animate particle traces,

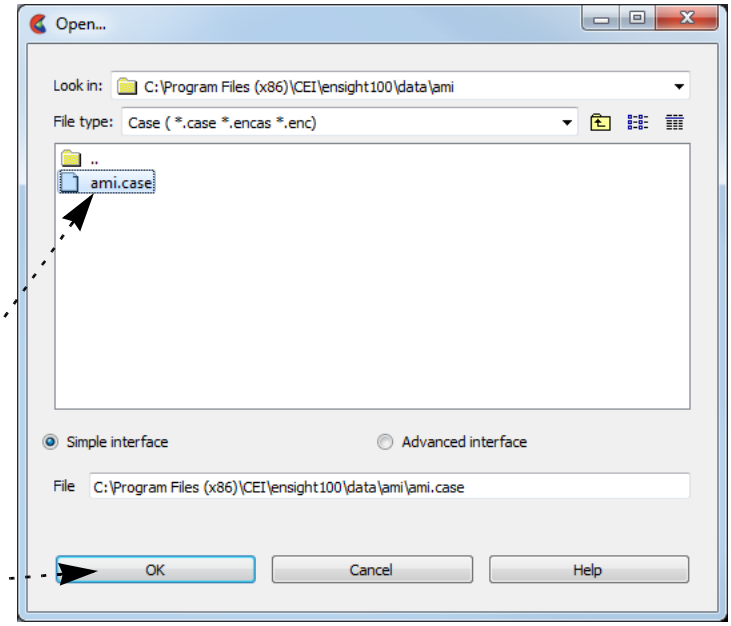
4.1 Reading a Dataset

In this demonstration, we will load a simple CFD model of a hypersonic vehicle with an 8 degree angle of attack and a 6 degree side-slip. The dataset includes velocity and pressure values.

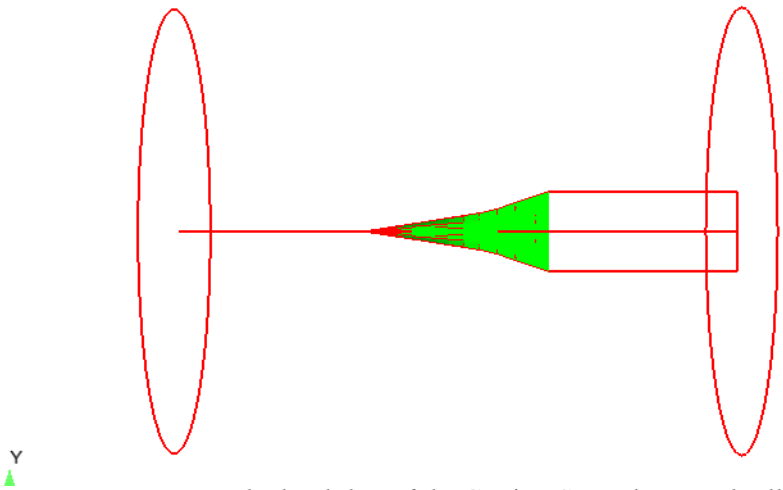
1. Start Ensight as described in Chapter 2 and cancel the Welcome Screen.
2. Select File > Open... from the EnSight Main menu.

This opens the File Selection dialog.

3. Make sure the Simple Interface is selected.
4. Navigate to the `$CEI_HOME/ensight100/data/ami` directory.
5. Click `ami.case` in the Files list.
6. Click OK to accept the selection and close the dialog window.



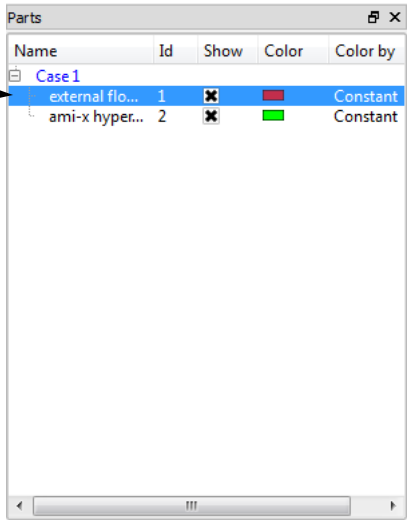
The two constituent parts are now listed in the Parts list and displayed in the Graphics Window (with the flow field in feature angle mode and the hypersonic body part in full mode)



Note: *To improve the legibility of the* Getting Started *manual, all images of the Graphics Window are shown with a white background rather than the default darkened background that EnSight uses.*

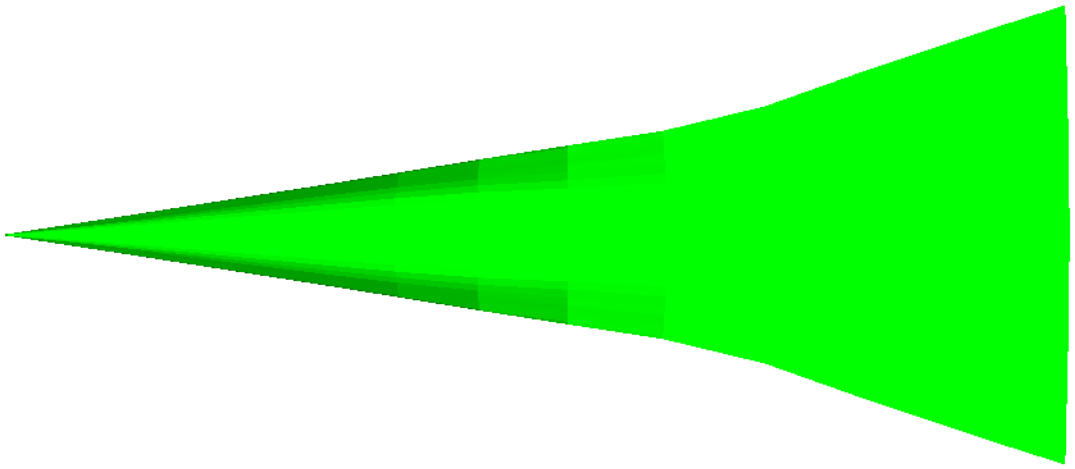
Since we don't really need to see the flow field, we will make it invisible and fit the geometry to the window.

7. Select the external flow field part in the Part List.
8. Click the Visibility Icon in the Quick Action Icon bar.



9. Click Fit.

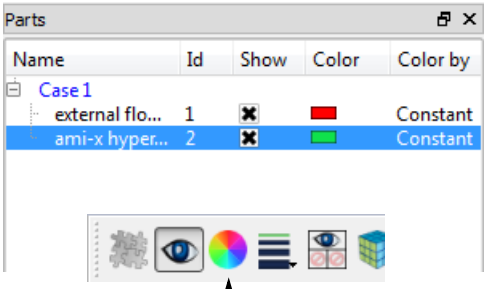
The model should now look like the following (but may be zoomed in further):



4.2 Feature Demonstration

Unlike the dataset used in the previous chapter, this data contains two variables: pressure and velocity. The first step is to color the model by the pressure variable and display a color legend showing the mapping from variable values to color.

1. Select the “ami-x hypersonic body” part in the Parts list.

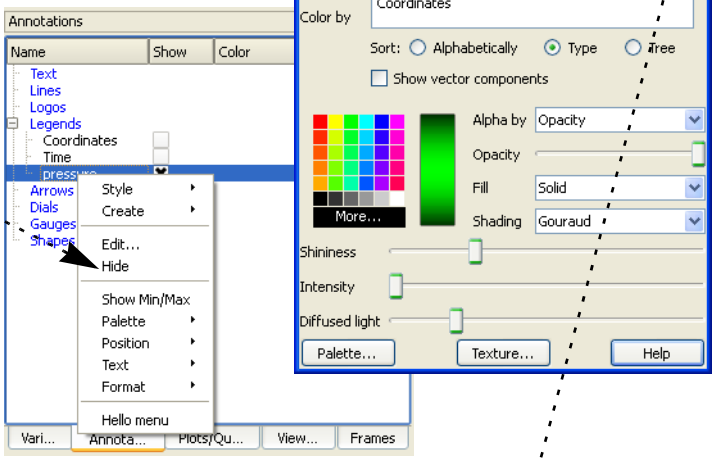


2. Toggle off the Highlight Parts When Selected toggle to turn off part selection highlighting.

3. Click the Color/transparency icon in the Quick Action icon bar to open the Color editor.

4. Select pressure variable in the ‘Color by’ list.

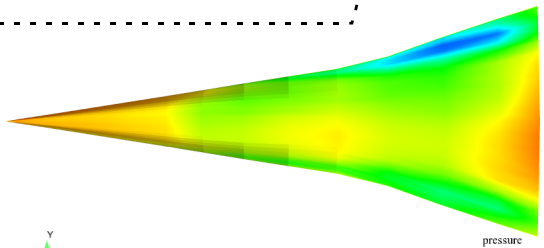
The color legend appears to the right of the model in the Graphics Window (default behavior - this is a user-controllable preference). Note that the display of legends can be controlled by right clicking on the proper Legend in the Annotation object list.



We won’t actually modify the legend visibility at this time. Also note that color legends have many display attributes – see *How To Create Color Legends* (Help > How To Manual ...) for more information.

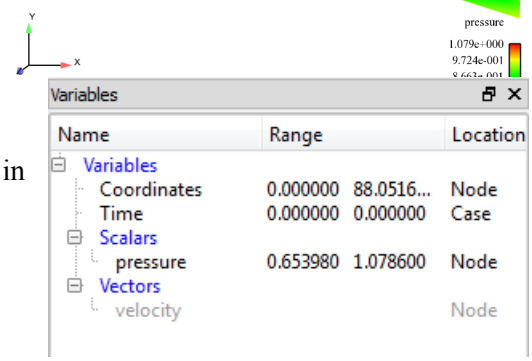
5. Click the X to close the dialog.

Your Graphics Window should now appear similar to the following:



There are several other ways to color a part by a variable. If your hardware is properly set up you can use drag and drop.

6. In the Variable list open the Vectors group to find Velocity
7. Click with the left mouse button on Velocity and keep the mouse button pressed while you drag the mouse into the background of the graphics window.
8. Release the left mouse button. This will now color all parts in the viewport by Velocity, as well as change the legend.
9. Open the Scalars group and drag and drop Pressure into the graphics window.

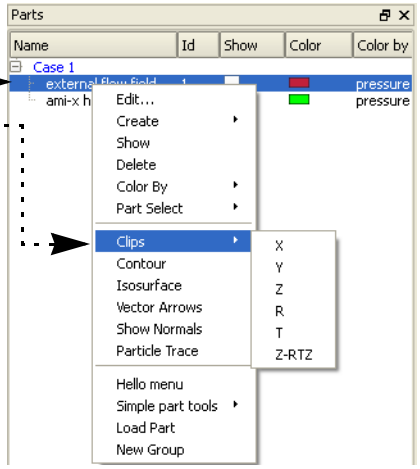


In the next sequence of operations, we will create a new part: an X clip.

Once the clipping plane has been created, we will build a contour part using the clipping plane as the parent part.

Clipping Plane Part

10. Like many things in EnSight there are several ways to perform the X clip operation. We will take advantage of the right click capability. In the Parts list right click on the "external flow field" part.
11. Select Clips, then X to create an X clip through the part you right clicked on at the default position, i.e., the mid point in the X direction.



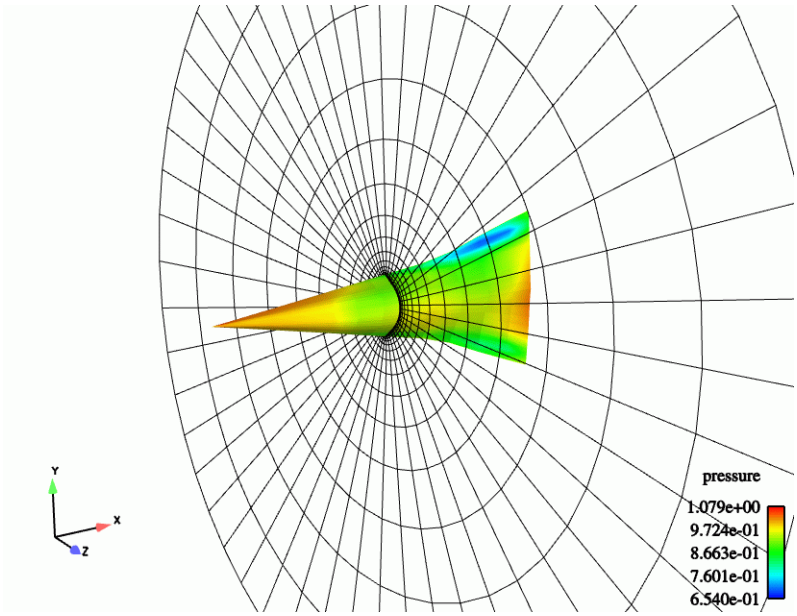
Note that a new part "Clip_plane" is now listed in the Parts list. There is also a new part in the Graphics Window but you cannot see it since it is perfectly parallel to our line of sight: rotate the model to see the clipping plane. See *How To Create Clips* (Help > How To Manual...) for more info.

12. Move the mouse pointer into the Graphics Window, click and hold the left mouse button and drag to rotate the model. *Similarly, you may want to zoom the model by dragging the right mouse button (or rotating the mouse wheel) in the Graphics Window.*

It might be better if the clip plane was not shaded, so we could see the model through it.

13. If the Quick Action Icon bar is not currently showing part icons (i.e., the toolbar is showing the first icon to be a puzzle piece), click on the Clip_plane part in the Parts list. Then click the Shaded icon in the Quick Action Icon bar.

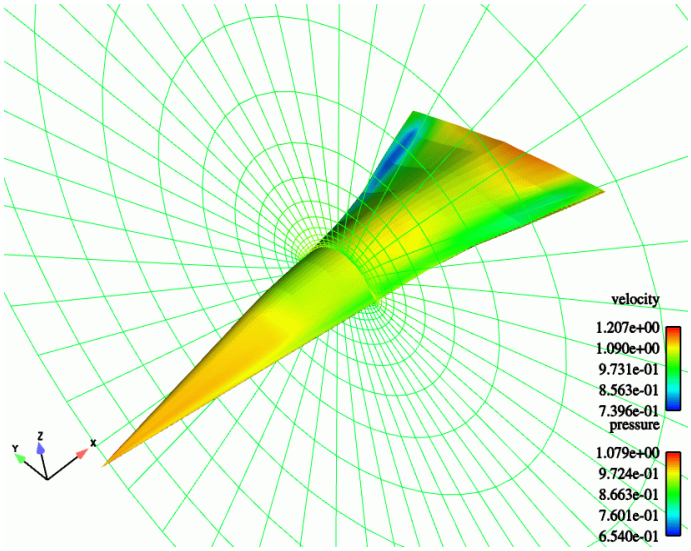
Depending on how you rotated and zoomed, you should now see something like the following in the Graphics window.

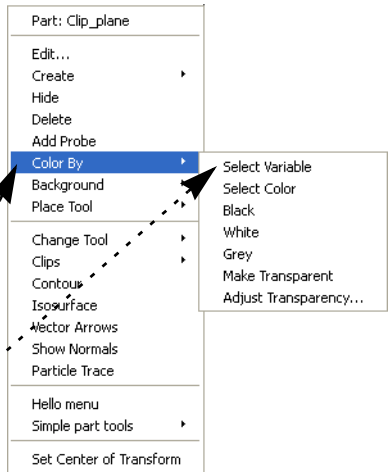


The new "Clip_plane" part listed in the Parts list has attributes just like the original model parts. For example, we can color the part based on the value of a variable and change other attributes as well (like we just did with its visibility).

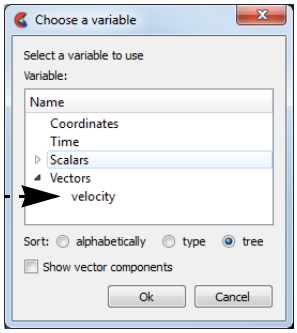
14. Click the Color icon in the Part Quick Action Icon bar.
15. Select velocity in the variables list and then close the dialog.

Your Graphics Window display should look something like the following:





16. A quick way to color a part is to right click on it and select Color By> Select Variable in the resulting popup
17. In the popup window select Velocity and click OK. The clip plane is colored by Velocity.
18. Or click and drag the variable out from the Variable list and drop it on the part in the graphics window.



The following should be kept in mind when creating new parts in EnSight:

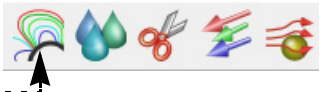
- A new part is created from one or more parent parts. You must select these parts either in the Part list or by selecting them in the graphics window. If you use right click the selection is performed during the right click operation.
- A part creation type is selected - either from right click, selecting the feature from the Secondary Feature bar, or selecting it from the Create pulldown.
- The new part is created either directly or through the Feature Panel after possibly modifying the creation attributes.
- Any desired attributes - visual or creation - are modified for the new part.

Contour Part

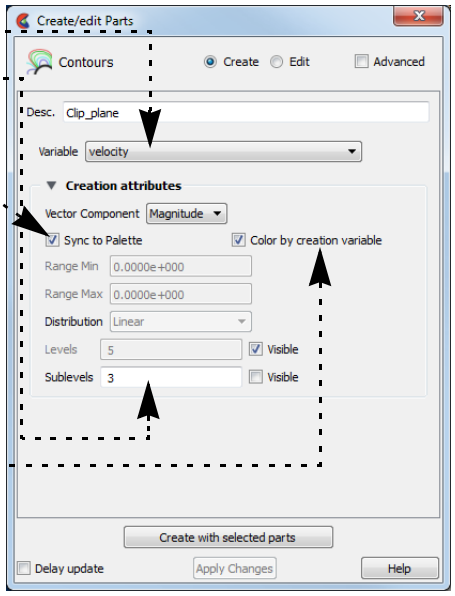
The clip plane was built using the external flow field as the parent part. Since the clip plane is itself a part, it can be used as a parent part to create other parts. To create contour loops of velocity on the clip plane, for example.

The Clip plane was created using right click capability. We could to the same for the contours, i.e., right click on the Clip_plane part and use the Contour function in the pulldown list. But here we build the part through the user interface so you can see how this is done.

19. Select the "Clip_plane" part in the Parts list.
20. Click the Contour icon in the Feature Icon bar to open the Feature Panel for Contours.



21. Select Velocity in the variable list.
22. Change the value of the Sublevels field to 3.
23. Click "Create with selected parts" to build the new contour part.



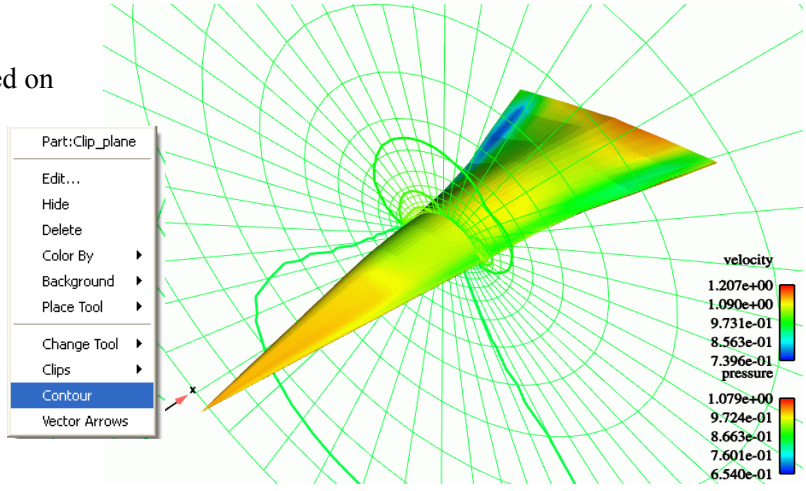
By default the Sync To Palette toggle is activated so that EnSight creates a contour loop at each *level* in the color palette assigned to the selected variable. Levels are evenly spaced values that span from the minimum to the maximum range of the variable. Default color palettes have five levels. The subcontour value sets the number of additional contour loops that will be calculated *between* each level. In this case, 17 total contours will be calculated: ((5–1)*3)+5.

If the preference called "Color by creation variable" toggle is set the contour loops are colored by velocity. If this toggle is not set, the contours will appeared as white lines initially (and can be colored later if desired). We can change the line width of the contours to make them a little more visible:

24. Click the Part Line Width pulldown from the Quick Action Icon bar and set the width to 3 pixels.

Your Graphics Window should look something like:

Alternatively, you could have right-clicked on the clip plane part and chosen contour.

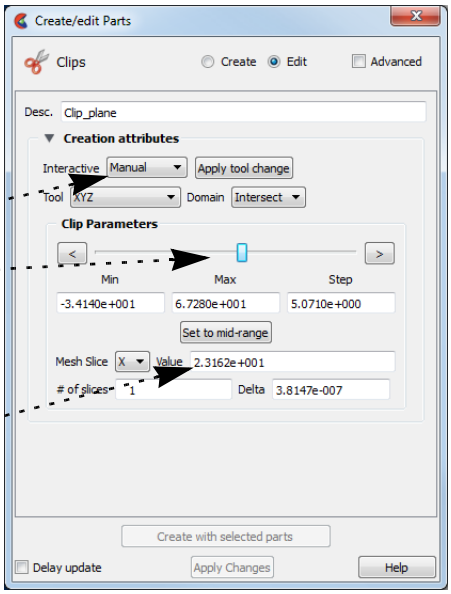


See *How To Create Contours* (Help > How To Manual ...) for more information on contours and how color palettes affect contours.

Clip parts in EnSight can be interactive: the tool that created the part can be grabbed with the mouse and moved. Or in cases like our X plane, a slider is provided. Once you release the mouse button, any parts that depend on the clip (*i.e.* that have the clip part as a parent), are automatically recalculated to reflect the new condition of the parent.

25. Double-click the "Clip_plane" part in the Parts list.
    (Or right click and Edit.)

Double-clicking a created part opens the Feature Panel and places it into Edit mode. **Any changes in the Feature Panel will affect only the parts marked with a pencil in the Parts list.**

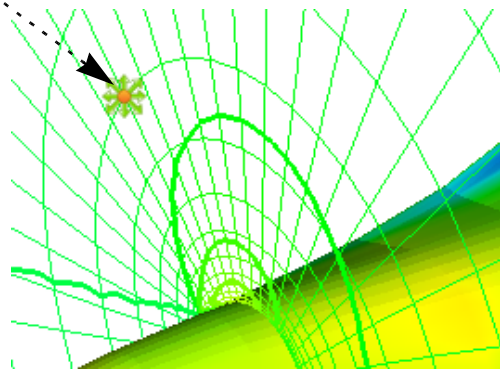


26. Click the Interactive pulldown and select Manual.
27. Manipulate the slider.

As the slider moves, the X plane is translated.

When the slider is released, the contour part recalculates to reflect the ending location of the clip plane. You can also type precise locations into the X Value field (and press Enter) if desired. You can also click the slider arrow buttons, which stride by the increment value.

28. Alternatively, you can left click on the clip plane until you get a handle and then drag the clip plane using the left mouse button on the handle.

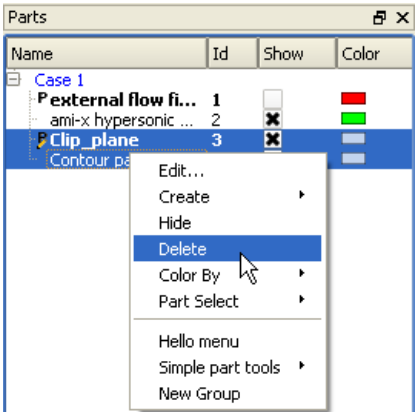


29. If the Feature Panel is no longer showing the "Clip_plane" part, follow step 22 to redisplay the part's Feature Panel, then click the Interactive pulldown again and select Off to disable interactive operation.

Any kind of part (model or created) can be deleted. Note that a deletion cannot be undone.

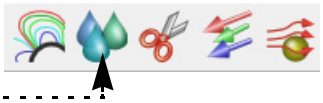
30. Select all the **created** parts in the Parts list: place the mouse pointer over part 3, click the left mouse button then Shift click on part 4 so it is selected as well.
31. Right click and "Delete" (or move the mouse into the graphics window and press the delete key, or choose Main Menu > Edit > Part > Delete...) and confirm the deletion if prompted.

The parts are removed from the display and the Parts list.



Isosurface Part

Another type of created part is an *isosurface*. An isosurface is a surface of constant value (the *isovalue*) in a 3D field. The region on one side of the isosurface has values greater than the isovalue and the region on the other side has values less than the isovalue. To create an isosurface:

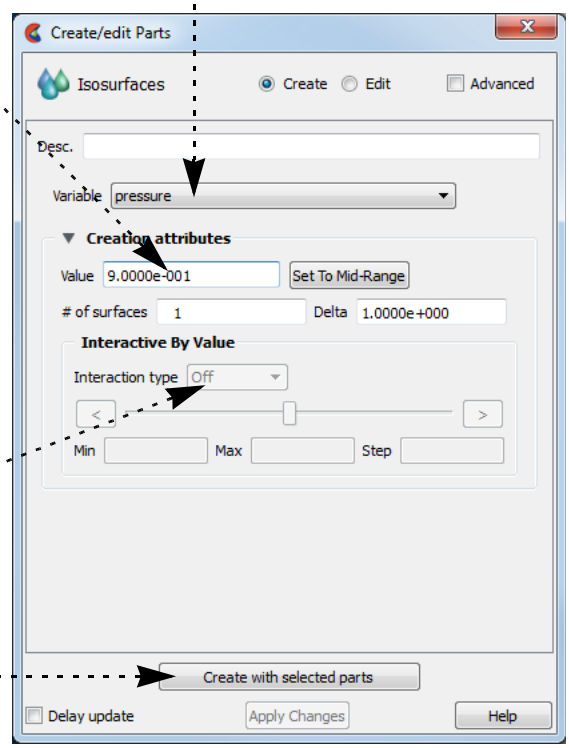


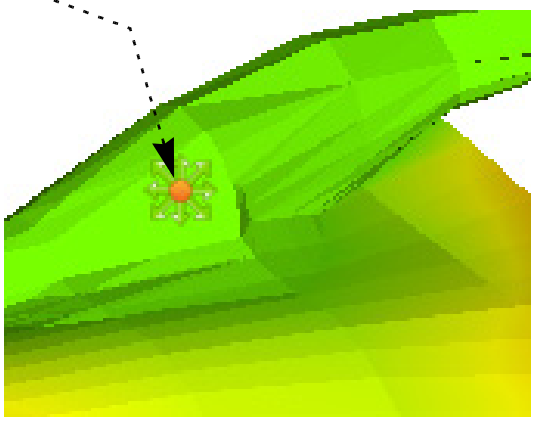
32. Select the “external flow field” part in the Parts list.
33. Click the Isosurface icon in the Feature Icon bar to open the Feature Panel.
34. Select pressure in the variables pulldown.
35. Triple-click the Value field to select the “MID-RANGE” value. Type “0.9”
36. Click “Create with selected parts” to build the isosurface part.
37. Color the new isosurface part by pressure by right clicking on the part (in the graphics window or in the Parts list) to “Color By->Select Variables” and choosing pressure in the resulting pop-up dialog.
38. Isosurfaces can be interactive. Manipulating a slider changes the isovalue and the isosurface is recalculated and redisplayed.
39. Change the Interactive setting from Off to Manual.
40. Grab the slider and move it left and right.

The isosurface changes as the slider is moved. Or better, left-click on the isosurface to see a part handle. Left click and drag to change the isovalue.

41. Turn the Interactive mode Off, and delete the isosurface part.

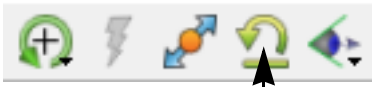
See *How To Create Isosurfaces* (Help > How To Manual ...) for more information.





Particle Trace Part EnSight provides particularly powerful tools for exploring flow with particle traces. Traces can be emitted from a point, a line, a plane, or even the nodes of an arbitrary part. A trace emitter can be made interactive: moving the emitter with the mouse will recalculate and re-display the traces. In this example, a simple point trace will be created. As before, the operation can be performed through the user interface or via right click. We will use right click.

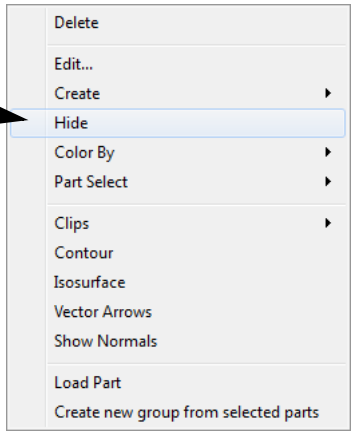
42. Click the Reinitialize Transforms icon to reset the view back to the default.



43. Click the Cursor tool toggle.

The Cursor tool is used to specify the position of a 3D point and, in this case, will be used to set the location of the particle trace emitter. Unfortunately, the Cursor may not be visible and may be initialized *inside* the hypersonic body part. The part needs to be made temporarily invisible so the Cursor can be moved.

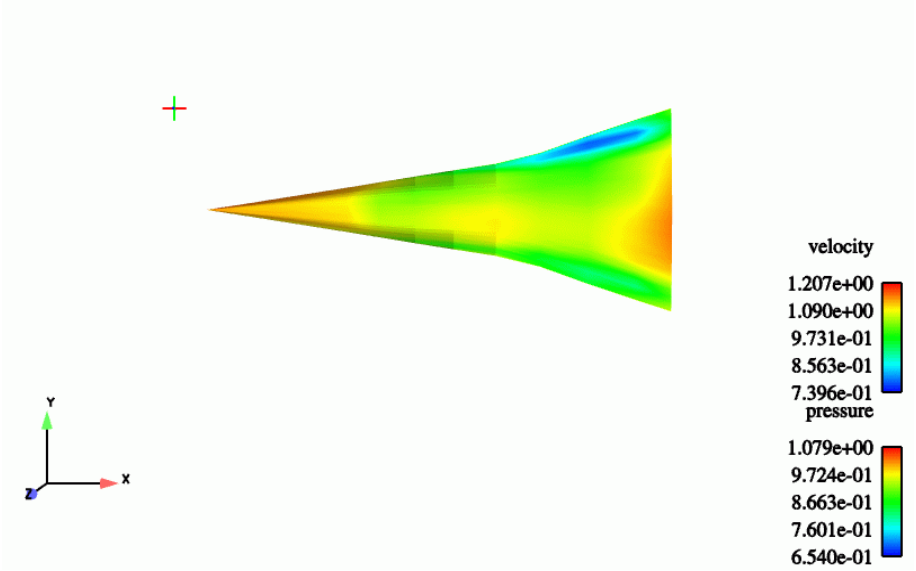
44. Right click on the “hypersonic body” part in the Parts list to Hide.



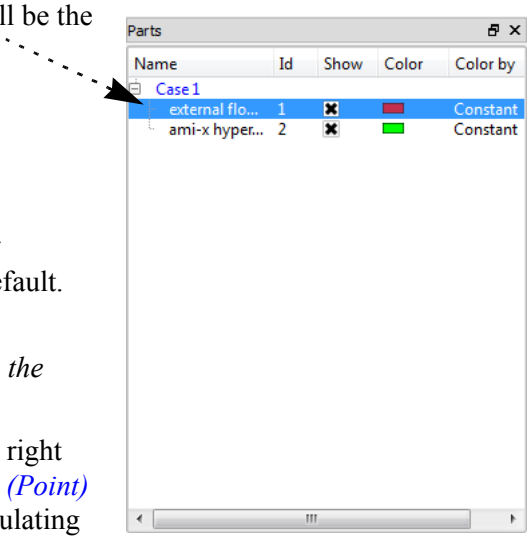
You should now see the Cursor tool (the red, green, blue cross) in the center of the screen.

45. Move the mouse pointer into the Graphics Window and directly on top of the center of Cursor tool. *Note: The mouse cursor will change to a “+” shape when over the tool.*
46. Click and hold the left mouse button and drag the Cursor to a location up and to the left (see image below).
47. Turn the visibility of the “hypersonic body” back on by again right clicking on it (this time it can only be done in the Parts list) and select “Show”.

Your Graphics Window should look something like the following:



48. Select the “external flow field” part in the Parts list. This will be the part (“parent part”) we trace the particle through.



49. Click the Particle Traces icon in the Feature Icon bar to open the Feature Panel.
50. Select velocity in the variable pulldown in the Feature Panel.

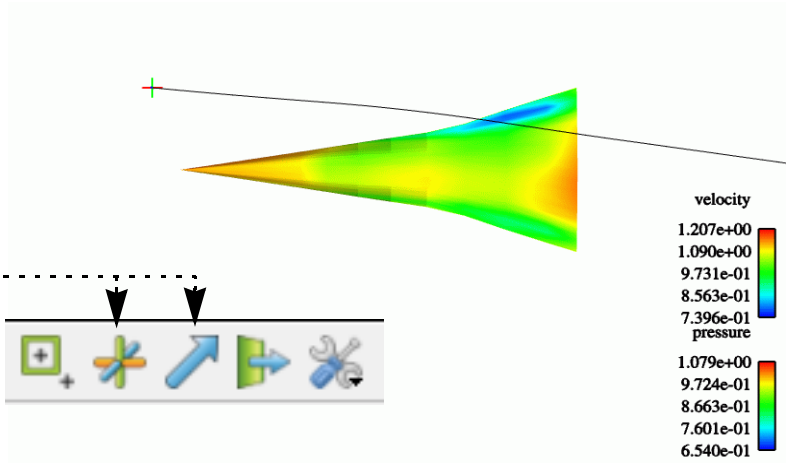
The Emit From setting in the Feature Panel is set to Cursor by default.

51. Click Create with selected part, to trace the particle.
Note that this could have also been done by right clicking on the cursor tool.

The trace should be visible extending from the Cursor tool to the right and down over the hypersonic body. See *How To Use the Cursor (Point) Tool* (Help > How To Manual ...) for more information on manipulating the Cursor tool.

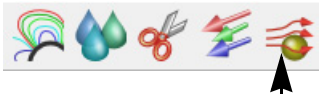
EnSight can also trace from the Line tool to create a rake of particles. This can also be done via the right click capability but we will do it here by following the user interface.

52. Right click on the particle trace part in the Parts list and delete it.
53. Click the Cursor toggle on the desktop to disable display of the Cursor tool and click on the Line toggle to turn the line tool visible.



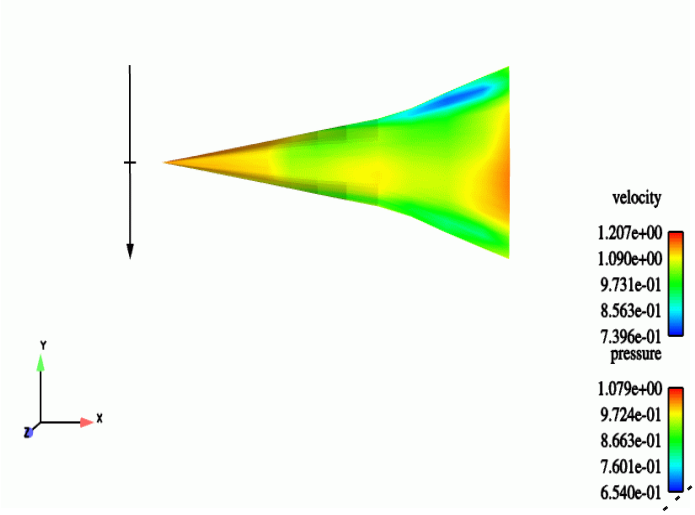
This selection displays the Line tool which is also completely enclosed within the hypersonic body part.

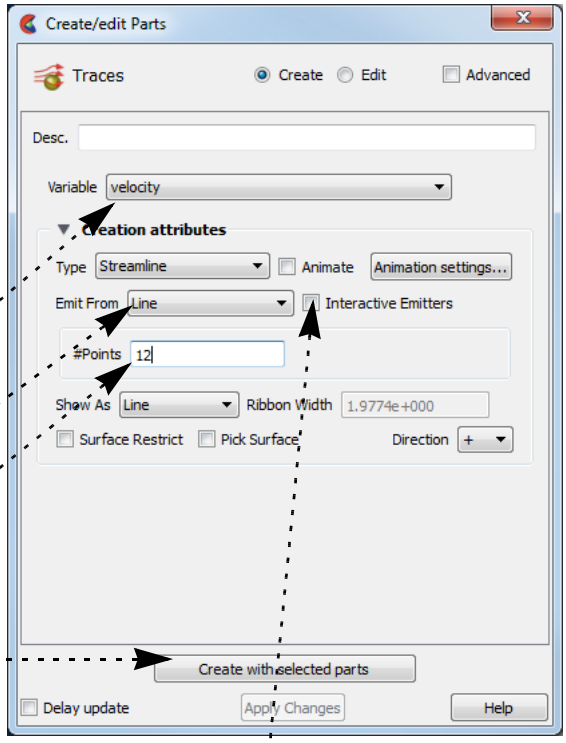
54. Right-click the on the hypersonic body part and choose Hide to disable part display. The Line tool (oriented horizontally) should now be visible.
55. Move the mouse pointer into the Graphics Window and directly on top of the center of Line tool. (The mouse cursor will change to a “+” when over a tool hotpoint.)
56. Click and hold the left mouse button and drag the Line to a location up and to the left.
57. Click the Part Visibility Toggle to re-enable display of the hypersonic body part.
58. Select the “external flow field” part in the Parts list.
59. Click on the Particle Trace icon in the Feature bar to bring up the Particle trace Feature Panel.



60. Move the mouse pointer back into the Graphics Window and directly over the right end of the Line tool.
61. Click and drag the end of the Line tool down and to the left such that the Line is vertically stretched across the front of the hypersonic body (see image below).

Your Graphics Window should look something like the following:





62. Select velocity from the variable pulldown.
63. Select Line to Emit From
64. Change the # Points field to 12.
65. Make sure that the external flow field is selected in the Parts list as the parent part.
66. Click “Create with selected parts”

This operation created 12 evenly spaced traces along the Line tool.

Interactive particle traces are particularly useful for exploring fluid flow.

67. In order to make the traces interactive you must click Interactive Emitters.
68. Move the mouse pointer into the Graphics Window and directly over the center of the Line tool.
69. Click and hold the left mouse button and drag the Line tool up and down. Release the mouse button.

The constituent traces are recalculated and re-displayed as the Line tool moves. It may help to rotate the model to a new orientation and then move the Line tool again.

70. Click Interactive Emitters again to disable interactive operation.

See *How To Create Particle Traces* (Help > How To Manual ...) for more information on particle tracing.

Particle traces can be animated to provide intuitive comprehension of flow characteristics. Traces are animated by displaying one or more *tracers* on all traces of the trace part. A tracer moves along the path of a trace with length proportional to the local velocity. EnSight provides complete control over all aspects of the tracers including length, speed, and release interval for multiple pulses.

71. Toggle Animate.

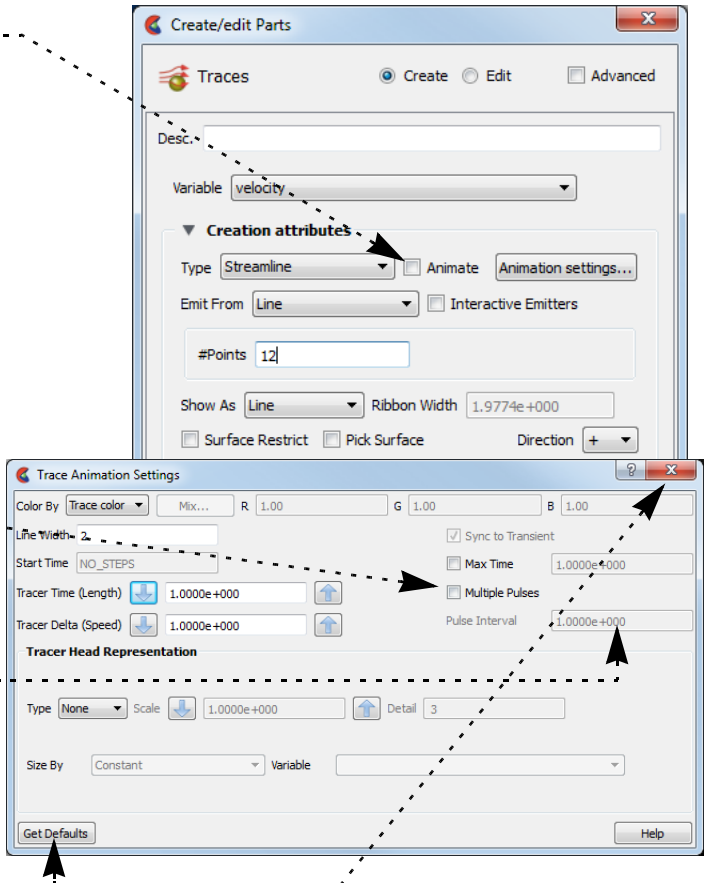
The tracers can be seen moving down the length of the traces. Numerous controls are provided for altering the appearance and behavior of the tracers.

It may be beneficial to turn off Highlighting - see step 2.

72. Click Animate settings... to open the Trace Animation Settings dialog.
73. Click Get Defaults to load suitable default values to the various trace parameter fields.
74. Click Multiple Pulses.

Note that there are now several tracers are moving down each trace. The Pulse Interval field controls the spacing between tracers.

75. Triple-click in the Pulse Interval field to select the value. Type “10” and press Enter.
76. You can lengthen or shorten the tracer via the Tracer Time field. Either enter a new value (like 1.0) or use the up/down arrows to modify the value.
77. The speed of the tracers is controlled by the Tracer Delta field. Enter a val (try 0.1 for example) or use the up/down arrows.
78. Click the X to Close the dialog.



Animating traces often look better with the trace part invisible.

79. Right click on the particle trace part and select Hide in the pulldown.
80. Click the Line Tool icon to disable the display of the Line tool.
81. Quit Ensight (File > Quit > OK)

See *How To Animate Particle Traces* (Help > How To Manual ...) for more information on trace animation.

4.3 Where's the Rest?

After successfully completing this tutorial, if you wish to work through an example utilizing a structured mesh, you should proceed to the next chapter. Otherwise, you might prefer to jump to Chapter 6 which presents a tutorial containing some material applicable to all users, but intended primarily for analysts doing structural mechanics (finite element) analysis.

The How To Manual contains details on the operations covered in this chapter. The following online articles are relevant to the topics covered here.

Note: you get to these by selecting Help from the main menu, then How To Manual... (Help > How To Manual...), then the topic indicated below:

Consult...	For More Information On...
How To Read Data	specifying data to read into EnSight
How To Change the Visual Representation	element representations
How To Create Color Legends	color legends and the variable to color mapping
How To Create Contours	creating contours
How To Create Particle Traces	creating particle traces
How To Animate Particle Traces	animating particle traces

5 Flow Visualization Example: Structured Mesh

This chapter is intended for users of structured mesh CFD software and those using the PLOT3D format to import data into EnSight. In addition, this chapter describes the use of the predefined CFD functions to compute variables derived from the fluid flow analysis. It is assumed that you have already worked through the features described in Chapter 4, flow visualization example for an unstructured mesh. After successfully completing this chapter, you should be able to:

- load a PLOT3D dataset
- create a part from a logical plane of the volume mesh
- create an IJK clipping plane
- use the predefined CFD functions
- save an archive

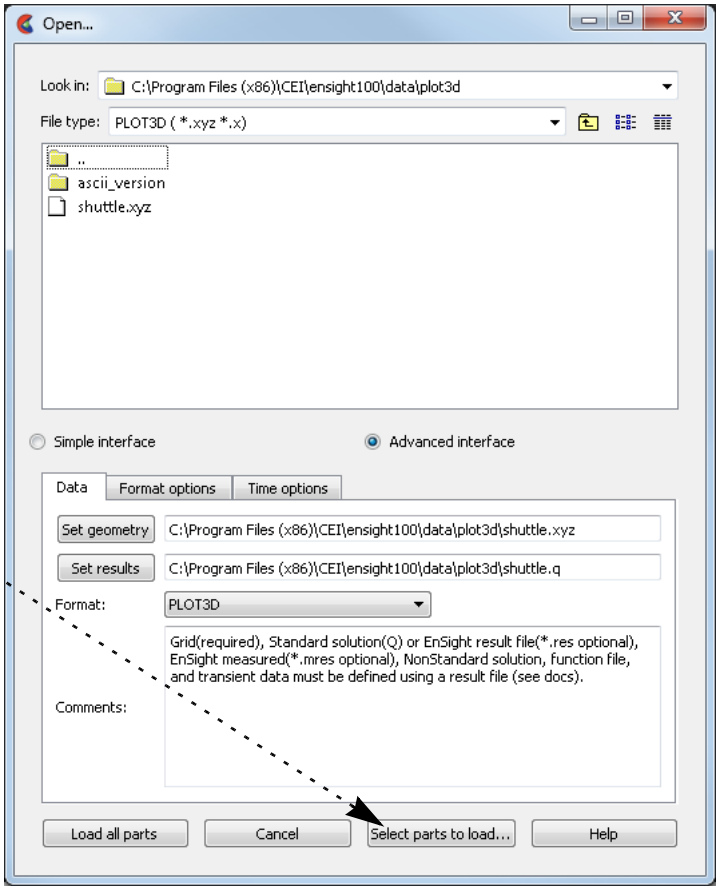
5.1 Reading a Dataset

In this demonstration, we will load a simple CFD model of the viscid, subsonic symmetric flow around a half-model of the shuttle orbiter. The dataset consists of the standard xyz (mesh) and q (results) PLOT3D format files.

1. Start EnSight as described in Chapter 2 and cancel the Welcome Screen.
2. Select File > Open... from the Main EnSight menu.

This opens the File Selection dialog.

3. Navigate to the `$CEI_HOME/ensight100/data/plot3d` directory

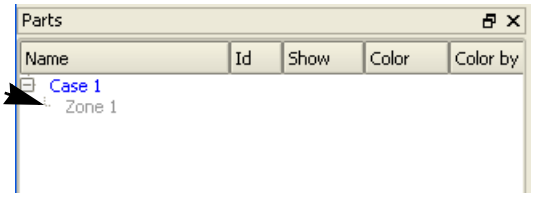


4. Click `shuttle.xyz` in the Files list.
5. Click “Select parts to load...” to accept the selections that appear in the File and results fields and to close the dialog window.

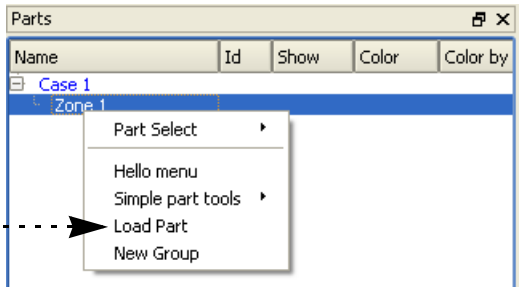
When the File Selection dialog is closed, EnSight reads the indicated file but does not load any of the geometry.

A number of different PLOT3D formats are supported by EnSight. For a complete description, see *How To Read Data* (Help > How To Manual ...) or *PLOT3D Reader* in Chapter 2 of the User Manual (Help > User Manual ...). EnSight scans the PLOT3D files to determine which format is being used.

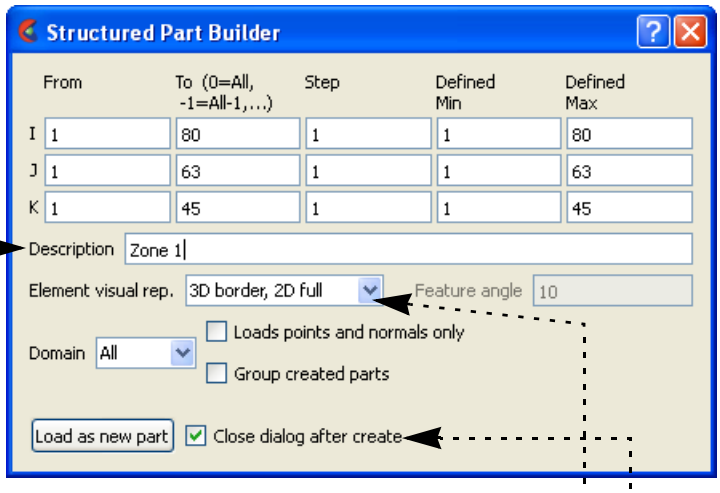
After the file has been read the Parts list will show “Zone 1” as a grayed out entry in the list indicating that the zone is available but not currently loaded.



6. Right click “Zone 1” in the Parts List and select “Load Part” from the pulldown to bring up the Structured Part Builder dialog.



This dialog allows us to create parts from a structured block. We can extract out any range of I, J, K from the block and we can extract out multiple parts from the same block.



The most basic part is the fluid flow region, in this case 8-noded hexahedral cells surrounding the surface of the shuttle geometry. For reference, this single block will be named by typing in a Part Description before creating the part. A second part, defined by the surface of the geometry, will be created by choosing an appropriate limited range of nodes (which are normally known by the author of the PLOT3D mesh). In this manner any number of surface parts may be created in addition to the 3D (fluid) parts. Note that the Data Part Loader may be used at any time to create new parts from the original PLOT3D data files.

7. Click Element Visual Rep. and change the setting to "Feature Angle".

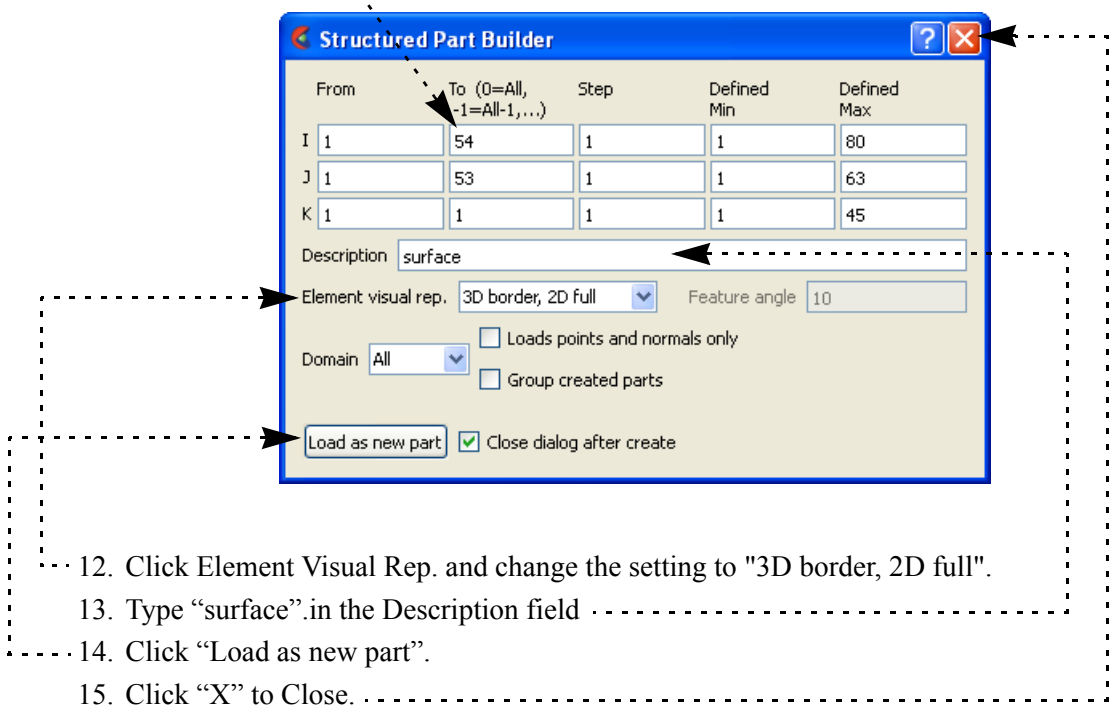
8. Type “external flow field” in the Description field.

9. Uncheck the “Close dialog after create” button - this will allow us to create two parts from this block

10. Click “Load as new part” button.

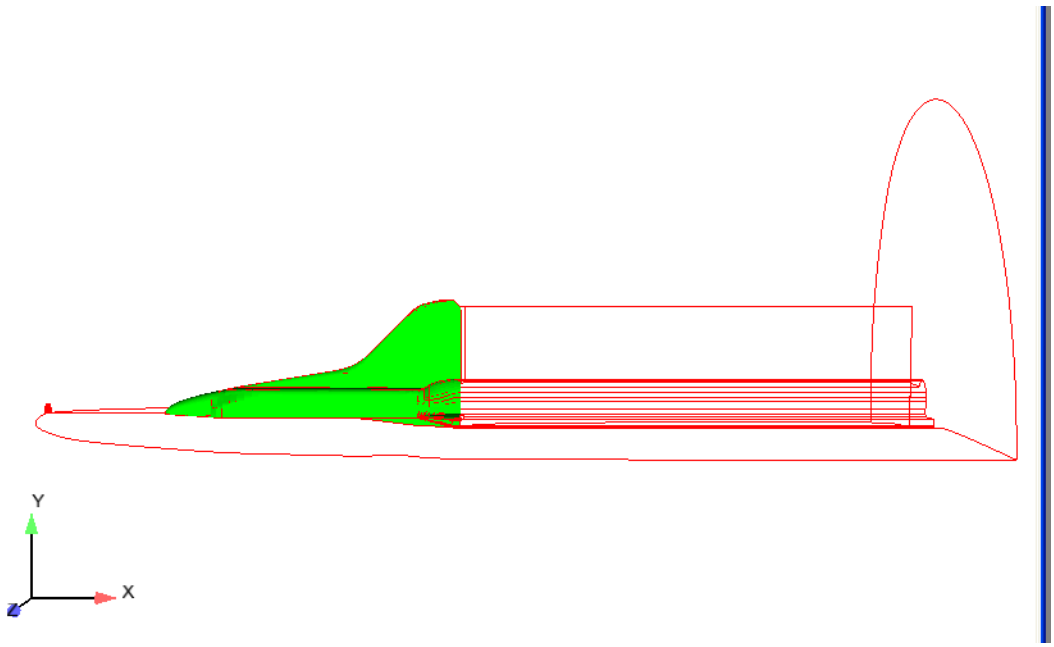
At this point the external flow field part will show up in the Parts list. The graphics window will not display the new part until the dialog is closed.

11. Change I indices to 1-54, the J indices to 1-53 and change the K indices to 1-1 (the geometry's surface).



12. Click Element Visual Rep. and change the setting to "3D border, 2D full".
13. Type “surface”.in the Description field
14. Click “Load as new part”.
15. Click “X” to Close.

The second part now appears in the Parts list and both parts will appear in the graphics window. The external flow field is displayed as a wire frame and the surface is displayed as a mesh. As the flow field and geometry are both symmetric, only a half-model was used. The image in your Graphics window should appear as follows:



5.2 Feature Demonstration

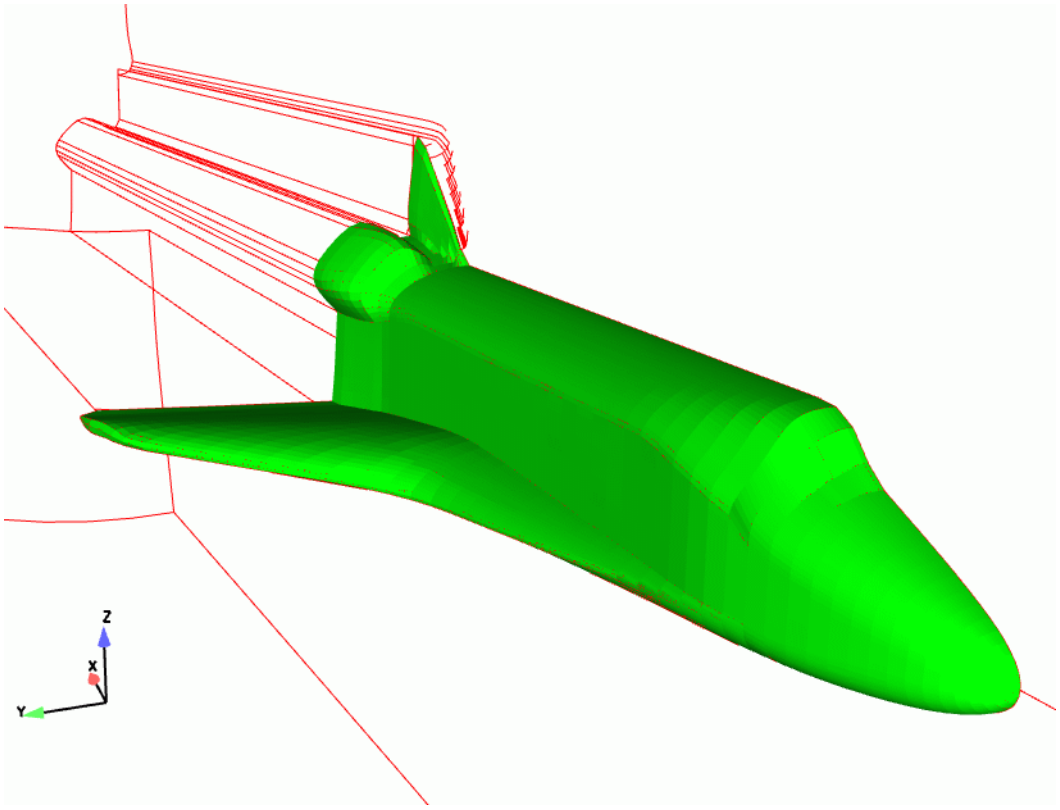
Unlike the case of unstructured meshes as in the previous example, logical planes of a structured mesh may be viewed in addition to the arbitrary planes described using the plane tool. In this demonstration, constant I, J, and K planes will be created and used to display results.

Remember that model reads in with shaded surfaces toggled ON:

1. Use the Shaded toggle on the Tools Icon bar to toggle shaded surface OFF then back ON for all parts.
2. Toggle OFF Highlight Selected Parts

After some rotations, translations, and a zoom - you should be able to produce an image approximately as follows:

(The model should be green)

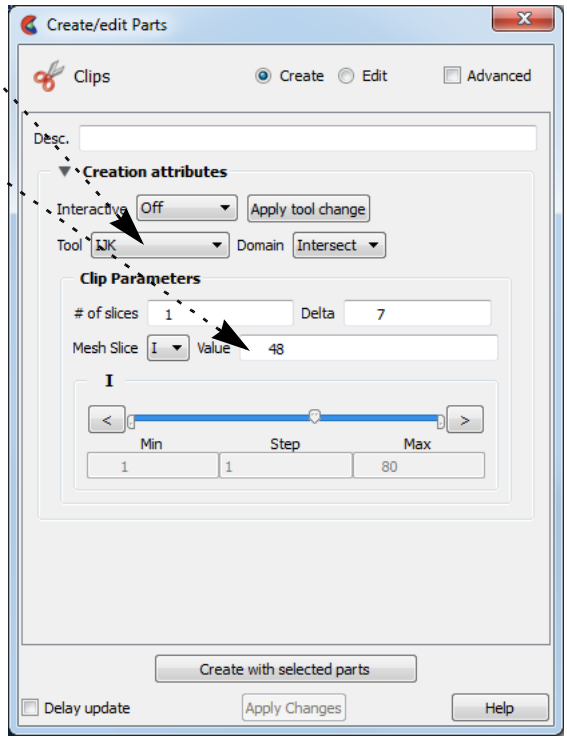


16. Select the “external flow field” part in the Parts list.
17. Click the Clip icon in the Feature Icon bar to open the Feature Panel for clip creation.

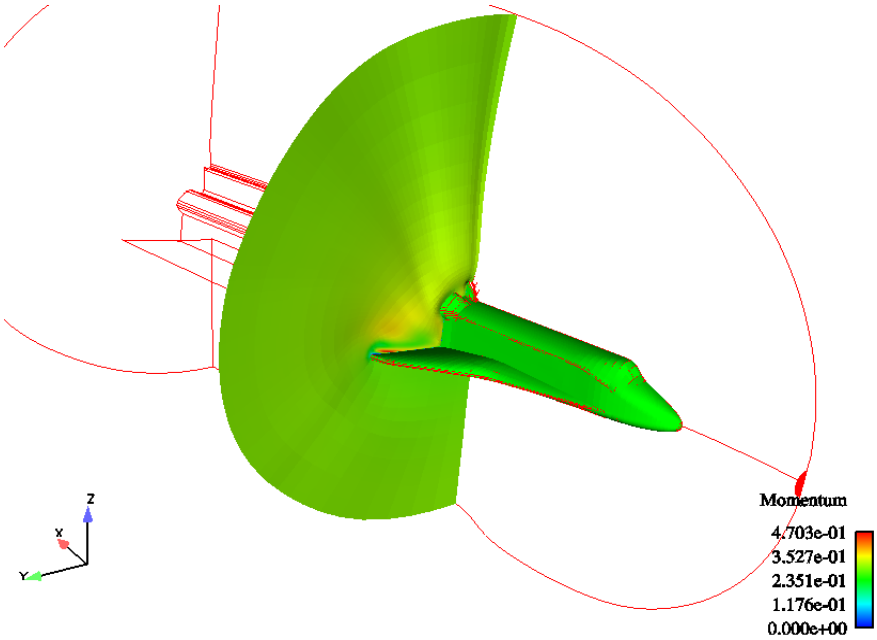
3. Click the Tool button and change the setting to IJK.
4. Change the value for an I Mesh Slice to 48.
5. Click “Create with selected parts”.

Notice that a new part appears in the Parts list, "Sweep surface"

6. Right click on the “Sweep surface” part to bring up the right click pulldown. From the pulldown select “Color By->Select Variable” and in the resulting pop-up dialog choose Momentum (under Vectors) followed by the OK button.



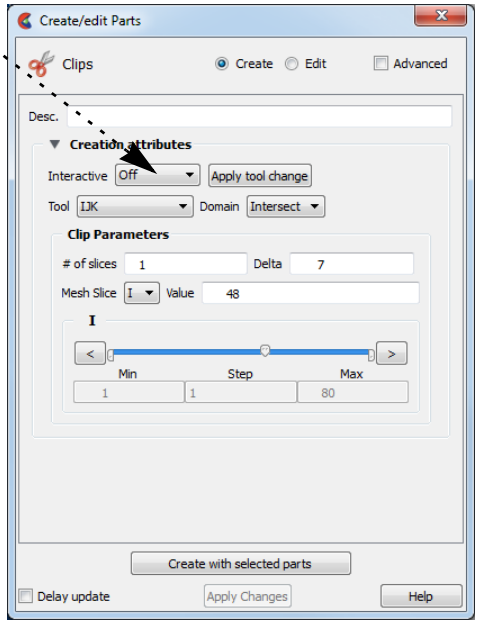
The image in your graphics window should appear as follows:



As in the previous example, it is possible to interactively manipulate the clip plane. In this case, however, the plane will move along a given logical coordinate as the slide bar is moved:

7. Select “Manual” from the Interactive menu.
8. Move the slider to view different constant I Mesh planes
9. Change the Mesh Slice from "I" to "J" and "K" and use the slider to view different constant J and constant K mesh planes. Note that the Min, Max and Step settings can be used to limit the extent and resolution of these planes.
10. Set the Mesh Slice back to “I”.
11. Set the Interactive setting back to OFF.

It is possible to use both IJK and arbitrary plane Clips within the same session of EnSight. The IJK Clip feature may also be used as an alternative to creating model parts via the Part Builder as described above.

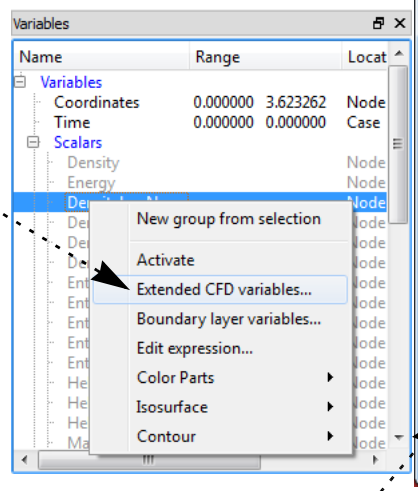


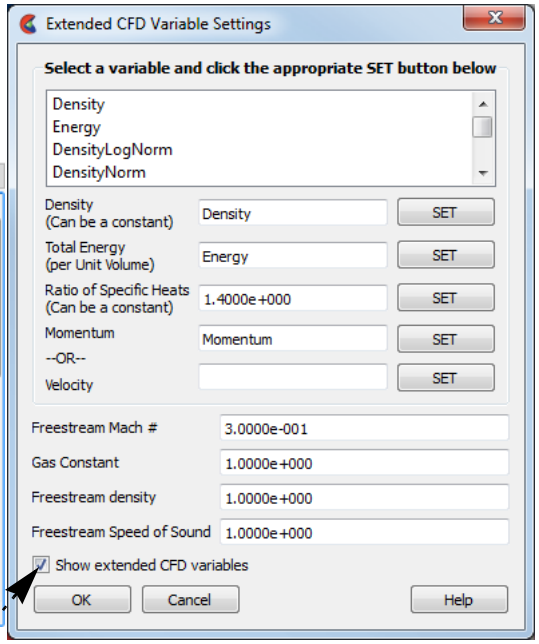
Extended CFD Variables

Often, the primitive variables solved by a given CFD code are of less interest than certain derived variables, for instance the Mach number, defined as the ratio of the local fluid speed to the freestream speed of sound. In the aerospace community, a number of these derived quantities are referred to as the "PLOT3D functions", as they were available in the program PLOT3D. EnSight includes most of these functions, and enhances their original form in two important ways. First, these Extended CFD Variables may be computed based on the "q file" variables of the PLOT3D format, namely the scalars density and energy and the momentum vector. In this case, EnSight makes the appropriate mapping of the variables. If, however, PLOT3D files are not used, or if the PLOT3D variables are used in a non-standard fashion, the user may define the appropriate mapping between the variable names as they were defined in the results and those quantities needed by EnSight to create the desired extended variable. The second enhancement is the ability to define any value for the ratio of specific heats, Cp, or use a scalar variable to define a different Cp value at each node of the mesh.

The Extended CFD variables are turned on by default when you read a Plot3d file. To edit the settings or to turn them off:

12. Right click on any variable in the Variable list.
13. And select “Extended CFD Variables...” from the pulldown which opens the Extended CFD Variable Settings dialog





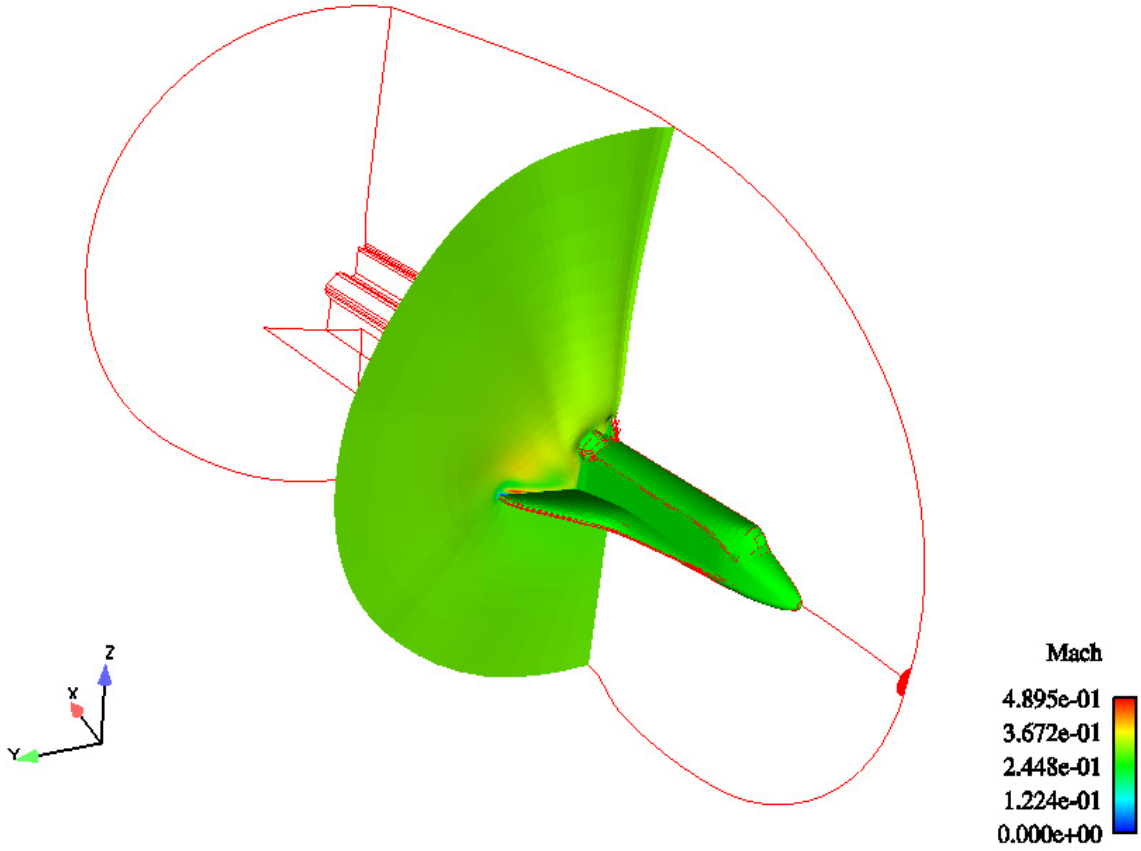
Normally this is where you would adjust any needed parameters or turn the extended CFD variables off. Things are good for this model.

14. Click OK.

With the Extended CFD Variables turned on the Variable list contains the original variables as well as the Extended CFD Variables. As with all others, these variables will not be activated until they are needed, for instance to color a model part. The activation of the extended variables involve a computation requiring one or more of the primitive variables. This, in turn, will require their activation, which EnSight performs automatically. All activated variables will remain so until they are explicitly deactivated. Note also that some extended variables (for instance vorticity) involve complex computations which may require significant time to complete. Variables that are grayed out in the list have not yet been activated (or computed). Solid black colored variables are active.

15. Right click on the “Sweep surface” part in the Parts list and color the part by Mach number.

The image in your Graphics Window should look something like the following:



See *How To Create New Variables* (Help > How To Manual ...) for more information.

Saving an Archive

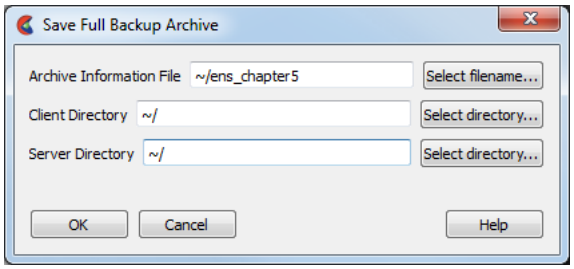
EnSight can save the complete state of a session as an *archive*. An archive consists of two binary files containing the state of the Client and Server as well as an "Archive Information file" that stores additional information (including pointers to the two binary files).

Although you can duplicate a session by replaying a saved command file, restoring an archive is much faster. When you replay a command file, EnSight has to re-execute every action performed by the user, even if that action had no effect on the final state. An archive restores quickly since only the final state is restored.

To save an archive:

1. Select File > Save > Full Backup... to open the Save Full Backup Archive dialog.

By default, EnSight will save the archive information file and the Client archive in the directory from which the Client is started. The Server archive will be saved on the Server host (by default, in the directory from which the Server was started). Since these directories are part of the EnSight distribution, they are probably write protected. To save the archive files in your home directory instead, prefix the entries with "~/".



2. Triple-click in the Archive Information File field and type "~/ens_chapter5.ar".
3. Double-click in the Client Directory field and type "~/".
4. Double-click in the Server Directory field and type "~/".
5. Click OK.

You can restore an archive during a session (either immediately after the Client-Server connection or after replacing a case) by selecting File > Restore > Full Backup.... You can also have an archive automatically load at startup. From a command line:

```
% ensight100 -ar ~/ens_chapter5.ar
```

And even easier - drag and drop the .ar file from a file browser onto the ensight100 desktop icon. See *How To Save or Restore an Archive* (Help > How To Manual ...) for more information.

5.3 Where's the Rest?

After successfully completing this tutorial, you should proceed to the next demonstration. Although the next tutorial contains some material applicable to all users, it is intended primarily for analysts doing structural mechanics (finite element) analysis.

The How To Manual contains details on the operations covered in the chapter. The following online articles are relevant to the topics covered here.

Note: you get to these by selecting Help from the main menu, then How To Manual... (Help > How To Manual...), then the topic indicated below:.

Consult...	For More Information On...
How To Read Data	specifying PLOT3D format results data to read into EnSight
How To Create IJK Clips	creating clip surfaces in structured models
How To Create New Variables	creating extended CFD variables
How To Save or Restore an Archive	saving and restoring archives

6 Structural Mechanics Example

This chapter provides step-by-step instructions for performing basic postprocessing operations – especially those relevant to non-linear dynamics (*e.g.* crash) analysis. Unlike the two previous datasets, the data used here is *transient* – the model variables vary over time. EnSight provides a wide range of features for postprocessing and animating transient data.

After successfully completing this chapter, you should be able to:

- group multiple parts into a single part,
- show geometry displacements,
- probe for data values,
- change time steps,
- perform a query and plot it,
- add annotation to an image,
- create a flipbook animation.

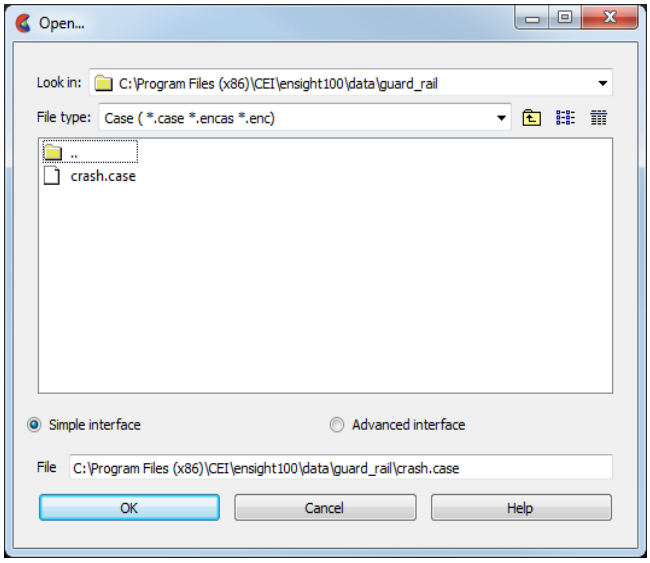
6.1 Reading a Dataset

In this demonstration, we will load a transient dataset of a car crash into a guard rail. The dataset includes displacement and plastic strain values.

1. Start EnSight as described in Chapter 2, and cancel the Welcome screen.
2. Select File > Open... from the EnSight Main menu.

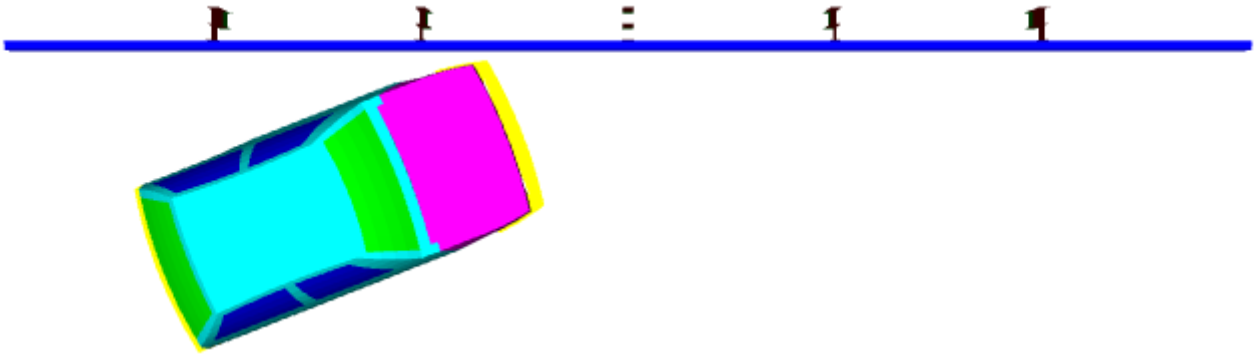
This opens the File Selection dialog.

3. Navigate to the `$CEI_HOME/ensight100/data/guard_rail` directory
4. Click `crash.case` in file list.
5. Make sure you are using the Simple Interface, and click OK to accept the selections and load all of the parts in the dataset.



This procedure (using File > Open...) is a quick read which loads all parts for file types that have been mapped (using a mapping file) to a specific reader and a specific part visualization representation. The site preferences mapping file (named `ensight_reader_extension.map`) is found in the site_preferences subdirectory in the EnSight 10.0 install directory, and the local mapping file (same name `ensight_reader_extension.map`) is found in the EnSight Defaults Directory (located at %HOMEDRIVE%%HOMEPATH%\(username)\.ensight100 commonly located at C:\Users\username\.ensight100 on Win7, C:\Documents and Settings\yourusername\.ensight100 on older Windows, and ~/.ensight100 on Linux, and in ~/Library/Application Support/EnSight100 on the Mac). The local file takes precedence over the site preference file.

The constituent parts are now listed in the Parts list and displayed in the Graphics Window:



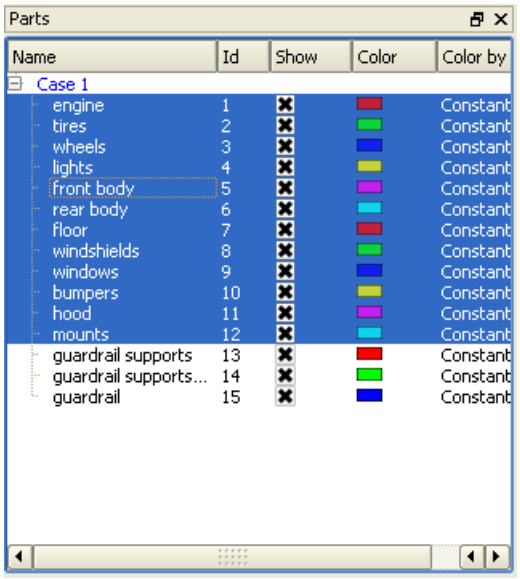
Note: *To improve the legibility of the* Getting Started *manual, all images of the Graphics Window are shown with a white background rather than the default dark background that EnSight uses.*

6.2 Feature Demonstration

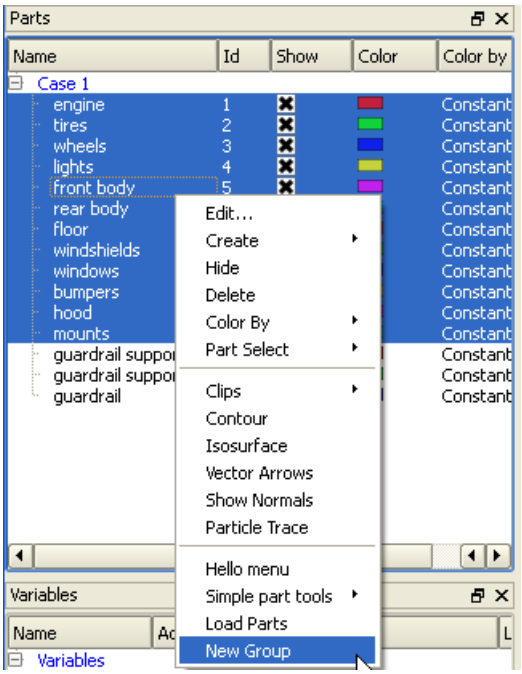
In many types of analysis, multiple parts are used to distinguish between various components or material types. To the extent allowed by the particular data format, EnSight maintains this distinction by assigning these entities to separate model parts. In some cases, however, this distinction is no longer useful for postprocessing. When manipulating objects in EnSight, you often want to apply the same attributes or operators to a group of parts. If the group is large, this process can become unwieldy. Fortunately, EnSight provides a mechanism, called *grouping*, for grouping multiple parts into a single group part. The original parts comprising the group will no longer be visible in the list.

In this example, we will group all parts associated with the car into a single part.

1. Select all the parts associated with the car: place the mouse pointer over the first part in the Parts list, click the left mouse button, then hold down the Shift button and left click on part 12. You should now have the first twelve parts selected in the list.

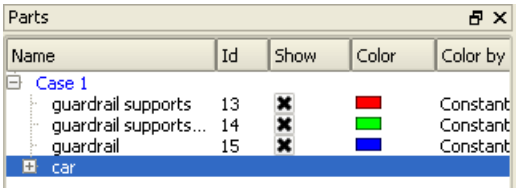


2. Right-click on the list of parts and select “New Group”.



3. In the pop-up dialog enter “car” as the group name (replacing the default name of “New group”), and click OK.
   You will now see a hierarchy in the Parts list with a group called “car” and parts 1 - 12 included in that group.

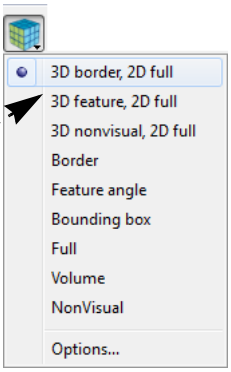
4. Click on the minus sign for the car group to hide the parts in the group.

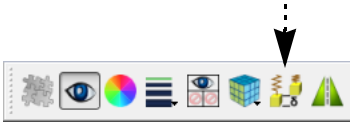


The interesting parts are now the four parts in the Parts list: three parts for the guard rail and the new group part (named “car”).

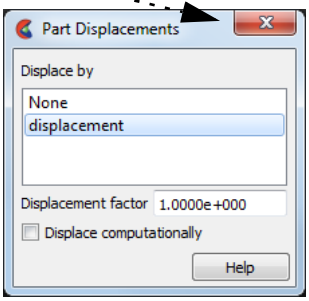
EnSight can displace geometry based on the value of a vector variable. Each displacement vector represents a translation of a node from its original position (an offset)

5. Select all of the parts in the Parts list. This can be done multiple ways (a) Click on the Case1 line in the Parts list which will select all the parts in the Case1 group, (b) Right click on any part and "Part Select -> All", (c) From the main toolbar Edit->Part->Select All.
6. Click on the Part element settings icon in the Quick Action Icon bar for parts and select 3D border, 2D full from the pulldown.
7. Click the Displacement icon in the Part Quick Action Icon bar to open a variable chooser dialog for displacement.





8. Select the displacement variable, then close the dialog.

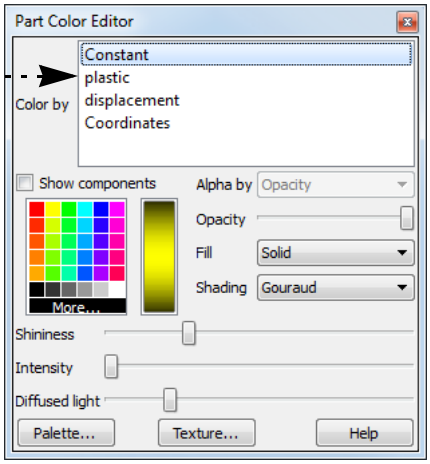


Note the new positions of the car and the guard rail. See *How To Display Displacements* (Help > How To Manual ...) for more information.

Now color the car by the plastic strain variable. Below we do this operation by using the user interface dialogs. Another way to do this would be to right click on the "car" group and then "Color By"

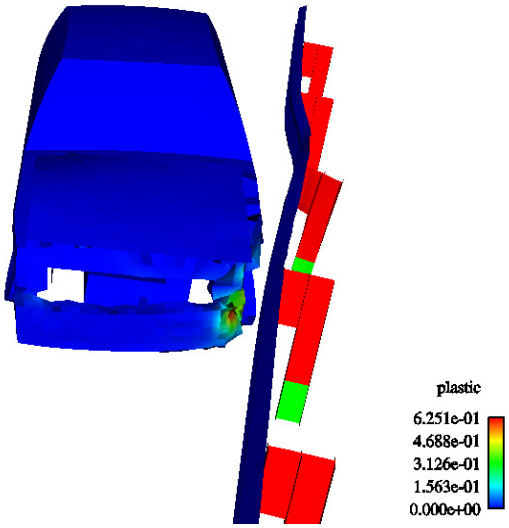
9. Select the "car" group in the Parts list.
10. Click the Color/transparency icon in the Quick Action Icon Bar to open the Part color, lighting, & transparency dialog.

11. Select the plastic variable from the 'Color by' list, then close the dialog by selecting the X in the upper right corner.



The color legend appears to the right of the model in the Graphics Window. Color legends have many display attributes – see *How To Create Color Legends* (Help > How To Manual ...) for more information.

12. Rotate and zoom the model until the view in the Graphics Window looks something like the image below:



13. Toggle OFF the Highlight Selected Parts icon

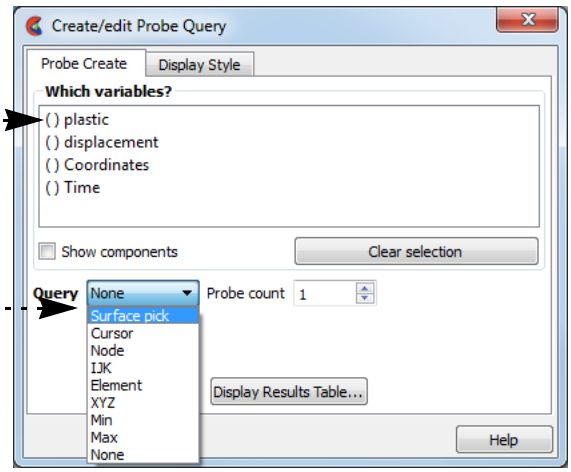
EnSight provides an interactive probe tool that uses the mouse pointer to select points of interest.

14. Click the Probe icon in the Feature Icon bar to open the Feature Panel.

15. Select the plastic variable.

16. Change the Query to Surface Pick.

17. Move the mouse pointer into the Graphics Window and place the pointer over the car (preferably in a non-blue region). Press the ‘p’ key.



The value of the plastic strain variable is calculated for the point under the mouse and displayed. A marker (the sphere) is also displayed.

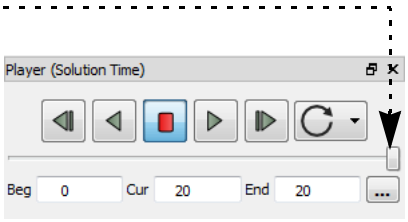
18. Change the Query back to None.

See *How To Probe Interactively* (Help > How To Manual ...) for more information.

By default, EnSight initially displays the last time step. However, it's easy to change timesteps.

The Solution Time panel provides several methods for working with time. Perhaps the easiest way to change time steps is to use the slider bar.

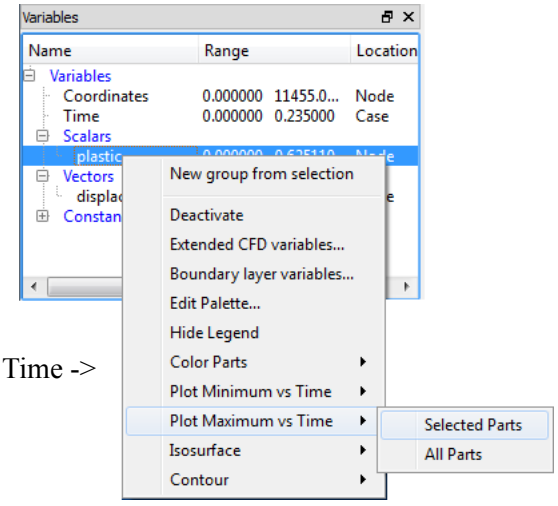
19. Place the mouse pointer over the slider bar. Click the left mouse button and drag the bar until the value in the Current field is "12".
20. Release the mouse button.



Note that the geometry in the Graphics Window has updated to reflect the data at the new time step. See *How To Change Time Steps* (Help > How To Manual ...) for more information.

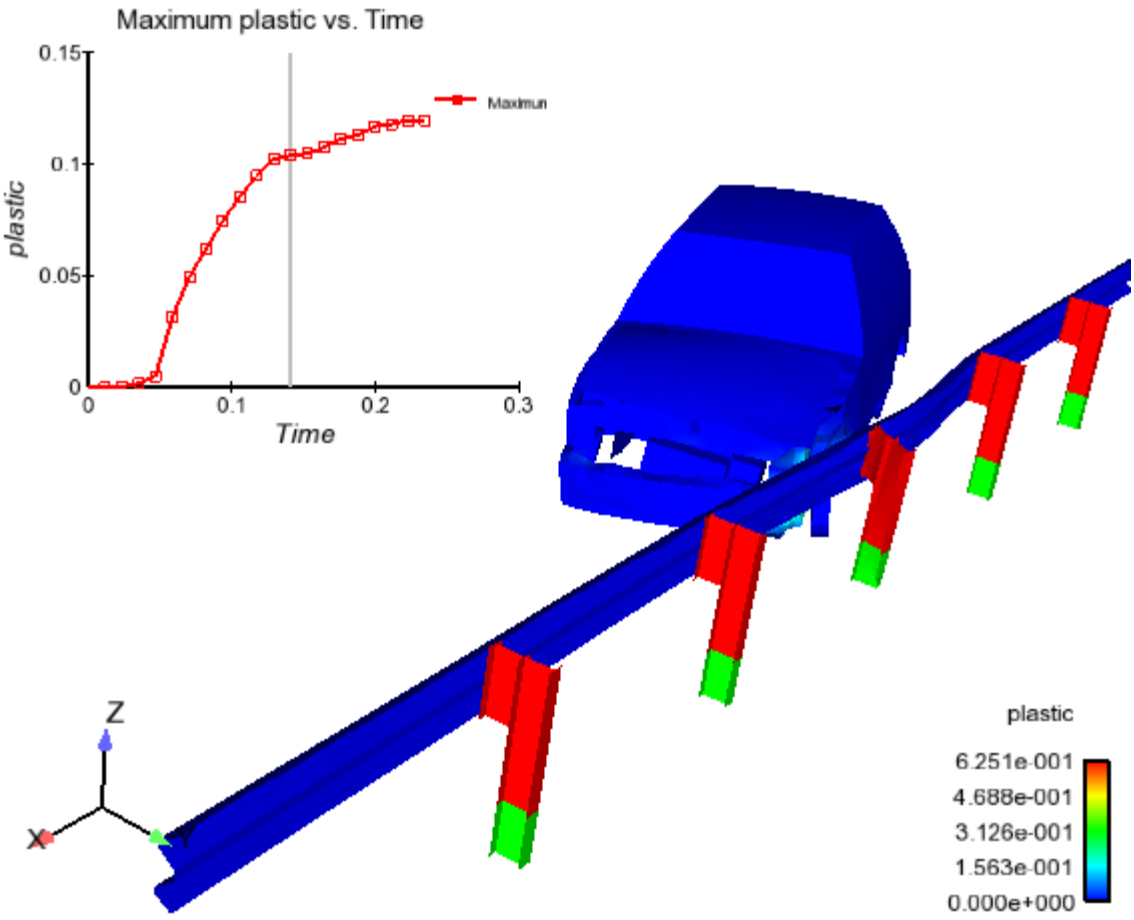
EnSight provides powerful query and plot features. Query/plot is fully integrated with the transient data handling facility so that plots will automatically update during time changes. Here we will query for the maximum plastic strain over all timesteps using right click functionality to perform the task.

21. Select the "car" group in the Parts list.
22. Right click on the plastic variable in the Variables list. You may need to open the Scalars to see it.
23. From the right click pulldown choose "Plot Maximum vs. Time -> Selected Parts".



The plot appears in the upper left corner of the Graphics Window.
See *How To Query/Plot* (Help > How To Manual ...) for more information.
And see *How To Change Plot Attributes* (Help > How To Manual ...) for more information regarding the plotter.

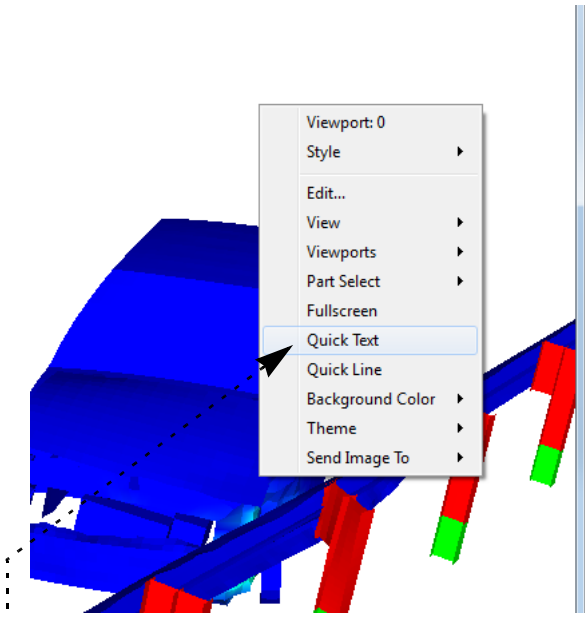
Your Graphics Window should now look something like the following:

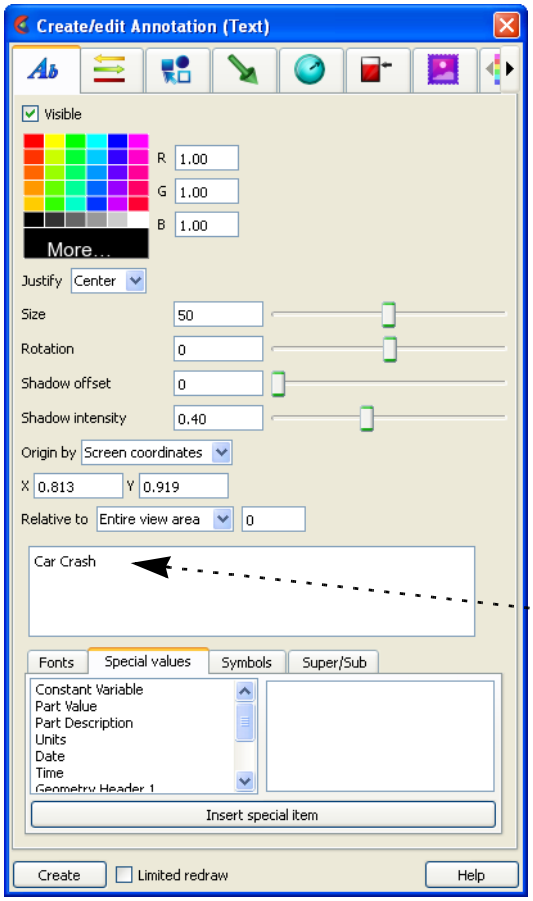


6.3 Annotation

Many postprocessing tasks require the production of hardcopy (or video) output with various types of annotation. EnSight provides comprehensive features for annotation: text, lines/arrows, color legends, and bit mapped logos.

To add text annotation:





1. Right click in the graphics window background towards the upper right of the window (the text annotation will appear where you right click).
2. From the right click pulldown select “Quick Text”. The pulldown will close and a “New Text” annotation will appear in the graphics window along with the Feature Panel.
3. Edit the “New Text” and type “Car Crash”
4. Close the dialog.

You now have a text annotation visible in the graphics window. It may not be the correct size and location. To adjust:

5. As you move the mouse pointer over the text annotation two handles will appear - a move and a resize. (Note, if they don’t appear when your mouse is over the text, click on the text to get them to appear.)
6. Click and drag the move and resize handles.

EnSight supports several “special strings” that let you automatically display constant variable values as well as various internal values. For example, you can have a text string that displays the current solution time. When the time step is changed, the text automatically updates.

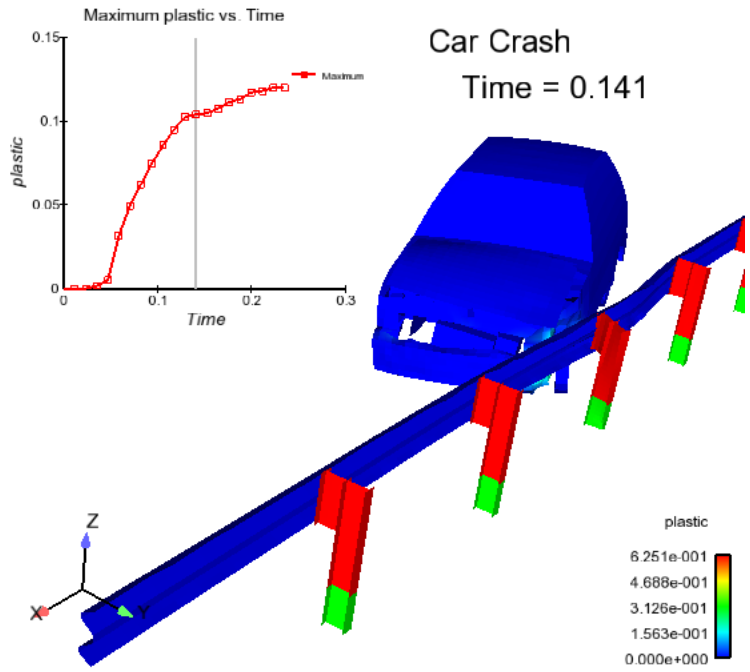
7. Left click on the Time variable in the Variables list. Keep holding the left mouse button and drag it out and drop it into the graphics window.

The text “Time = 0.141” appears in the Graphics Window.

This value corresponds to the solution time at time step 12.

8. Use the handles on the new annotation string to move it off of the plotter. Hint - since the string is on top of the plotter you may not get the pop-up handles because you are also over the plotter. So click on the annotation string and you will see the handles appear on the string.

Your Graphics Window should look something like the following:

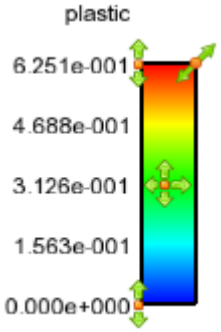


If you change time steps (as described on page 6-6) or load a new flipbook (as we will do on page 6-12) the text will automatically update to reflect the current time.

See *How To Create Text Annotation* (Help > How To Manual ...) for more information.

As you have seen, EnSight can display color legends for any variable. Legends are annotations and can be adjusted.

Color legends can be resized and repositioned. The quick way is to use the touch-and-go and right click shortcuts

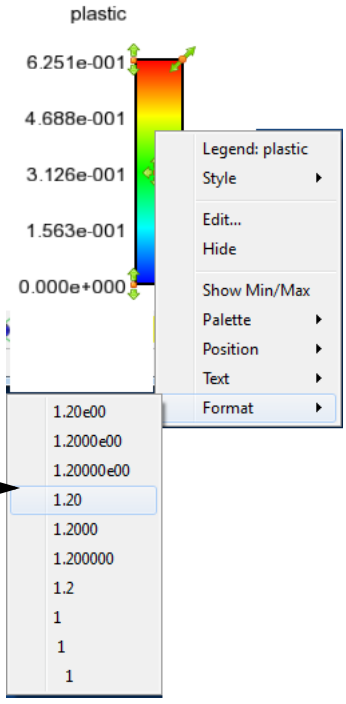


9. Move the mouse pointer to the legend. The Touch-n-go handles will appear. You can drag anywhere on the legend or you can resize by the upper right corner icon. The two left (up down arrows) icons adjust the min/max values of the palette.

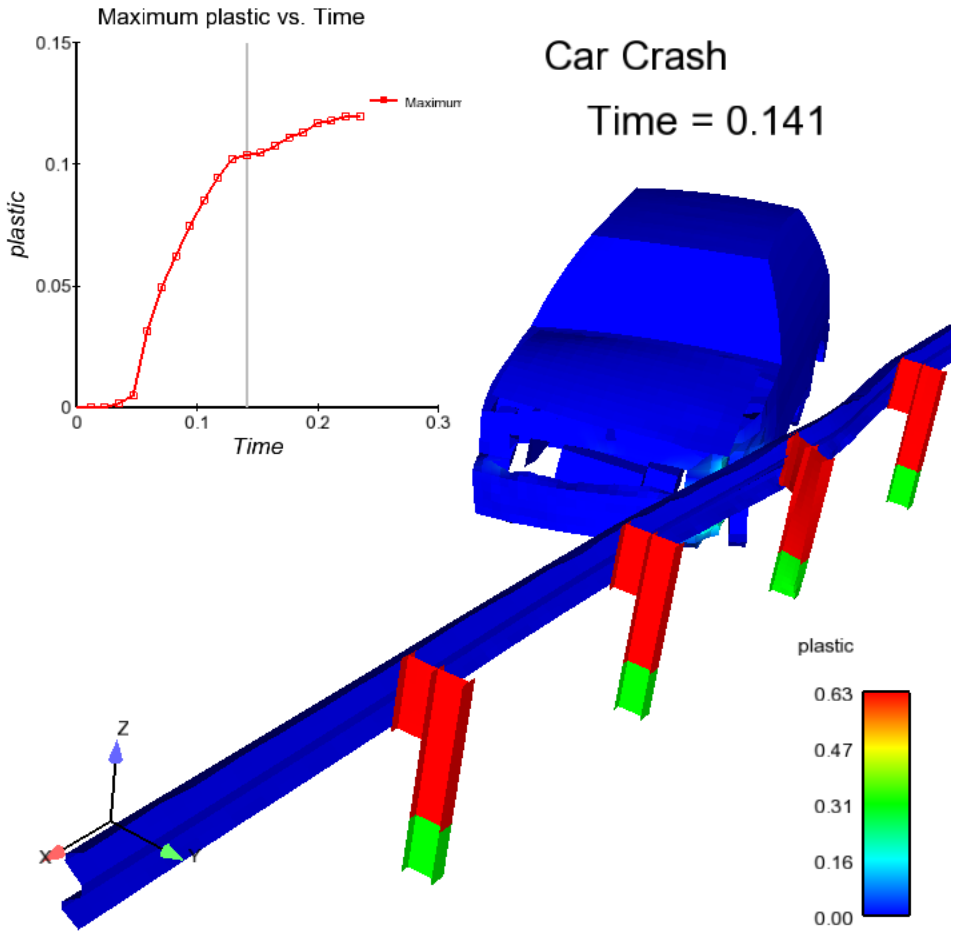
10. Many of the legend attributes can be adjusted via the Quick Action Icon bar. Make sure you select the legend (click on it in the graphics window or select it in the Annotations list)

11. Click on the Font size and select "30" from the pulldown list.

12. To adjust the format for the legend values right click on the legend and pick Format-> and choose "1.20", i.e., two decimal places.



Your Graphics Window should look something like the following:



See *How To Create Color Legends* (Help > How To Manual ...) for more information.

6.4 Flipbook Animation

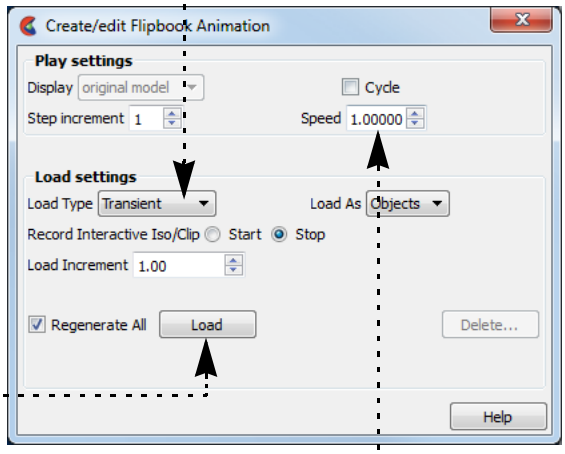
In many cases, dynamic phenomena can only be understood through interactive exploration as a transient dataset is animated. EnSight provides this capability in a *transient flipbook*. The process of creating a flipbook begins with an initial load. During this process, EnSight builds 3D graphics objects from the existing parts modified by the results at each time step. Once loading is complete, the objects can be replayed as fast as the graphics hardware permits while still allowing transformations (such as rotation).

To load a flipbook:

13. Click the Flipbook icon in the Feature Icon bar to open the Flipbook Panel.

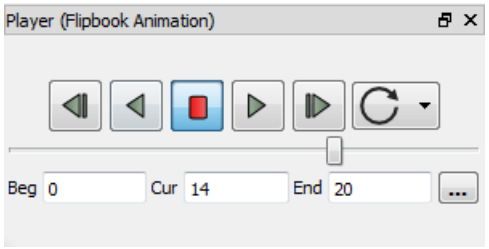
14. Be sure the Load Type is set to Transient.
15. Click Load to begin the loading process.

The Load Flipbook Status dialog displays the progress of the load. Once the load is complete the flipbook will begin to play.



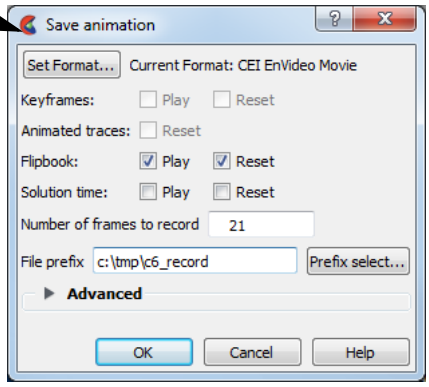
16. Click on the VCR buttons to control the animation.
17. Use the mouse to rotate the model while the animation is playing.

Note that both the plot and the time variable text update during playback. By default, the animation plays as fast as the graphics hardware permits. In some cases (for small models or very fast hardware), this can be too fast. To adjust the display speed:



18. Adjust the Speed from the Flipbook Panel by clicking the up/down arrows or entering a value.
19. You can record the animation to an animation format by utilizing the record button.

20. When you are done viewing and saving the animation, click the stop button to stop the animation, then Delete to remove the flipbook from memory.



See *How To Animate Transient Data* (Help > How To Manual ...) for more information. For greater animation control, EnSight also provides a *keyframe animation* system – see *How To Create a Keyframe Animation* (Help > How To Manual ...)for details.

6.5 Where's the Rest?

The How To Manual contains details on the operations covered in the chapter. The following online articles are relevant to the topics covered here.

Note: you get to these by selecting Help from the main menu, then How To Manual... (Help > How To Manual...), then the topic indicated below:

Consult...	For More Information On...
How To Group Parts	grouping parts into a single part
How To Display Displacements	displaying displacement variables on parts
How To Probe Interactively	probing interactively with the mouse
How To Create Text Annotation	creating annotating text
How To Create Lines	creating annotating lines and arrows
How To Create Color Legends	displaying and modifying color legends.
How To Create a Flipbook Animation	creating flipbook animations.

7 Where Do I Go From Here?

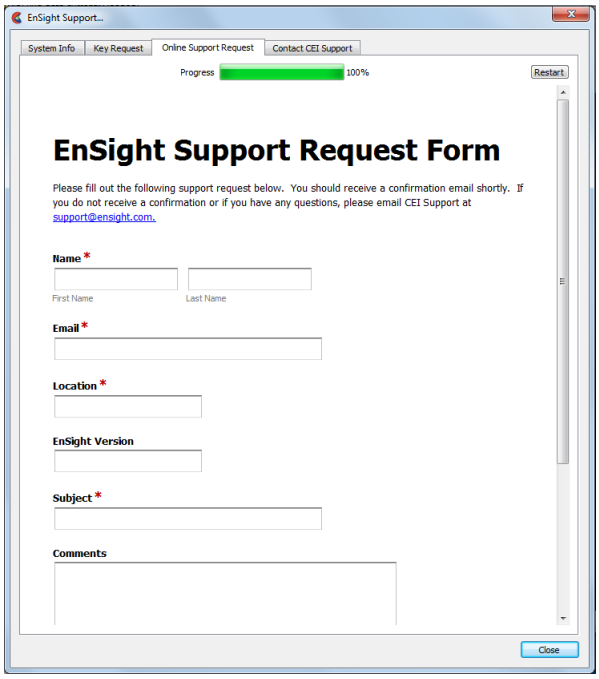
Browse the Manuals

After completing the *Getting Started* tutorials, we suggest the following approach to learning EnSight:

1. Load one of your own datasets and practice the techniques presented here with your results. See Chapter 2, *User Manual*, for information on EnSight data readers.
2. Learn new features by using the articles in the *How To Manual* (Help > How To Manual ...). The *Table of Contents* (which is where you will be sent first) is a good place to search for the topics you need. Clicking the Help buttons in the dialog windows is also a good way to find topics of interest.

Online Support Report Trouble

Still need help? Rather than sending an email to support that may lack critical details, we recommend you use our Report Trouble tab. Click this tab to email a detailed description of your problem to your EnSight Distributor. In this Tab, Toggle on the System Information to send, fill out the form, Select a Destination that is your CEI distributor (CEI in the U.S.), and click Send Report.



Contact Support

If you are evaluating EnSight for purchase or have an active support and maintenance contract you can contact CEI Support at:

Email:	support@ceisoftware.com
Hotline:	800-551-4448 (U.S)
	919-363-0883 (Non-U.S.)
Fax:	919-363-0833

The support hotline in the U.S. is staffed Monday through Friday from 8:00 AM to 5:00 PM Eastern time except during U.S. holidays. If you are outside the U.S., please contact your local distributor of EnSight for support.

Software Maintenance

CEI is constantly working to improve both the quality and functionality of EnSight. New major releases are automatically made available to all customers with an active support and maintenance contract. EnSight updates are available from CEI's website:

www.ceisoftware.com/download/

Also see our website support page for Frequently Asked Questions, white papers, performance metrics, list of available readers/translators, and updated product information.

www.ceisoftware.com/support/

Training

Training courses are held regularly by CEI and in conjunction with CEI's international distributors. Courses may be arranged at customer facilities. Contact your EnSight representative for details.