2 Input/Output

This chapter provides information on data input and output for EnSight.

2.1 Internal Readers provides a brief description of the data formats that can be read into EnSight using direct readers. It then describes how each format's data can be loaded into EnSight. Suggestions on minimizing memory usage is also noted.

2.2 User Defined Readers describes how the user defined reader API can be used to read data into EnSight.

2.3 Other External Data Sources describes other ways in which model data can be prepared to be read into EnSight.

2.4 Command Files provides a description of the files that can be saved for operations such as automatic restarting, macro generation, archiving, hardcopy output, etc.

2.5 Archive Files describes options for saving and restoring the entire current state of the program.

2.6 Context Files describes the options for saving and restoring context files.

2.7 Scenario Files describes the options for saving scenario files that can be displayed in the EnLiten program.

2.8 Saving Geometry and Results Within EnSight describes how to save model data, from any format which can be read into EnSight, as EnSight gold casefile format.

2.9 Saving and Restoring View States describes options for saving and restoring given view orientations.

2.10 Saving and Printing Graphic Images describes options for saving and printing graphic images.

2.11 Saving and Loading XY Plot Data describes options for saving and loading xy plot data.

2.12 Saving and Restoring Animation Frames describes options for saving and restoring flipbook and keyframe animation frames.

2.13 Saving Query Text Information describes options for saving query information to a text file.

2.14 Saving Your EnSight Environment describes options for saving various environment settings which affect EnSight.

2.15 Parallel Rendering Setup describes how to setup EnSight for parallel rendering.

Note: Formats for EnSight related files are described in chapters 10 and 11. Formats for the various Analysis codes are not described herein.

2.1 Internal Readers

Included in this section:

Dataset Format Basics

Reading and Loading Data Basics

EnSight Case Reader

EnSight5 Reader

ABAQUS Reader

ANSYS RESULTS Reader

ESTET Reader

FAST UNSTRUCTURED Reader

FIDAP NEUTRAL Reader

FLUENT UNIVERSAL Reader

Movie.BYU Reader

MPGS 4.1 Reader

N3S Reader

PLOT3D Reader

Dataset Format Basics

EnSight is designed to be an engineering postprocessor, yet its many features can be used in other areas as well. Its native data is defined as general finite elements or curvilinear structured data. EnSight has been used to visualize and animate results from simulations of diesel combustion, cardiovascular flow, petroleum reservoir migration, pollution dispersion, meteorological flow, and from many other disciplines. EnSight has two native data formats (EnSight5 and EnSight6) which are defined so that they can be easily interfaced to your analysis code.

(see Chapter 11, EnSight Data Formats)

EnSight reads node and element definitions from the geometry file and groups elements into an entity called a *Part*. A Part is simply a group of nodes and elements (the Part can contain different element types) which all behave the same way within EnSight and share common display attributes (such as color, line width, etc.).

EnSight allows you to read multiple datasets and work with them individually in the same active session. Each data set comprises a new “Case” and is handled by its own Server process.

EnSight also supports data formats for popular engineering simulation codes and generally used data formats.

Formats Used For Both Computational Fluid Dynamics and Structural Mechanics

- The **EnSight6** and **EnSight Gold** format supports the following files:

Case Defines all of the variables, time steps, etc. that completely describe the files which will be used for an EnSight Case.

Geometry Defines all geometric model Parts in terms of groups of finite elements, or ijk blocks.

Variable A file for each variable, which contains either scalar or vector information for every node defined in the geometry file (per_node) or for elements of various parts (per_element).

Measure/Particle Defines discrete Particles in space directly from a simulation or measured information from an experiment. The measured information can be used to compare actual versus simulated results

Boundary Defines boundary portions within and across structured blocks. (Can be EnSight's boundary file definition or a .fvbnd file.)

- The **EnSight5** format supports the following files:

  Geometry Defines all geometric model Parts in terms of groups of finite elements.

  Result Defines variable names such as Stress, Strain, and Velocity, and indicates what files these are tied to. It also, defines time information if you have a transient data case. This file is optional (and is unnecessary if your geometry is static and you have no results data).

  Variable A file for each variable, which contains either scalar or vector information for every node defined in the geometry file.

  Measured/Particle Defines discrete Particles in space directly from a simulation or measured information from an experiment. The measured information can be used to compare actual versus simulated results.

- **MPGS4** is composed of the following files:

  Geometry Defines all geometric model Parts in a general n-sided polygon format.

  Result Utilizes the EnSight results file format. This file is optional.

  Variable A file for each variable, which contains either scalar or vector information for every node defined in the geometry file.

  Measured/Particle Utilizes the EnSight5 measured/Particle file.

Formats Generally Used For Computational Fluid Dynamics

- **ESTET** contains the geometry and results information in one file. This is the native binary data format for the ESTET simulation code. The EnSight5 measured/Particle file can also be used in conjunction with these.

- **FIDAP Neutral** contains the geometry and results in one file. This file is produced by a separate procedure defined in the FIDAP documentation. If the data is time dependent this information is also defined here. The EnSight5 measured/Particle file can also be used in conjunction with these.
- **N3S** is native to the N3S simulation code and is composed of the files:

  Geometry Defines the geometry.

  Result Contains all result information describing variables and the scalar and vector information. This file is required.

  Measure/Particle Utilizes the EnSight5 measured/Particle files.
- **PLOT3D** is composed of the following files:

  Geometry Defines the geometry. This is known as a GRID file in PLOT3D and FAST. This file is a structured file format with FAST enhancements.

  Result Utilizes a modified EnSight results file format. This file is optional.

  Variable This file is a solution file (Q-file) defined in PLOT3D or a function file as defined by FAST. The modified EnSight results file provides access to multiple solution files that are produced by time dependent simulations.

  Measured/Particle Utilizes the EnSight5 measured/Particle files.

  Boundary Utilizes the EnSight's boundary file definition or a.fvbnd file.
- **FAST UNSTRUCTURED** is composed of the following files:

  Geometry Defines the geometry as unstructured triangles and/or tetrahedrons. It is the FAST unstructured single block grid file.

  Result Utilizes a modified EnSight results file format. This file is optional.

  Variable This file is a solution file (Q-file) defined in PLOT3D or a function file as defined by FAST, with I equal to the number of points and J=K=1. The modified EnSight results file provides access to multiple solution files that are produced by time dependent simulations.

  Measured/Particle Utilizes the EnSight5 measured/Particle files.

Formats Generally Used For Structural Mechanics

- **ABAQUS** can produce a .fil file which contains the geometry and results requested. EnSight can read this file in either ASCII or binary format. EnSight will read the commonly used nodal and element based results contained in this file.
- **ANSYS RESULTS** contains the geometry and results in one file. The files are defined as .rst, .rth, rfl, and .rmg files in the ANSYS documentation (EnSight

5.5 supports only the .rst file). If the data is time dependent this information is also defined here. The EnSight5 measured/Particle file can also be used in conjunction with these.

- **Movie.BYU** is composed of the following files:

  Geometry Defines all geometric model Parts in a general n-sided polygon format.

  Result Utilizes the EnSight results file format. This file is optional.

  Variable A file for each variable, which contains either scalar or vector information for every node defined in the geometry file.

  Measured/Particle Utilizes the EnSight5 measured/Particle files.

Data files are never altered by EnSight. They are used only for reading the dataset information. EnSight can produce a set of files in its native format to save geometric information that may have been read from another format or created through the postprocessing techniques. Section 2.8, Saving Geometry and Results Within EnSight

Reading and Loading Data Basics

Reading and then Loading Data into EnSight is a two step process. First, files are specified through the File Selection Dialog and then read by EnSight to the Server. Data from the files is then loaded to the Client using the Data Part Loader dialog. All Parts or a subset of those available on the Server may be loaded to the Client. You should try to reduce the amount of information that is being processed in order to minimize required memory. Here are some suggestions:

- When writing out data from your analysis software, consider what information will actually be required for postprocessing. Any filtering operation you can do at this step greatly reduces the amount of time it takes to perform the postprocessing.
- Load to the Client only those Parts that you need. For example, if you were postprocessing the air flow around an aircraft you would normally not need to see the flow field itself, but you would like to see the aircraft surface and Parts created based on the flow field (which remains available on the Server).
- For each Part you do load to the Client, a *representation* must be chosen. This visual representation can be made very simple (through the use of the Feature Angle option), or can be made complex (by showing all of the surface elements). The more you can reduce the visual representation, the faster the graphics processing will occur on the Client (see Node, Element, and Line Attributes in (see Section 3.3, Part Editing).
- If you have multiple variables in your result file, activate only those variables you want to work with. When you finish using a variable, consider deactivating it to free up memory and thereby speed processing (see Section 4.1, Variable Selection and Activation).
- When dealing with transient data in an EnSight flipbook, consider loading initially only a sampling of the available time steps—you can always load the in-between steps later if you find something interesting.

Troubleshooting Loading Data

Problem	Probable Causes	Solutions
Data loads slowly	Loading more Parts than needed	For some models, especially external fluid flow cases, there is a flow field Part which does not need to be visualized. Try eliminating the loading of this Part.
	Too many elements	Make sure the default element representation for Model Parts is set to 3D Border/2D Full before loading the data. In some cases it is helpful to set the representation to Feature Angle before loading.
	Client is swapping because it does not have enough memory to hold all the Parts specified.	Try loading fewer Parts or installing more memory to handle the dataset size.
	Server is swapping because it does not have enough memory to hold all of the Parts contained in the dataset.	Install more memory in your Server host system, reduce the number of variables activated, or somehow reduce the geometry's size. (If you can get the data in, you can cut away any area not now needed. What is left can then be saved as a geometric entity and that new dataset used for future postprocessing.)
Error reading data	Incorrect path or filename	Reenter the correct information
	Incorrect file permissions	Change the permissions of the relevant directories and files to be readable by you.
	Temporary file space is full	Temporary files are written to the default temporary directory or the directory specified by the environment variable TMPDIR for both the Client and Server. Check file space by using the command "df" and remove unnecessary files from the temporary directory or other full file systems.
	Format of the data is incorrect	Recheck the data against the data format definition.
EnSight format scalar (or vector) data loads, but appears incorrect. Often range of values off by some orders or magnitude.	Scalar (or vector) information not formatted properly in data file	Format the file according to examples listed under EnSight Variable Files in Section 2.5
	Extra white space appended to one or more of the records	Check for and remove any extra white space appended to each record

EnSight Case Reader

EnSight6 and EnSight Gold input data consists of the following files:

- Case file (required)
- Geometry file (required)
- Variable files (optional)
- Measured/Particle files (optional)
  - Measured/Particle geometry files
  - Measured/Particle variable files

The Case file is a small ASCII file which defines geometry and variable files and names, as well as time information. The Case file points to all other files which pertain to the model. The geometry file is a general finite-element format describing nodes and Parts, each Part being a collection of elements, and/or structured curvilinear ijk blocks. Measured/Particle files contain data about discrete Particles in space from the simulation code or information directly from actual experimental tests.

EnSight data is based on Parts. The Parts defined in the data are always available on the Server. However, all Parts do not have to be loaded to the Client for display. Large flow fields for CFD problems, for example, are needed for computation by the Server, but do not generally need to be seen graphically.

EnSight data can have changing geometry, in which case the changing geometry file names are contained in the Case file.

File Selection dialog

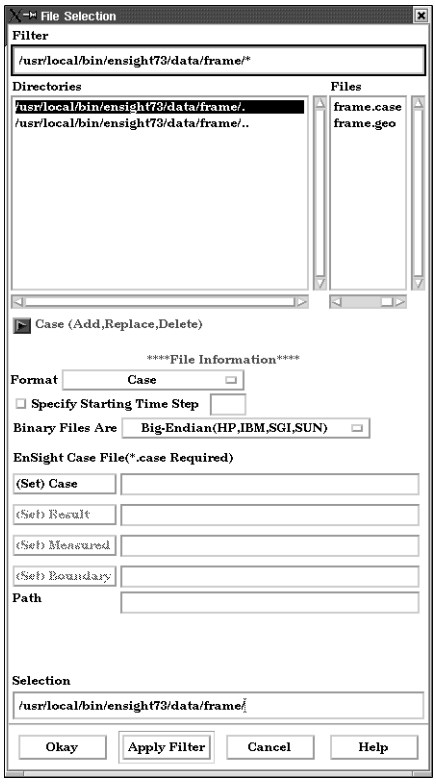


Figure 2-1

File Selection dialog for EnSight6 data

The File Selection dialog is used to specify which files you wish to read.
`Access: Main Menu > File > Data (Reader)...`

Filter — This field specifies the directory name that your data files reside in. Enter a /* at the end of the name to list all of the files and directories contained there. To filter to a smaller file list you can be more specific by entering Parts of the file names, such as /my* which will list all files and directories starting with “my”. If you only enter a /, then only the directories found will be listed. To apply the Filter, click the Apply Filter button and the Directories and Files lists will be updated and the directory will be listed in the Selection field below as the current selection.

Directories — Selection of directories available to use in the current directory. Single click to place the directory string in the Filter field. Double click to use the directory as the filter (same effect as clicking once and then clicking Apply Filter button), the Directories and Files lists will be updated and the directory will be listed in the Selection field below as the current selection. The sliding controls to the right and bottom of the list let you view all available directories.

Files — Single click to select a file. This will insert the file name after the directory listed in the Selection field. This list contains all unfiltered files that are in the filter directory.

Case

- Add... — Specify an additional case. Additional data can be read into another connected Server.
- Replace... — Specify a new case to replace an existing case.
- Delete — Delete an existing case. Case 1 cannot be deleted, but it can be replaced.

Format — Specifies the Format of the dataset. To read EnSight6 or EnSight Gold data, use the Case format.

Specify Starting Time Step — Specify starting time step. If not specified, EnSight will load the last step.

Binary Files Are — If the file is binary, sets the byte order to:
Big-Endian - byte order used for HP, IBM, SGI, SUN, NEC, and IEEE Cray.
Little-Endian - byte order used for Intel and alpha based machines.
Native to Server Machine - sets the byte order to the same as the server machine.

(Set) Geometry — Model file name for file containing at least the geometry. Clicking this button inserts the file name shown in the Selection field and inserts the path information into the Path field. File name can alternatively be typed into field.

(Set) Result — Result file name corresponding to the geometry file. For most data formats this file is optional. Clicking button inserts file name shown in Selection field and also inserts path information into Path field. File name can alternatively be typed into field.

(Set)Measured — Name of a measured file. This is an optional file. Clicking button inserts file name shown in Selection field and also inserts path information into Path field. File name can alternatively be typed into field.

Path — Path to dataset location is inserted by clicking (Set) buttons or may be entered. If blank, files are read from the Server‘s current working directory. Can use the tilde character (~) to specify home directory on the Server host system.

Selection — File or directory selected. Click the appropriate (Set) button to use information in this field.

Okay — Click to read the files specified in the (Set) fields and close the File Selection dialog.

Apply Filter — Click to apply the string in the Filter field.

Cancel — Click to close the File Selection dialog without reading the files specified in the (Set) fields.

(see How To Read EnSight6 Data)

Loading Parts from EnSight6 or EnSight Gold Data

Data Part Loader dialog for Unstructured EnSight6 or EnSight Gold Data

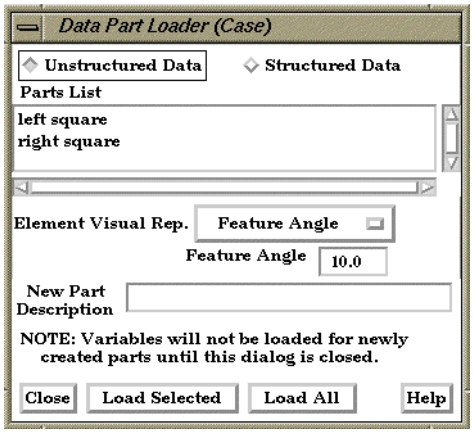


Figure 2-2

Data Part Loader dialog for EnSight6 or EnSight Gold Unstructured Data

You use the Data Part Loader dialog to control which Parts will be loaded to the Server (and made available on) the EnSight Client. It will automatically open after you have read in data and clicked Okay in the File Selection dialog.

`Access: Main Menu > File > Data (Part Loader)...`

Unstructured Data — This toggle indicates that the Part(s) listed in the Part List is(are) unstructured.

Parts List — Lists all unstructured EnSight6 format Parts in the data files which may be loaded to the Server (and subsequently to the Client). An EnSight6 or EnSight Gold data file can have unstructured, structured, or both types of Parts.

Element Visual Rep. — Parts are defined on the server as a collection of 1, 2, and 3D elements. EnSight can show you all of the faces and edges of all of these elements, but this is usually a little overwhelming, thus EnSight offers several different *Visual Representations* to simplify the view in the graphics window. Note that the Visual Representation only applies to the EnSight client—it has no affect on the data for the EnSight server.

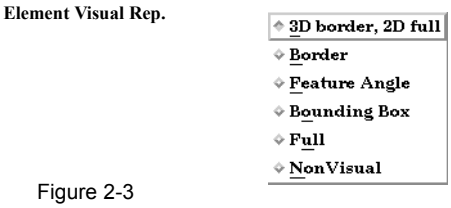


Figure 2-3

Element Visual Representation pulldown

3D Border, 2D Full — In this mode, you will see all 1D and 2D elements, but only the outside surfaces of 3D elements.

Border — In Border mode all 1D elements will be shown. Only the unique (non-shared) edges of 2D elements and the unique (non-shared) faces of 3D elements will be shown.

Feature Angle — When EnSight is asked to display a Part in this mode it first calculates the 3D Border, 2D Full representation to create a list of 1D and 2D elements. Next it looks at the angle between neighboring 2D elements. If the angle is above the Angle value specified the shared edge between the two elements is removed. Only 1D elements remain on the EnSight client after this operation.

Bounding Box — All Part elements are replaced with a bounding box surrounding the Cartesian extent of the elements of the Part.

Full	In Full Representation mode all 1D and 2D elements will be shown. In addition, all faces of all 3D elements will be shown.
Non Visual	This specifies that the loaded Part will not be visible in the Graphics Window because it is only loaded on the Server. Visibility can be turned on again later by changing the representation.
Load Selected	Loads Parts selected in Parts List to EnSight Server. The Parts are subsequently loaded to the EnSight Client using the specified Visual Representation. If Non Visual is specified, the selected Parts will be loaded to the Server, but not to the Client.
Load All	Loads all Parts in Parts List to EnSight Server. The Parts are subsequently loaded to the EnSight Client using the specified Visual Representation. If Non Visual is specified, the selected Parts will be loaded to the Server, but not to the Client.

Data Part Loader dialog for Structured EnSight6 or EnSight Gold data

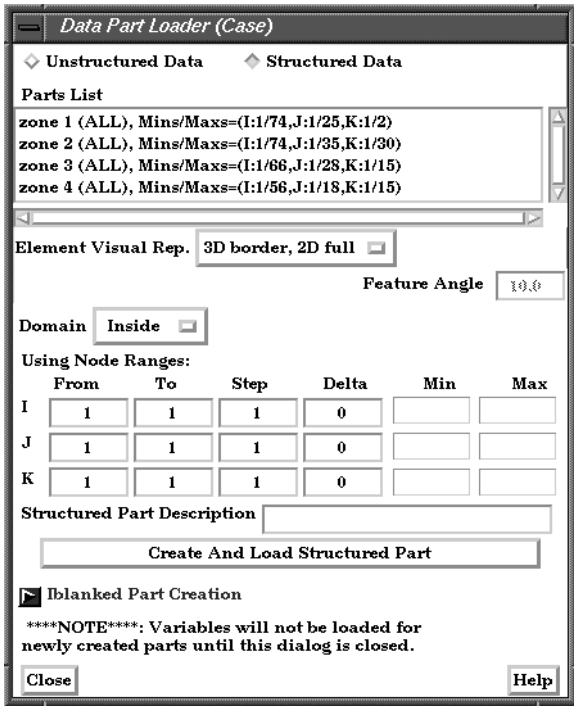


Figure 2-4

Data Part Loader dialog for EnSight6 or EnSight Gold Structured Data

You use the Data Part Loader dialog to control which structured Parts will be loaded to the EnSight Server (and subsequently to the EnSight Client). It will automatically open after you have read in data and clicked Okay in the File Selection dialog.

`Access: Main Menu > File > Data (Part Loader)...`

Structured Data	This toggle indicates that the Part data listed is structured.
Parts List	Lists all structured Parts on Server which may be loaded to the EnSight Server (and subsequently to the EnSight Client). When one part is highlighted in this list, the Domain and Node Range fields are updated accordingly.
Element Visual Rep	.Parts are defined on the server as a collection of 1, 2, and 3D elements. EnSight can show you all of the faces and edges of all of these elements, but this is usually a little overwhelming, thus EnSight offers several different *Visual Representations* to simplify the view in the graphics window. Note that the Visual Representation only applies to the

EnSight client—it has no affect on the data for the EnSight server.

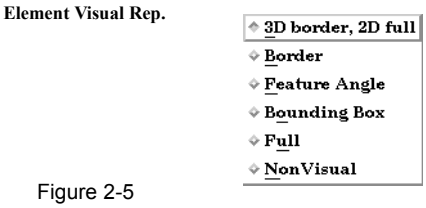


Figure 2-5
Element Visual Representation pulldown

3D Border, 2D Full — In this mode, you will see all 1D and 2D elements, but only the outside surfaces of 3D elements.

Border — In Border mode all 1D elements will be shown. Only the unique (non-shared) edges of 2D elements will be shown, and only unique (non-shared) faces of 3D elements will be shown.

Feature Angle — When EnSight is asked to display a Part in this mode it first calculates the 3D Border, 2D Full representation to create a list of 1D and 2D elements. Next it looks at the angle between neighboring 2D elements. If the angle is above the Angle value specified the shared edge between the two elements is removed. Only 1D elements remain on the EnSight client after this operation.

Bounding Box — All Part elements are replaced with a bounding box surrounding the Cartesian extent of the elements of the Part.

Full — In Full Representation mode all 1D and 2D elements will be shown. In addition, all faces of all 3D elements will be shown.

Non Visual — This specifies that the loaded Part will not be visible in the Graphics Window because it is already loaded on the EnSight Server. Visibility can be turned on again later by changing the representation.

Domain — Specifies the general iblanking option to use when creating a structured Part. If the model does not have iblanking, InSide will be specified by default.

Inside	Iblank value = 1 region
Outside	Iblank value = 0 region
All	Ignore iblanking and accept all nodes

Using Node Ranges:

From IJK — Specifies the beginning I,J,K values to use when extracting the structured Part, or a portion of it. Must be >= Min value.

To IJK — Specifies the ending I,J,K values to use when extracting the structured Part, or a portion of it. Must be <= Max value.

Valid values for the From and To fields can be positive, zero, or negative. Positive numbers are the natural 1 through Max values. Zero is a special way to indicate the max surface of a given direction. Negative values indicate surfaces back from the max, so -1 would be the next to last surface, -2 the next to next to last surface etc. There are therefore two ways to indicate any of the range values; the positive number from the min towards the max, or the zero or negative number from the max toward the min. The zero/negative method is provided for ease of use because of varying max values per part.

```
   1, 2, 3,... --->                <--- ...-3, -2 ,-1, 0
   |---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
 min                                                      max
(always 1)                                  (varies per zone)
```

Step IJK — Specifies the step increment through I,J,K. A Step value of 1 extracts all original data. A Step value of 2 extracts every other node, etc.

Delta IJK	Specifies the delta to use when creating more than one surface from the same ijk part. Only one of the directions may be non-zero. *Note that an unstructured part is the result of any non-zero delta values.*
Min IJK	Minimum I,J,K values for Part chosen
Max IJK	Maximum I,J,K values for Part chosen
Part Description	Text field into which you can enter a description for the Part
Create And Load Part	Extracts the data from the data files and creates a Part on the Server (and on the Client unless NonVisual has been specified for Representation) based on all information specified in the dialog. If only one part is highlighted, the values shown in the From and To fields (as well as the Min and Max fields) are the actual values for the selected part. Using the From and To fields you can control whether an EnSight part will be created using the entire ijk ranges or some subset of them. The Step field allows you to sample at a more coarse resolution. And the Delta field allows for multiple “surfaces” in a given part (like blade rows of a jet engine). Please note that use of a non-zero delta produces an unstructured part instead of a structured one. If more than one Part is highlighted, the values shown in the From and To fields are the combined bounding maximums of the selected parts. The same basic functionality described for a single part selection applies for multiple part selection, with one part being created for each selected part in the dialog. If the specified ranges for the multiple selection exceed the bounds of a given part, they are modified for that part so that its bounds are not exceeded. (see How To Do Structured Extraction)

Iblanked Part Creation section of Structured Part Loader dialog

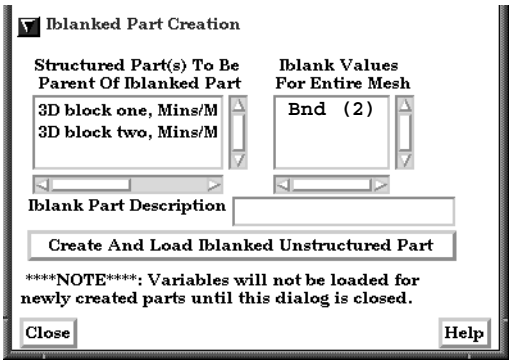


Figure 2-6

Iblanked Part Creation Section of EnSight6 or EnSight Gold Structured Part Loader dialog

You use this portion of the Part Loader dialog to further extract iblanked regions from structured parts which were created either as inside, outside, or all portions of the model.

Structured Part(s) To Be Parent of Iblanked Part	Lists all structured parts that have been created thus far in the dialog above.
Iblank Values For Entire Mesh	Lists all possible iblank values found in the model. This is a global list and may not apply to all parts.
Iblank Part description	Text field into which you can enter a description for the iblanked part.

Create And Load Iblanked Unstructured Part

Extracts a new iblanked part from an existing structured part. This new part will actually be an unstructured part.

(see How To Read EnSight6 Data)

EnSight5 Reader

EnSight5 input data consists of the following files:

- Geometry file (required)
- Result file (optional)
- Variable files (optional)
- Measured Particle Files (optional)

  - Measured/Particle geometry files
  - Measured/Particle results files
  - Measured/Particle variable files

The geometry file is a general finite-element format describing nodes and Parts, each Part being a collection of elements. The result file is a small file allowing the user to name variables and provide time information. The result file points to variable files which contain the scalar or vector information for each node. Measured/Particle files contain data about discrete Particles in space from the simulation code or information directly from actual experimental tests.

EnSight5 data is based on Parts. The Parts defined in the data are always available on the Server. However, all Parts do not have to be loaded to the Client for display. Large flow fields for CFD problems, for example, are needed for computation by the Server, but do not generally need to be seen graphically.

EnSight5 data can have changing geometry, in which case the changing geometry file names are contained in the results file. However, it is still necessary to specify an initial geometry file name.

File Selection dialog

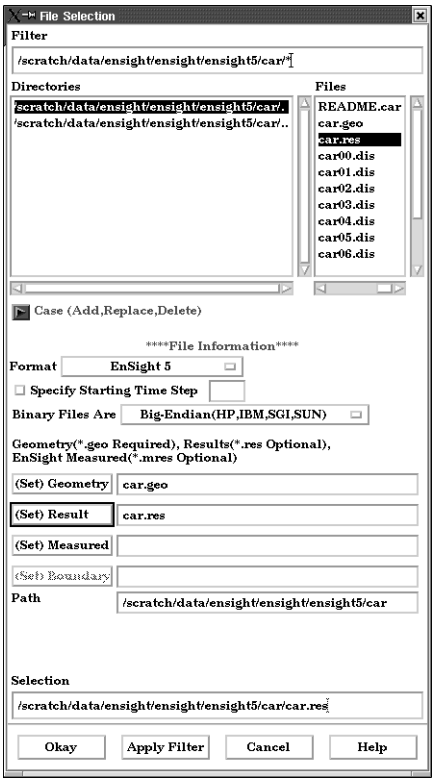


Figure 2-7

File Selection dialog for EnSight5 data

The File Selection dialog is used to specify which files you wish to read.
`Access: Main Menu > File > Data (Reader)...`

Filter — This field specifies the directory name that your data files reside in. Enter a /* at the end of the name to list all of the files and directories contained there. To filter to a smaller file list you can be more specific by entering Parts of the file names, such as /my* which will list all files and directories starting with "my". If you only enter a /, then only the directories found will be listed. To apply the Filter, click the Apply Filter button and the Directories and Files lists will be updated and the directory will be listed in the Selection field below as the current selection.

Directories — Selection of directories available to use in the current directory. Single click to place the directory string in the Filter field. Double click to use the directory as the filter (same effect as clicking once and then clicking Apply Filter button), the Directories and Files lists will be updated and the directory will be listed in the Selection field below as the current selection. The sliding controls to the right and bottom of list let you view all available directories.

Files — Single click to select a file.This will insert the file name after the directory listed in the Selection field. This list contains all unfiltered files that are in the filter directory.

Case

- Add... — Specify an additional case. Additional data can be read into another connected Server.
- Replace... — Specify a new case to replace an existing case.
- Delete — Delete an existing case. Case 1 cannot be deleted, but it can be replaced.

Format — Specifies the Format of the dataset. To read EnSight6 or EnSight Gold data, use the Case format.

Specify Starting Time Step — Specify starting time step. If not specified, EnSight will load the last step.

Binary Files Are — If the file is binary, sets the byte order to:
Big-Endian - byte order used for HP, IBM, SGI, SUN, NEC, and IEEE Cray.
Little-Endian - byte order used for Intel and alpha based machines.
Native to Server Machine - sets the byte order to the same as the server machine.

(Set) Geometry — Model geometry file name. Clicking button inserts file name shown in Selection field and inserts path information into Path field. File name can alternatively be typed into field.

(Set) Result — Result file name corresponding to the geometry file. For most data formats this file is optional. Clicking button inserts file name shown in Selection field and also inserts path information into Path field. File name can alternatively be typed into field.

(Set) Measured — Name of a measured file. This is an optional file. Clicking button inserts file name shown in Selection field and also inserts path information into Path field. File name can alternatively be typed into field.

Path — Path to dataset location is inserted by clicking (Set) buttons or may be entered. If blank, files are read from the Server's current working directory. Can use the tilde character (~) to specify home directory on the Server host system.

Selection — File or directory selected. Click the appropriate (Set) button to use information in this field.

Okay — Click to read the files specified in the (Set) fields and close the File Selection dialog.

Apply Filter — Click to apply the string in the Filter field.

Cancel — Click to close the File Selection dialog without reading the files specified in the (Set) fields.

(see How To Read EnSight5 Data)

Loading Parts from EnSight5 data

Data Part Loader dialog

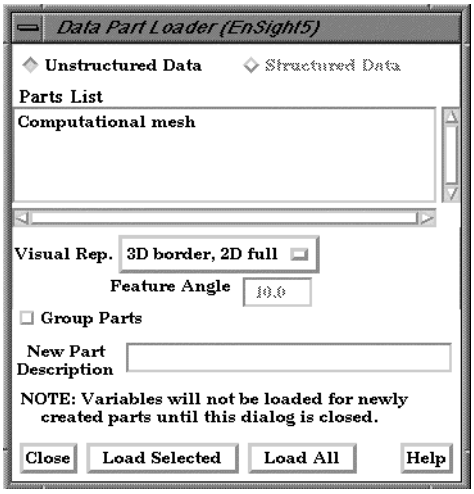


Figure 2-8
Data Part Loader dialog for EnSight5 data

You use the Data Part Loader dialog to control which Parts will be loaded to the EnSight Server (and subsequently, to the Client). It will automatically open after you have read in data and clicked Okay in the File Selection dialog.

`Access: Main Menu > File > Data (Part Loader)...`

Unstructured Data — This toggle indicates that Part data is unstructured. It must be on for EnSight5 format.

Structured Data — This toggle is not available for EnSight5 data.

Parts List — Lists all Parts in the data files which may be loaded to the EnSight Server (and subsequently, to the Client).

Element Visual Rep. — Parts are defined on the server as a collection of 1, 2, and 3D elements. EnSight can show you all of the faces and edges of all of these elements, but this is usually a little overwhelming, thus EnSight offers several different *Visual Representations* to simplify the view in the graphics window. Note that the Visual Representation only applies to the EnSight client—it has no affect on the data for the EnSight server.

Figure 2-9
Element Visual Representation pulldown

3D Border, 2D Full — In this mode, you will see all 1D and 2D elements, but only the outside surfaces of 3D elements.

Border — In Border mode all 1D elements will be shown. Only the unique (non-shared) edges of 2D elements will be shown, and only unique (non-shared) faces of 3D elements will be shown.

Feature Angle — When EnSight is asked to display a Part in this mode it first calculates the 3D Border, 2D Full representation to create with a list of 1D and 2D elements. Next it looks at the angle between neighboring 2D elements. If the angle is above the Angle value specified the

shared edge between the two elements is removed. Only 1D elements remain on the EnSight client after this operation.

Bounding Box — All Part elements are replaced with a bounding box surrounding the Cartesian extent of the elements of the Part.

Full — In Full Representation mode all 1D and 2D elements will be shown. In addition, all faces of all 3D elements will be shown.

Non Visual — This specifies that the loaded Part will not be visible in the Graphics Window because it is only loaded on the EnSight Server. Visibility can be turned on again later by changing the representation.

Load Selected — Loads Parts selected in Parts List to EnSight Server. The Parts are subsequently loaded to the EnSight Client using the specified Visual Representation. If Non Visual is specified, the selected Parts will be loaded to the Server, but not to the Client.

Load All — Loads all Parts in Parts List to EnSight Server. The Parts are subsequently loaded to the EnSight Client using the specified Visual Representation. If Non Visual is specified, the selected Parts will be loaded to the Server, but not to the Client.

(see How To Read EnSight5 Data)

ABAQUS Reader

ABAQUS input data consists of the following files:

- Geometry/Results file (required). This file (the ABAQUS .fil file) contains both the geometry and any requested results. It can be either ASCII or binary.
- EnSight5 Measured/Particle Files (optional). The measured .res file references the measured geometry and variable files.

EnSight will read ASCII or binary .fil files directly. Geometry and commonly used results contained in the file will be read.

The element sets in the .fil file will be used for creating parts.

(see How To Read ABAQUS Data)

ANSYS RESULTS Reader

ANSYS input data consists of the following files:

- Geometry and Results file (required). The ANSYS .rst file (or similar results files such as .rfl, .rmq, .rth) contains geometry and results and should be entered in the geometry field of the Data dialog.
- EnSight5 Measured/Particle Files (optional). The measured .res file references the measured geometry and variable files.

EnSight allows you to read the geometry and results data directly from an ANSYS results data file. Not all element types possible in ANSYS can be converted to EnSight format. However, EnSight will handle most practical cases just fine.

Note that certain variables may read slower than others. Displacement, acceleration, and velocity vector variables, as well as nodal solution scalars (pressure, temperature, etc.), read in quickly because they are provided at the nodes directly. Stress and strain variables, on the other hand, can be quite time consuming to read because the process involves:

1. getting the element nodal values,
2. if necessary, computing principal stresses (or strains),
3. if necessary, applying equivalent stress (or strain) and/or stress (or strain) intensity equations,
4. if shell elements, using one side or the other (user selectable),
5. averaging the values at shared nodes, and,
6. if higher order elements, averaging to get mid-side node values.

ANSYS data is based on Parts. The Parts defined in the data are always read on the Server. These Parts, however, do not all have to be loaded to the Client for display.

(see How To Read ANSYS Data)

ESTET Reader

ESTET input data consists of one file that contains all geometry and results information. The ESTET data is a structured grid. The data file is binary.

When reading this data into EnSight, you extract Parts from the mesh interactively based on indices assigned to the nodes in the data. Currently, three domains are possible for extracting Parts: inside, outside, and symmetry plane. As you extract Parts, you can limit the domains according to ranges in I, J and K.

The data can be in rectangular, cylindrical, or curvilinear coordinates. EnSight will interpret and convert properly for any of these types.

Once the desired geometry has been extracted as Parts, you are presented with a list of the results variables contained in the file. There is no way to automatically determine which of the results variables are actually vector components, so you are given the opportunity to build the vectors from the variables. The descriptions usually make this a straightforward process. All variables not used as components to vectors are assumed to be scalar variables.

ESTET Vector Builder and Data Part Loader dialogs

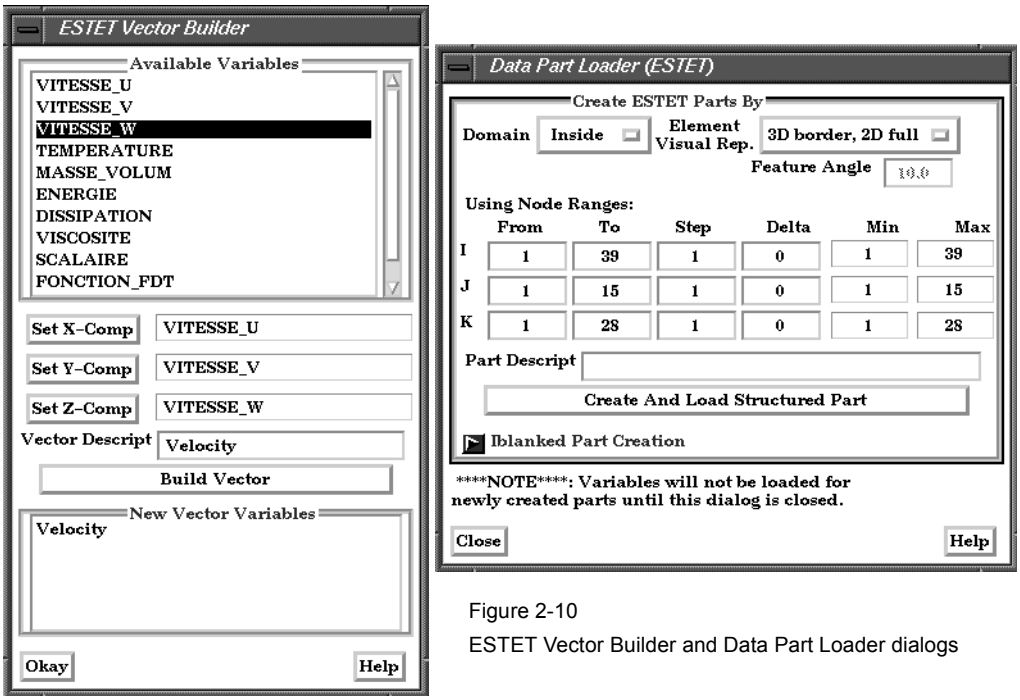


Figure 2-10
ESTET Vector Builder and Data Part Loader dialogs

You use the File Selection dialog to read ESTET data files, the ESTET Vector Builder dialog to build vector variables from scalar components for an ESTET dataset, and the Data Part Loader dialog to extract Parts from an ESTET dataset. The latter two dialogs open in sequence automatically after you click Okay in the File Selection dialog.

`Access: Main Menu > File > Data (Reader)…> ESTET`

Create ESTET Parts By

Domain	Select domain to extract Part.
Inside	Create a structured Part that contains elements whose nodes are flagged as being “inside”.
Outside	Create a structured Part that contains elements whose nodes are flagged as “outside”.
All	Create a structured Part that contains all elements because node iblanking is ignored.

Using Node Ranges: Specification of node range when creating a Part. Values must be between Min and Max.

From IJK — Specifies the beginning I,J,K values to use when extracting the structured Part, or a portion of it.

To IJK — Specifies the ending I,J,K values to use when extracting the structured Part, or a portion of it.

Valid values for the From and To fields can be positive, zero, or negative. Positive numbers are the natural 1 through Max values. Zero is a special way to indicate the max surface of a given direction. Negative values indicate surfaces back from the max, so -1 would be the next to last surface, -2 the next to next to last surface etc. There are therefore two ways to indicate any of the range values; the positive number from the min towards the max, or the zero or negative number from the max toward the min.

```
   1, 2, 3,... --->                <--- ...-3, -2 ,-1, 0
   |---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
 min                                                      max
(always 1)                                  (varies per zone)
```

Step IJK — Specifies the step increment through I,J,K. A Step value of 1 extracts all original data. A Step value of 2 extracts every other node, etc.

Delta IJK — Specifies the delta to use when creating more than one surface from the same ijk part. Only one of the directions may be non-zero. *Note that an unstructured part is the result of any non-zero delta values.*

Min IJK — Minimum I,J,K values for zone chosen

Max IJK — Maximum I,J,K values for zone chosen

Part Descrip — Set the name of the Part. If empty, EnSight will assign a name.

Create Part — Click to create a Part according to the range, step by, and delta specifications. Using the From and To fields you can control whether an EnSight part will be created using the entire ijk ranges or some subset of them. The Step field allows you to sample at a more coarse resolution. And the Delta field allows for multiple “surfaces” in a given zone (like blade rows of a jet engine). *Please note that use of a non-zero delta produces an unstructured part instead of a structured one*.

Available Variables — Selection to specify a variable to use for the next Set...Comp action.

Set X-Comp — Click to set the current selection to be the X component of the vector to build.

Set Y-Comp — Click to set the current selection to be the Y component of the vector to build.

Set Z-Comp — Click to set the current selection to be the Z component of the vector to build.

Vector Descript — Set the name of the vector variable.

Build Vector — Click to define the vector variable.

New Vector Variables — List of vector variables that have been defined.

Okay — Click Okay to load the variable information.

WARNING: You should build all the vectors you are going to use before clicking Okay, because you cannot return to this dialog. If you fail at this point to make all of the vectors desired, it is possible to do so later using the Make Vector function

(see Section 4.3, Variable Creation)

Iblanked Part Creation section of Data Part Loader (ESTET) dialog

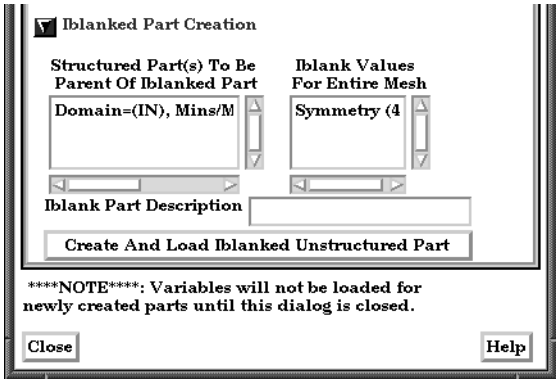


Figure 2-11

Iblanked Part Creation Section of Data Part Loader (ESTET) dialog

You use this portion of the Part Loader dialog to further extract iblanked regions from structured parts which were created either as inside, outside, or all portions of the model.

Structured Part(s) To Be Parent of Iblanked Part

Lists all structured parts that have been created thus far in the dialog above.

Iblank Values For Entire Mesh

Lists all possible iblank values found in the model. This is a global list and may not apply to all parts.

Iblank Part description

Text field into which you can enter a description for the iblanked part.

Create And Load Iblanked Unstructured Part

Extracts a new iblanked part from an existing structured part. This new part will actually be an unstructured part.

(see How To Read ESTET Data)

FAST UNSTRUCTURED Reader

FAST UNSTRUCTURED input data consists of the following files:

- Geometry file (required) This is the FAST UNSTRUCTURED single zone grid file.
- Modified Result file (optional)
- Variable files (optional) These are either a PLOT3D solution file (Q-file) or FAST function file with I = number of points and J=K=1.
- EnSight5 Measured/Particle Files (optional). The measured .res file references the measured geometry and variable files.

FAST UNSTRUCTURED is a format containing triangle and/or tetrahedron elements. The triangles have tags indicating a grouping for specific purposes. EnSight will read the unstructured single zone grid format for this data type, placing all tetrahedral elements into the first Part, and the various triangle element groupings into their own Parts.

The modified EnSight results file allows results to be read from PLOT3D-like Q-files or FAST-like function files. They can be time dependent.

FAST UNSTRUCTURED data can have changing geometry. When this is the case, the changing geometry file names are contained in the results file. However, it is still necessary to specify an initial geometry file name.

(see How To Read FAST Unstructured Data)

FIDAP NEUTRAL Reader

A FIDAP Neutral file contains all of the necessary geometry and result information for use with EnSight.

FIDAP data is based on Parts. The Parts defined in the data are always read on the Server. They do not, however, all have to be loaded to the Client for display. Large flow fields for CFD problems, for example, are needed for computation by the Server, but do not generally need to be seen graphically.

EnSight5 Measured/Particle Files can also be read with a FIDAP model. The measured .res file references the measured geometry and variable files.

(see How To Read FIDAP Neutral Data)

FLUENT UNIVERSAL Reader

FLUENT input data files consist of the following:

- Universal file (required)
- EnSight5 format Results file (optional)
- EnSight5 Measured/Particle Files (optional).

The FLUENT Universal file contains all of the necessary geometry and result information for use with EnSight for a steady-state case. If the case is transient, EnSight needs a Universal file for each time step of the analysis and a modified version of the EnSight results file.

FLUENT data is based on Parts. The Parts defined in the data are always read on the Server. They do not, however, all have to be loaded to the Client for display. Large flow fields for CFD problems, for example, are needed for computation by the Server, but do not generally need to be seen graphically.

EnSight5 Measured/Particle Files can also be read with a FLUENT model. The measured .res file references the measured geometry and variable files.

(see How To Read FLUENT Universal Data)

Movie.BYU Reader

Movie.BYU input data consists of the following files:

- Geometry file (required)
- Results file (optional)
- Variable files (optional)
- EnSight5 Measured/Particle Files (optional). The measured .res file references the measured geometry and variable files.

Movie.BYU has a general n-sided polygon data format. In translating this format to the element-based EnSight data format, not all elements possible in the Movie.BYU format can be converted to EnSight format. However, for most practical cases there are no problems.

Movie.BYU datasets can be read directly by EnSight. Additionally, an external translator, “movieto5”, is provided if you wish to convert the actual data files to EnSight format.

In order to read Movie.BYU data result files into EnSight, you must create a results file of the same format as EnSight. The external translator, “mpgs4to5,” can be used to generate a results file if you do not want to create your own using a text editor.

Movie.BYU data is based on Parts. The Parts defined in the data are always read on the Server. They do not, however, all have to be loaded to the Client for display.

Movie.BYU data can have changing geometry. When this is the case, the changing geometry file names are contained in the results file. However, it is still necessary to specify an initial geometry file name.

(see How To Read MOVIE.BYU Data)

MPGS 4.1 Reader

MPGS4 data files consist of the following:

- Geometry file (required)
- EnSight format Results file (optional)
- Variable files (optional)
- EnSight5 Measured/Particle Files (optional). The measured .res file references the measured geometry and variable files.

MPGS4.x uses a general n-sided polygon, n-faced polyhedral data format. In going from this format to the specific element data format of EnSight, you encounter the problem associated with translating from a general format to a specific format. Not all elements possible in MPGS4.x can be converted to EnSight format. However, there will not be a problem in most situations.

MPGS4.x models of modest size can be read directly into EnSight. Size can become an issue since the amount of memory needed to do the conversion in EnSight to the internal data format in a reasonable length of time can become excessive for large models. An external translator, “mpgs4to5”, is provided for the larger models. You should also consider using the external translator to convert MPGS4.x data to EnSight data if you need to continue loading the same dataset, as this will perform the data conversion one time while reading it into EnSight will continue to take resources each time the data is read. Converting it from MPGS4.x to EnSight format also has the advantage of taking less disk space as the EnSight format is more compact.

In order to read MPGS4.x results directly into EnSight, you must create a results file of the same format as EnSight. The external translator, “mpgs4to5,” can be used to generate a results file if you do not want to create your own using an editor.

MPGS4.x data is based on Parts. The Parts defined in the data are always read on the Server. They do not, however, all have to be loaded to the Client for display. Large flow fields for CFD problems, for example, are needed for computation on the Server, but do not generally need to be seen graphically.

MPGS4.x data can have changing geometry. When this is the case, the changing geometry file names are contained in the results file. However, it is still necessary to specify an initial geometry file name.

(see How To Read MPGS Data)

N3S Reader

N3S input data consists of the following files:

- Geometry file (required)
- Results file (required)
- EnSight5 Measured/Particle Files (optional). The measured .res file references the measured geometry and variable files.

N3S is a data format developed by Electricité de France (EDF) consisting of a geometry file and a results file. For this data format, both files are always required. Versions 3.0 and 3.1 are both supported.

When reading N3S data into EnSight, you extract Parts from the mesh interactively based on different color numbers or boundary conditions. The available color numbers and boundary conditions for the model are presented.

N3S Part Creator dialog

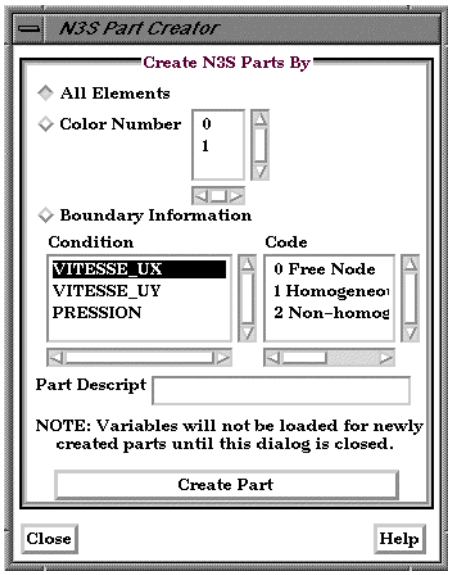


Figure 2-12

N3S Part Creator dialog

You use the File Selection dialog to read in N3S dataset files. You use the N3S Part Creator dialog to extract Parts from a N3S dataset.

`Access: Main Menu > File > Data (Reader)...> N3S`

Create N3S Parts By

All Elements	Selection to create a Part using all of the elements available within the data file.
Color Number	Selection to create a Part according to the color number associated with each element.
Boundary Information	Selection to create a Part according to specified conditions and codes.
Condition	Select boundary condition to use for Part creation.
Code	Select Code to use for boundary condition.

Part Descript — Specify name for Part.

Create Part — Click to create a Part. The Part is listed in the main Parts list of the Parts & Frames dialog and is displayed in the Main View window.

(see How To Read N3S Data)

PLOT3D Reader

PLOT3D is a commonly used structured data format and input data consists of the following files:

- Geometry file. This is a required file. (Structured GRID file with FAST enhancements)
- Modified EnSight Results file (optional). A standard plot3d Q-file can be read in the results field in place of a modified EnSight Results file.
- Variable files, which are solution (PLOT3D) or function (FAST) files (optional)
- EnSight5 Measured/Particle Files (optional). The measured .res file references the measured geometry and variable files.
- EnSight Boundary File (optional). The boundary file defines boundary portions within and/or across structured blocks. (Note: this can be EnSight's boundary file format or a .fvbnd file.)

When reading PLOT3D files into EnSight, you extract Parts from the mesh based on a domain, a list of zones, and/or indices assigned to the nodes in the data. Currently, three domains are possible for extracting structured Parts: (a) inside, (b) outside, or (c) all. These options are dependent on what the file format is from the parameters defined in the previous paragraph. For instance, when using single zone, non-iblanked data the domain is fixed at "Inside" and the one zone listed in the zones list is selected. As you extract Parts from a single zone file, however, it is possible to limit the domains according to ranges in I, J, and K.

Once the desired structured Parts have been extracted from the geometry, further iblanking options can be used to extract unstructured parts, such as for boundaries. When the Data Reader (PLOT3D) dismissed, the user is presented with a list of the result variables available from the result file

To successfully read PLOT3D data, the following information must be known about the data:

1. format - ASCII, C binary, or Fortran binary
2. whether single or multizone
3. dimension - 3D, 2D, or 1D
4. whether iblanked or not
5. precision - single or double

EnSight attempts to determine these five settings automatically from the grid file. The settings that were determined (for the first four) are shown in the Part Builder dialog, where you can override them manually if needed.

The precision setting is not reflected in the dialog, but is echoed in the Server shell window. The q (or function) file precision will by default be set the same as that of the grid file. In the rare case where the automatic detection is wrong for the grid file or the precision is different for the q (or function) file than for the grid file, commands can be entered into the Command dialog to manually set the precision.

test:	`plot3d_grid_single`	to read grid file as single precision
test:	`plot3d_grid_double`	to read grid file as double precision
test:	`plot3d_qr_single`	to read q (or function) file as single precision
test:	`plot3d_qr_double`	to read q (or function) file as double precision

PLOT3D Part Loader dialog

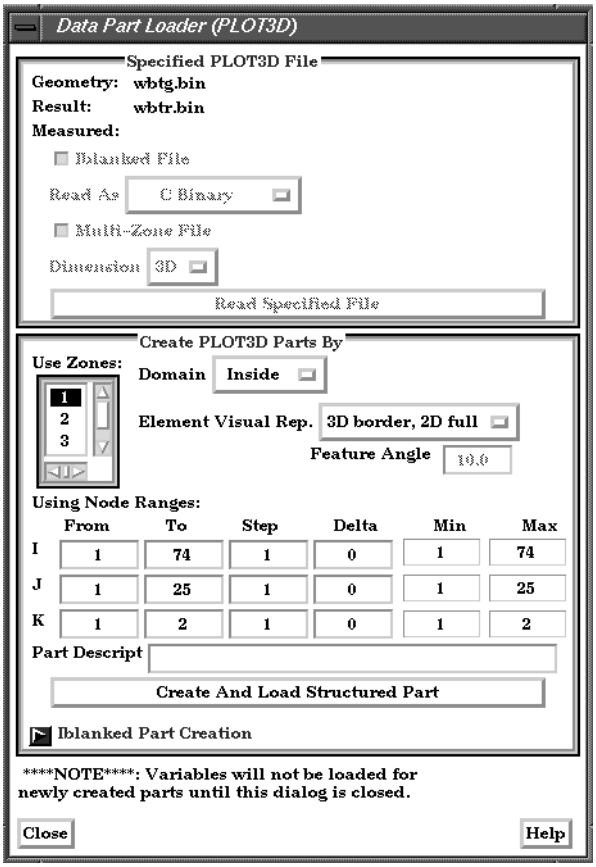


Figure 2-13
Part Data Loader (PLOT3D)

You use the Part Data Loader (PLOT3D) dialog to read a specified PLOT3D file and to extract parts out of the PLOT3D geometry.

`Access: Main Menu > File > Data (Reader)... > PLOT3D`

Geometry — Specifies name of file which geometry data will be read.

Result — Specifies name of file which result data will be read.

Measured — Specifies name of file which measured/discrete data will be read.

IBlanked File Toggle — Turn on if geometry field has iblanking.

Read As — Specifies file type. Choices are:

ASCII
C Binary (Note: Files may not be portable across hardware platforms).
FORTRAN Binary (Note: Files may not be portable across hardware platforms).

Multi-Zone File Toggle — Turn on if dataset contains multiple zones. If Multi-zoned and you are not doing a "between boundary" domain option (see below), a part can span several zones (see Use Zone list below).

Dimension — Specifies the dimension of the dataset. Options are 1D, 2D, or 3D. If multi-zone, the dimension of the problem is forced to be 3D.

Read Specified File — Click to initiate the reading process.

Use Zones — List of Zones defined in the data that can be used to create Parts. If there are multiple zones you can select one or more of them.

Domain Select the domain to create a structured Part from. Options are:

Inside	Create structured Part from grid points flagged with Iblanking = 1
Outside	Create structured Part from grid points flagged with Iblanking = 0.
All	Create part from all grid points (ignores Iblanking).

Using Node Ranges

From IJK Specifies the beginning I,J,K values to use when extracting the structured Part, or a portion of it.

To IJK Specifies the ending I,J,K values to use when extracting the structured Part, or a portion of it.

Valid values for the From and To fields can be positive, zero, or negative. Positive numbers are the natural 1 through Max values. Zero is a special way to indicate the max surface of a given direction. Negative values indicate surfaces back from the max, so -1 would be the next to last surface, -2 the next to next to last surface etc. There are therefore two ways to indicate any of the range values; the positive number from the min towards the max, or the zero or negative number from the max toward the min. The zero/negative method is provided for ease of use because of varying max values per part.

```
   1, 2, 3,... --->                <--- ...-3, -2 ,-1, 0
   |---|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
 min                                                      max
(always 1)                                  (varies per zone)
```

Step IJK Specifies the step increment through I,J,K. A Step value of 1 extracts all original data. A Step value of 2 extracts every other node, etc.

Delta IJK Specifies the delta to use when creating more than one surface from the same ijk part. Only one of the directions may be non-zero. Note that an unstructured part is the result of any non-zero delta values.

Min IJK Minimum I,J,K values for Part chosen

Max IJK Maximum I,J,K values for Part chosen

Part Descrip Specify name of Part you wish to create.

Create Part Click to create a Part.

If only one part is highlighted, the values shown in the From and To fields (as well as the Min and Max fields) are the actual values for the selected part. Using the From and To fields you can control whether an EnSight part will be created using the entire ijk ranges or some subset of them. The Step field allows you to sample at a more coarse resolution. And the Delta field allows for multiple “surfaces” in a given part (like blade rows of a jet engine). Please note that use of a non-zero delta produces an unstructured part instead of a structured one.

If more than one Part is highlighted, the values shown in the From and To fields are the combined bounding maximums of the selected parts. The same basic functionality described for a single part selection applies for multiple part selection, with one part being created for each selected part in the dialog. If the specified ranges for the multiple selection exceed the bounds of a given part, they are modified for that part so that its bounds are not exceeded.

(see How To Do Structured Extraction)

Iblanked Part Creation section of Data Part Loader (PLOT3D) dialog

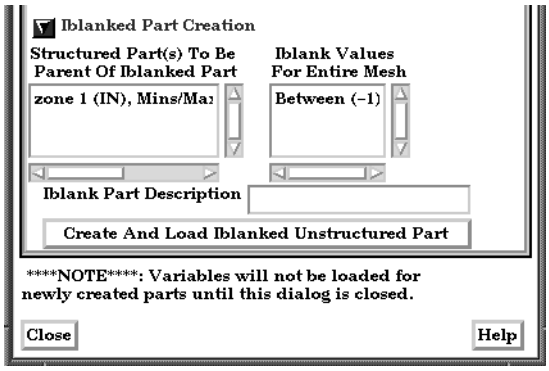


Figure 2-14

Iblanked Part Creation Section of Data Part Loader (PLOT3D) dialog

You use this portion of the Part Loader dialog to further extract iblanked regions from structured parts which were created either as inside, outside, or all portions of the model.

Structured Part(s) To Be Parent of Iblanked Part — Lists all structured parts that have been created thus far in the dialog above.

Iblank Values For Entire Mesh — Lists all possible iblank values found in the model. This is a global list and may not apply to all parts.

Iblank Part description — Text field into which you can enter a description for the iblanked part.

Create And Load Iblanked Unstructured Part — Extracts a new iblanked part from an existing structured part. This new part will actually be an unstructured part.

(see How To Read PLOT3D Data)

2.2 User Defined Readers

A user defined reader capability is included in EnSight which allows otherwise unsupported structured or unstructured data to be read. In other words, the user can create their own data readers. Each user defined reader utilizes a dynamic shared library produced by the user. Once produced, these readers show up in the list of data formats in the Data Reader Dialog just like an internal readers.

The readers are produced by creating the routines of an API. Two versions of the userd defined API are available starting with EnSight version 7.2. The 1.0 API (which has been available since EnSight version 6) was designed to be friendly to those producing it, but requires more manipulation internally by EnSight and accordingly requires more memory and processing time. The 2.0 API is considerably more efficient, and was designed with that in mind. It requires that all data be provided on a part basis, and as such lends itself closely to the EnSight Gold type format. A few of the advantages of the new 2.0 API are:

- Considerably more memory efficient
- Considerably faster
- Tensor variables are supported
- Complex variables are supported
- Geometry and variables can be provided on different time lines
- If boundary representation is available, provides for its use instead of having EnSight compute it.

The process for creating and using a user-defined reader is explained in detail in the README files on the installation CD or in your installation directory.

On the CD: /CDROM/ensight74/user_defined_src/readers

In installation directory: $CEI_HOME/ensight74/user_defined_src/readers

Therein you will also find a detailed description of each routine in the API and the order in which the routines are called by EnSight. In the subdirectories at the same location as the README's, you will find source code for some sample readers and a README file for working readers. The sample are often helpful examples when producing your own reader.

The actual working user defined readers included in the EnSight distribution may vary, but typically would include such formats as:

CFF	PXI	SCRYU
CFX4	FLOW-3D	SILO
Cobalt	LS-DYNA	STL
MSC/Dytran	NASTRAN	TECPLOT
EXODUS II	OP2	VECTIS

2.3 Other External Data Sources

External Translators

Translators supplied with the EnSight application enable you to use data files from many popular engineering packages. These translators are found in the translators directory on the EnSight distribution CD. A README file is supplied for each translator to help you understand the operation of each Particular translator. These translators are not supported by CEI, but are supplied at no-cost and as source files, where possible, to allow user modification and porting.

Exported from Analysis Codes

Several Analysis codes can export data in EnSight file formats. Examples of these include Fluent, STAR-CD, CFX and others.

2.4 Command Files

Command files contain EnSight command language as ASCII text that can be examined and even edited. They can be saved starting at any point and ending at any point during an EnSight session. They can be replayed at any point in an EnSight session. However, *some command sequences require a certain state to exist*, such as connection to the Server, the data read, or a Part created with a Particular Part number.

There are a multitude of applications for command files in EnSight. They include such things as being able to play back an entire EnSight session, easily returning to a standard orientation, connecting to a specific host, creating Particle traces, setting up a keyframe animation, etc. Anything that you will want to be able to repeatedly do is a candidate for a command file. Further, if it is a task that you frequently do, you can turn the command file into a macro (see To Use Macros below).

Documenting Bugs

Command files are one of the best ways of documenting any bugs found in the EnSight system. Hopefully that is a rare occasion, but if it occurs, a command file provided to CEI will greatly facilitate the correction of the bug.

Nested Command Files

Command files can be nested, which means that if you have a command file that does a specific operation, you can play that command file from any other command file, as long as any prerequisite requirements are completed. This is done by adding the command `play: <filename>` in the command file.

Default Command File

EnSight is always saving a command file referred to as the *default command file* (unless the you have turned off this feature with a Client command line option). This command file can be saved (and renamed) when exiting EnSight, as described later in this section. The default command file is primarily intended to be a crash recovery aid. If something unforeseen were to prematurely end your EnSight session, you can recover to the last successfully completed command by restarting EnSight and running the default command file. Saving the Default Command File for EnSight Session

Command dialog

Figure 2-15
Command dialog

You use the Command dialog to control the execution of EnSight command language. The language can be entered by hand, or as is most often the case, played from a file. This

dialog also controls the recording of command files as well as Macro path definition.
`Access: Main Menu > File > Command...`

Command History	Displays most recent command language executed (or recorded). Can click on an entry which will bring entry to the Command Entry field.
Command Entry	Command language entry. Enter command and press RETURN. During file playback, next command to be executed is shown here. Any command preceded by a # is a comment line.
Record	Select to start play file recording. Will be prompted for file name. Can simultaneously record and play files. When engaged, all actions in EnSight are recorded to the specified file.
Record Part Selection By	Select the method by which parts will be recorded in the command language - either by Number (default) or by Name.
Play	Select to start playing a command file. You must provide the command language file name. Command play continues as long as there are commands in file, an interrupt: command has not been processed, or the Interrupt button has not been pressed.
Interrupt	Interrupt playback of the command file.
Step	Step though commands of play file. File playback must be stopped. Each click will execute next command shown in Command Entry field.
Skip	Skip over the playing command file's next command (shown in Command Entry box).
Continue	Continue playing interrupted command file.
Delay Between Commands	Set the delay between commands in seconds when playing a command file.
Delay Refresh	When enabled, will cause the EnSight graphics window to refresh only after the playfile processing has completed or has been interrupted by the user.
Reload Macros	Causes the Macro definitions to be reread from the site preferences directory and from the user's .ensight directory.

Troubleshooting Command Files

This section describes some common errors when running commands. If an error is encountered while playing back a command file you can possibly retype the command or continue without the command.

Problem	Probable Causes	Solutions
Error in command category	Incorrect spelling in the command category	Check and fix spelling
Command does not exist	Incorrect spelling in the command	Check and fix spelling
Error in parameter	Incorrect integer, float, range, or string value parameter	Fix spelling or enter a legal value
Commands do not seem to play	Command file was interrupted by an error or an interrupt command	Click continue in the Command dialog

(see How To Record and Play Command Files)

Saving the Default Command File for EnSight Session

EnSight is always saving a command file referred to as the *default command file* (unless the you have turned off this feature with a Client command line option). This default command file receives a default name starting with “ensigAAA” and is written to your /usr/tmp directory (unless you set your TMPDIR environment variable). This command file can be saved (and renamed) when exiting EnSight. If you do not save this temporary file in the manner explained below, it will be deleted automatically for you when you Quit EnSight.

Quit Confirmation dialog

Figure 2-16
Quit Confirmation dialog

You use the Quit Confirmation dialog to save either or both the default command file and an archive file before exiting the program.

`Access: Main Menu > File > Quit...`

Save Command Backup File To: Toggle-on to save the default command file. Can also specify a new name for the command file. (see Section 2.4, Command Files for more information on using command files.)

Save Full Backup Toggle-on and specify a name to create a Full Backup file.

Yes Click to save the indicated files and terminate the program.

(see How To Record and Play Command Files)

2.5 Archive Files

Saving and Restoring a Full backup

The current state of the EnSight Client and Server host systems may be saved to files. An EnSight session may then be restored to this saved state after restarting at a later time. A Full Backup consists of the following files. First, a small archive information file is created containing the location and name of the Client & Server files that will be described next. Second, a file is created on the Client host system containing the entire state of the Client. Third, a file is created on each Server containing the entire state of that Server. You have control over the name and location of the first file, but only the directories for the other files.

Restoring EnSight to a previously saved state will leave the system in exactly the state EnSight was in at the time of the backup. For a restore to be successful, it is important that EnSight be in a "clean" state. This means that no data can be read in before performing a restore. During a restore, any auto connections to the Server(s) will be made for you. If manual connections were originally used, you will need to once again make them during the restore. (If more than one case was present when the archive was saved, then connection to all the Servers is necessary).

An alternative to a Full Backup is to record a command file up to the state the user wishes to restore at a later date, and then simply replaying the command file. However, this requires execution of the entire command file to get to the restart point. A Full Backup returns you right to the restart point without having to recompute any previous actions.

A Full Backup restores very quickly. If you have very large datasets that take a significant time to read, consider reading them and then immediately writing a Full Backup file. Then, use the Full Backup file for subsequent session instead of reading the data.

Save Full Backup Archive dialog

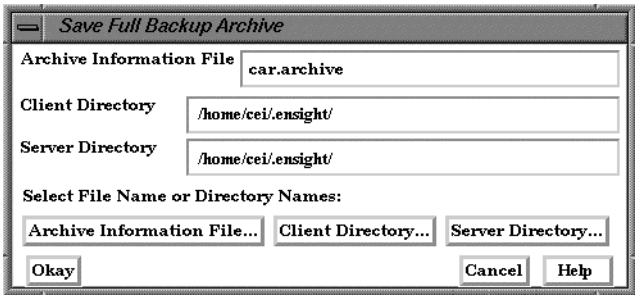


Figure 2-17

Save Full Backup Archive dialog

You use the Save Full Backup Archive dialog to control the files necessary to perform a full archive on EnSight.

`Access: Main Menu > File > Backup > Save Full Backup...`

Archive Information File — Specifies name of Full Backup control file.

Client Directory — Specifies the directory for the Client archive file.

Server Directory	Specifies the directory for the Server archive file.
Archive information File...	Click to display the file selection dialog for specifying the Archive Information File.
Client Directory...	Click to display the file selection dialog for specifying the Client Directory.
Server Directory...	Click to display the file selection dialog for specifying the Server Directory (for the selected case if there is more than one). Choose a common path if there is more than one.
Okay	Click to perform the full backup. *NOTE: This command is written to the command file, but is preceded with a # (the comment character). To make the archive command occur when you play the command file back, uncomment the #.* (see How To Save and Restore an Archive)

File Selection for Restarting from an Archive

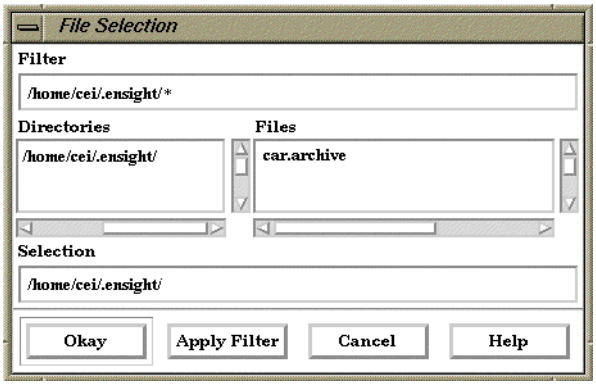


Figure 2-18

File Selection for Restarting from an Archive

You use the Restore Full Archive Backup dialog to read and restore a previously stored archive file.

`Access: Main Menu > File > Backup > Restore Full...`

Troubleshooting Full Backup

Problem	Probable Causes	Solutions
Error message indicating that all dialogs must be dismissed	When saving and restoring archives, all EnSight dialogs, except for the Client GUI, must be dismissed to free up any temporary tables that are in use. Temporary tables are not written to the archive files.	Dismiss all the Motif dialogs except the main Client GUI.
Backup fails for any reason	Ran out of disk space on the Client or Server host system	Check the file system you are writing to, on both the Server and the Client host systems, with the command “df” then remove any unnecessary files to free up disk space.
	Directory specified is not writable	Change permissions of destination directory or specify alternate location.

2.6 Context Files

EnSight context files can be used to duplicate the current EnSight state with a different, but similar, dataset. The context file works best if the dataset it is being applied to contains the same variable names and parts, but can also be used when this is not the case.

Input and output of context files is described below as well as in How To Save or Restore a Context File and under Save and Restore of Section 6.1, File Menu Functions

Saving a Context File

To save the current context, simply entered the desired file name in the dialog under:

```
Access:  File > Save > Context...
```

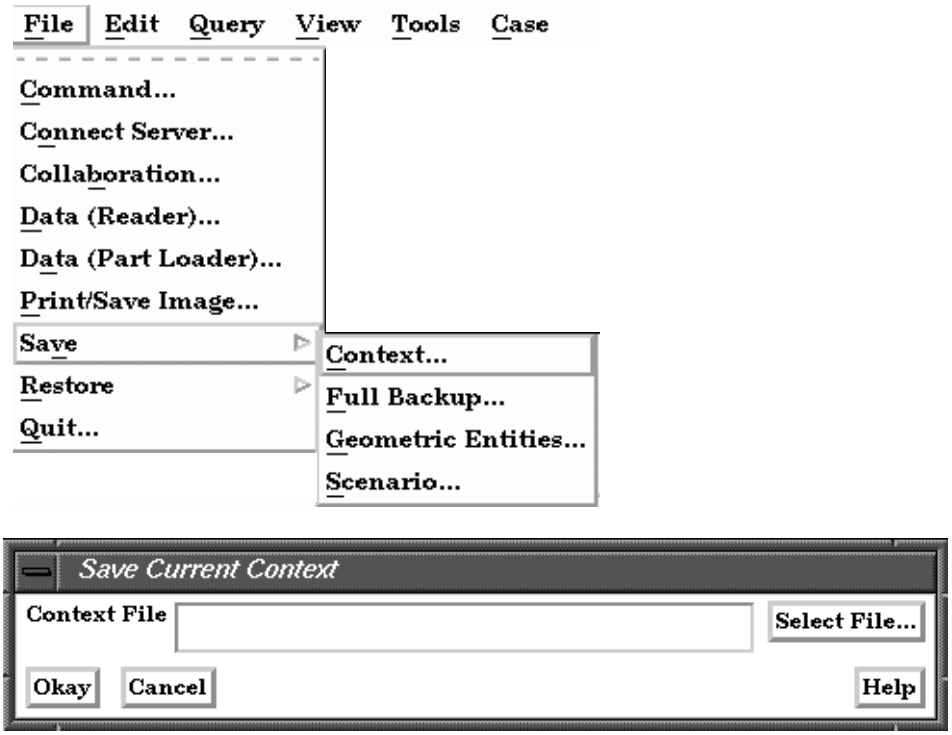
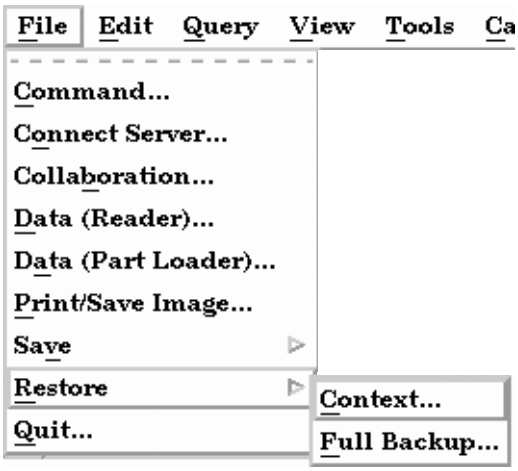


Figure 2-19
Saving a Context File

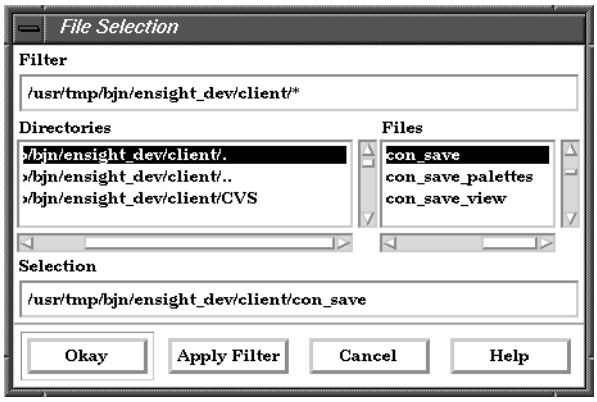
Restoring a Context

When restoring a context you can either read the new dataset and build the new parts and then restore the context file, or you can read the new dataset, close the part builder without building any parts and restore the context file (whereupon the context file will build the same parts as existed when it was saved). The way you decide to do this depends upon whether the same parts exist in the new dataset.

If the same parts do not exist, you would typically read the new dataset and build the desired parts in the normal way. Then:



select the context file saved in an earlier session and restore it. Either or both of the next dialogs may appear if they are needed to resolve part or variable name differences. As you can see, you can map these as needed.



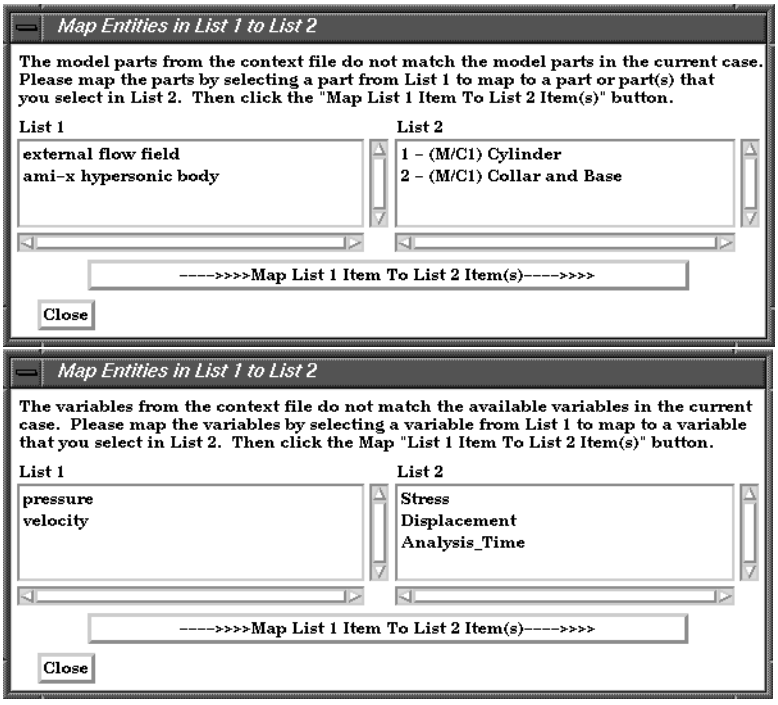


Figure 2-20
Restoring a Context

Flipbook animations are not restored using the context file because it is unknown at the time the context file is created what state existed when the flipbook was saved.

Context files use EnSight's command language and other state files (such as palette, view, and keyframe animation). Context files cannot be used to restore state from multiple cases. This limitation will be removed in the future.

2.7 Scenario Files

Scenario files are used by CEI's EnLiten product which is capable of viewing all geometry (such as parts, annotation, plots, etc.) that EnSight can display, including flipbook, keyframe, and particle trace animations.

A "scenario" defines all visible entities you wish to view with EnLiten and includes any saved views and notes that you want to make available to the EnLiten user.

When you create a scenario, the following may be saved: (a) EnLiten file containing geometric display information, saved views, and attached nodes. (b) A palette file for each visible variable legend. (c) A JPEG image file (not used by EnLiten). (d) A scenario description file (not used by EnLiten). (e) A EnSight context file (not used by EnLiten).

When saving a scenario, either the scenario file itself can be saved, or the scenario project - which includes all of the files in the previous paragraph.

EnLiten is a geometry viewer only. As such it is not capable of creating or modifying any new/existing information such as variables or parts, or of changing timesteps.

Since EnLiten is only a geometry viewer, only keyframe transformation information is stored when saving a scenario file, i.e., no transient data keyframing is possible (consider loading a flipbook instead)

.

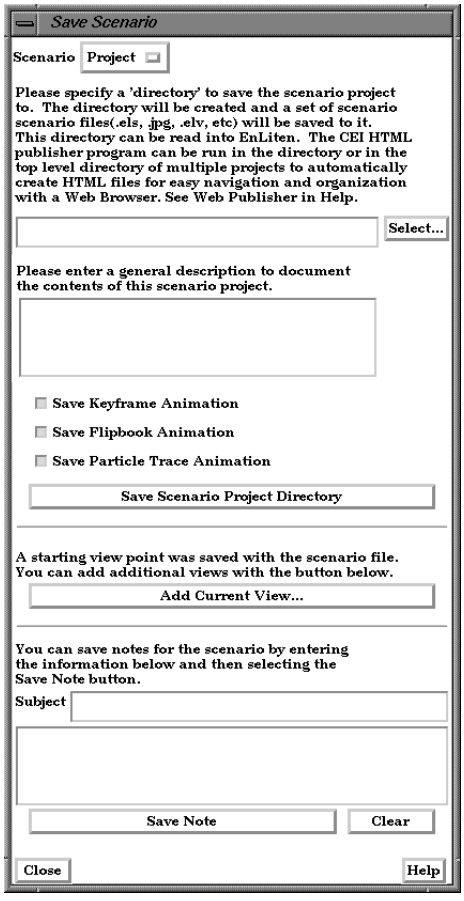


Figure 2-21
Save Scenario Dialog

You use the Save Scenario dialog to control the options of the scenario files to be saved in EnSight for display in EnLiten.

`Access: Main Menu > File > Save > Scenario...`

Scenario	*Project* - will save the scenario file plus files mentioned on the previous page. *File* - will save only the scenario file.
Scenario Directory/ File	Specifies the directory or file to which the scenario information will be written.
General Description	Specifies the general description which will be used when a html page is generated for the scenario.
Save Keyframe Animation Toggle	Select to have any currently defined keyframe animation sequence saved to the scenario.
Save Flipbook Animation Toggle	Select to have any currently defined flipbook animation saved to the scenario.
Save Particle Trace Animation Toggle	Select to have any currently defined particle trace animation saved to the scenario.
Save Scenario	Click to actually save the scenario.
Add Current View...	After the scenario has been saved you may save additional views by setting the desired view in EnSight, then selecting this button You will be asked to name the view in a resulting pop-up dialog.
Save Note	After the scenario has been saved you may write notes regarding the scenario by entering a Subject line and typing in the notes input area. When satisfied, select this button.

(see How To Save Scenario)

2.8 Saving Geometry and Results Within EnSight

Saving Geometric Entities

Sometimes you may wish to output geometric information from EnSight to be included in a different analysis code, or be part of a HTML based presentation.

EnSight allows you to save the geometric information in Case(EnSight6), VRML, or Case(EnSight Gold) (if you have a Gold license) formats. If you choose to save the geometric information in one of the EnSight Case formats, the files will be written on the Server in either ASCII or binary format. You may also choose to save multiple time steps. The Case file is also created. The geometry, as well as all of the active variables will be saved to files. This feature is limited to saving the following Part types: Model, 2D-Clips, Elevated Surfaces, Developed Surfaces, and Isosurfaces.

The other choice, which is to save the information in a VRML formatted file will allow you to save all of the visible Parts that reside on the server (thus, particle traces, vector arrows, contours, etc. will not be saved) in their current visual state except for Parts which have limit fringes set to transparent. The VRML file will be saved on the Client.

Save Geometric Entities dialog

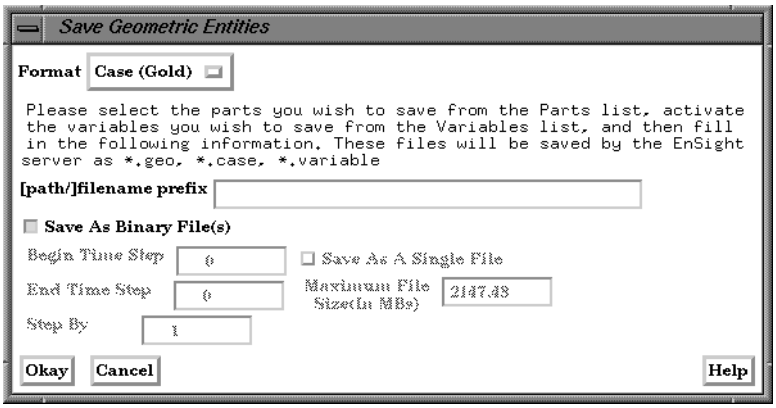


Figure 2-22

Save Geometric Entities dialog

The Save Geometric Entities dialog is used to save Selected Model, 2D-Clip, Isosurface, Elevated Surface, and Developed Surface Parts as EnSight6 files. Thus modified model Parts and certain classes of created Parts can become model Parts of a new dataset. All geometric and variable information is saved for all indicated time steps in either EnSight6 ASCII or binary format.

`Access: Main Menu > File > Save > Save Geometric Entities...`

Format — Specify the desired format: Case(EnSight Gold), VRML, or STL

[path]/filename prefix — Specify path and filename prefix name for the saved files. The saved geometry file will be named filename.geo, the result file will be filename.res, and the active variables will be filename.variablename. The VRML file will be filename.wrl.

Save As Binary File(s) — Save as Binary File(s) specifies whether to save the data in EnSight6 ASCII (button toggled off - default) or binary (button toggled on) format.

Begin Time Step — Begin Time Step field specifies the initial time step for which information will be saved for all selected Parts and activated variables.

End Time Step	End Time Step field specifies the final time step for which information will be saved for all selected Parts and activated variables.
Step By	Step By field specifies the time step increment for which information will be saved for all selected Parts and activated variables starting with Begin Time Step and finishing with End Time Step. The Step By value MUST be an integer.
Save as a Single File	Toggle on to have a single file per variable - containing all values for all time steps for that variable. The default is to have a file per variable per time step.
Maximum file Size	For Single File option, can specify the maximum file size. Continuation files are created if the file size would exceed this maximum.

Troubleshooting Saving Geometric Entities

Problem	Probable Causes	Solutions
A Part was not saved	User attempted to save an unsupported Part type.	Select only Model, Isosurface, 2D-Clip, and Elevated Surface Parts.
Variable(s) not saved	The variable was not activated or the variable was a constant.	Activate all scalar and vector variables you want saved.
Error saving	File prefix indicates a directory that is not writable or disk is out of space.	Re-specify a writable directory and valid prefix name. Remove unneeded files.

(see How To Save Geometric Entities)

2.9 Saving and Restoring View States

EnSight's viewports provide a great deal of flexibility in how objects are displayed in the Graphics Window. Given the complicated transformations that can be performed, it is imperative that users be able to save and restore accumulated viewport transforms.

View saving and restoring is accessed from the Transformations dialog.

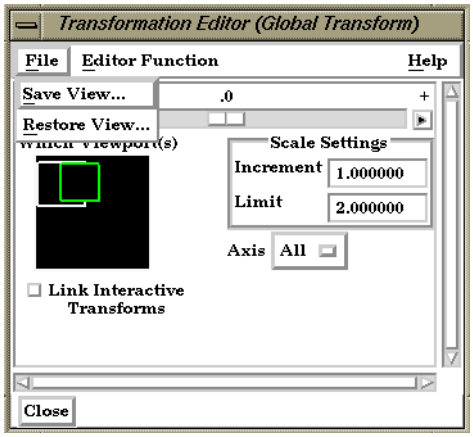


Figure 2-23
View Saving and Restoring in Transformation Dialog

```
Access: Desktop > Transformation Edit... > File
```

When either the Save View... or Restore View... selection is made, the user is presented with the typical File Selection dialog from which the save or restore can be accomplished.

(see also How To Save and Restore Viewing Parameters)

2.10 Saving and Printing Graphic Images

EnSight enables you to save an image of the Main View to a disk file or send it directly to a printer. The choice of save file formats depends on the implementation, but in all cases it is possible to obtain formats compatible with printers and plotters. Currently Apple PICT, PCL, PostScript, SGI RGB, JPEG, and TARGA formats are available.

EnSight also enables you to save images of an animation to disk files. These files can then be converted and printed or recorded to video equipment (see Section 7.15, Keyframe Animation).

Print/Save Image dialog

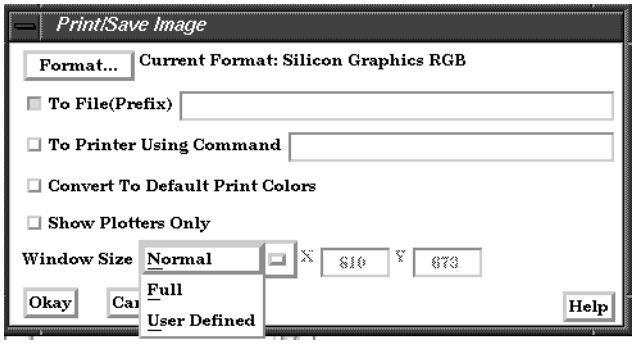


Figure 2-24
Print/Save Image dialog

You use the Print/Save Image dialog to specify the format and destination of an image to save. The destination can be a disk file or a printer. You also access the Image Format Options dialog for the various types from this dialog.

`Access: Main Menu > File > Print/Save Image...`

Format...	Click to select image format. (See next figure)
To File Toggle/Field	The image will be saved to this disk file name if toggle is on. This is a filename prefix. An appropriate suffix, according to the file format chosen, will be added.
To Printer Using Command Toggle/ Field	The command to send a file to the printer if toggle is on
Convert to default print colors	Clicking this toggle on will convert all black to white and all white to black but will leave all other colors as they are.
Show Plotters Only	Clicking this toggle will cause the graphics window to only display plotters.
Window Size	Specifies the size of the Graphics Window and the resulting image size.
Normal	Creates a window which is the size of the current Graphics Window.
Full	Creates a window which is the size of the full screen.
User Defined	Creates a window which is specified in terms of its width and height in the X and Y fields.

Format... .

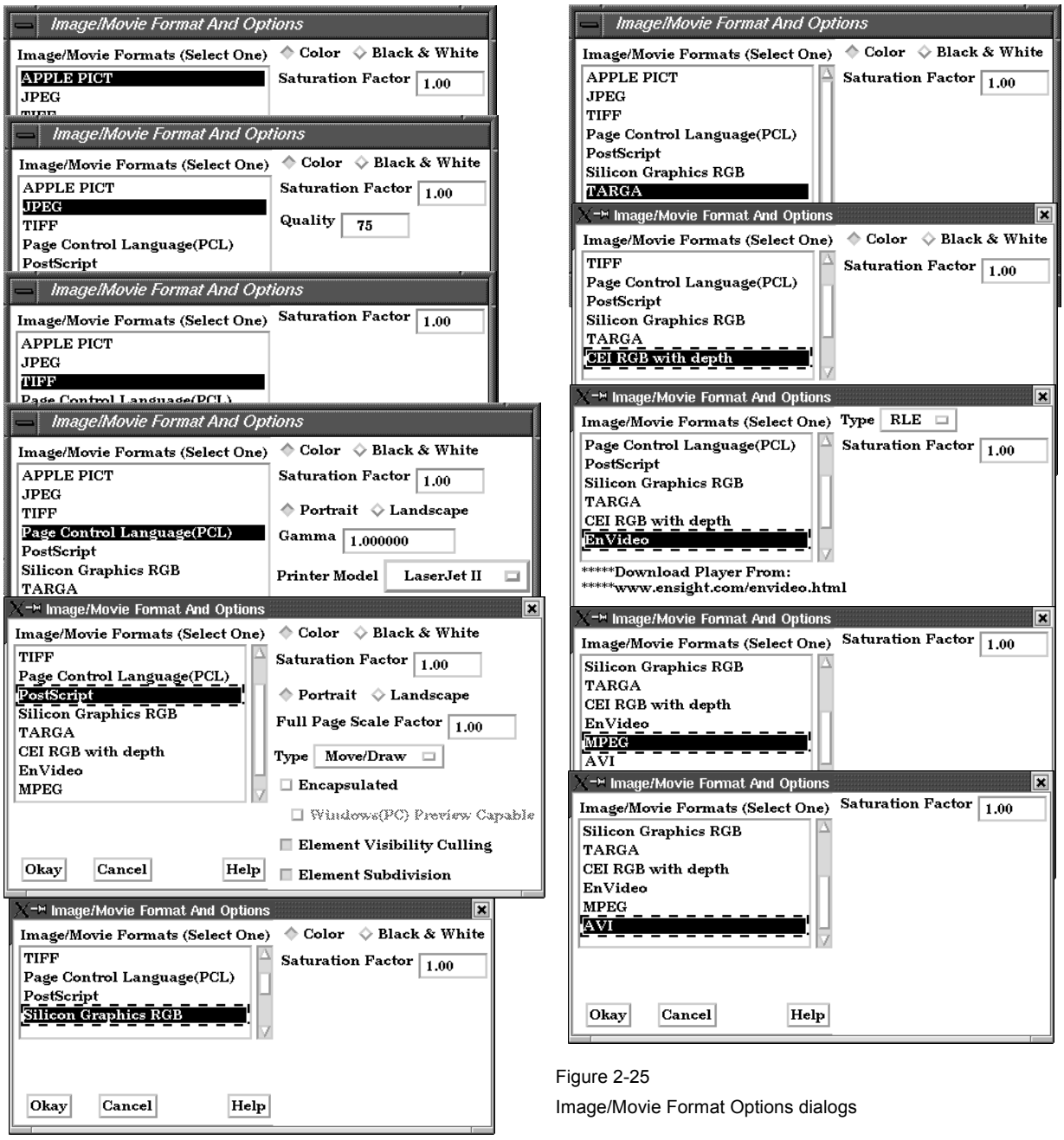


Figure 2-25
Image/Movie Format Options dialogs

Color/Black & White (Several Formats)	Color versus Black and White toggle.
Saturation Factor (all formats)	At a value of 1.0, no change to the image. At lower values, a proportionate amount of white is added to each pixel. At a value of 0.0, the image would be all white.
Quality (JPEG)	Specifies trade-off between fidelity & compression. 100 max fidelity; 0 max compression.
Portrait/Landscape (PCL, Postscript)	Page Orientation for printing.
Gamma (PCL)	Gamma correction factor.

Full Page Scale Factor (Postscript)	The percentage of full page scaling to do. This is according to Orientation as well. Values are from 0.0 to 1.0.
Printer Model (PCL)	The destination PCL printer model.
Type (Postscript)	Type of Postscript output: Move/Draw (vector) or Image Pixels. If type is Image Pixels, shaded 3D objects will be output as pixels while overlay graphics (annotation text, plots, color legends) will be output Move/Draw for higher print quality.
Type (EnVideo)	Type of EnVideo output: Run Length Encoded or Jpeg.
Encapsulated Toggle (Postscript)	Generate Encapsulated PostScript (EPS) for importing into other applications. (The graphic typically will appear as a gray box in the importing application on all systems unless the Windows(PC) Preview Capable toggle is also On).
Windows(PC) Preview Capable Toggle)	Create an Encapsulated PostScript (EPS) file which also has a preview image for use in Windows® applications. (The graphic will still appear as a gray box in the importing application on Macintosh systems).
Element Visibility Culling Toggle (Postscript)	Hidden geometry will be removed from the output stream if toggle is on. Valid for Move/Draw output only. On by default.
Element Subdivision (Postscript)	Subdivide output primitives (lines and polygons) if toggle is on. Although the output file will be larger, the color distribution will be far superior.Valid for Move/Draw output only. On by default.

Troubleshooting Saving an Image

Problem	Probable Causes	Solutions
Image has blotches or ghosts of other windows in it	A viewport or menu was popped in front of the Main Graphics Window as the image was being saved.	Do not perform any window manager functions until image is finished recording to disk file.
Error while saving image file	Directory or file specified is not writable	Rename the file or change the permissions.
	Ran out of disk space	Check the file system you are writing to with the “df” command then remove any unnecessary files to free up disk space.
	Image format not selected	Select an image format before saving.
Image looks bad when printed	Original on-screen image has low resolution	Make the graphics window as large as possible before saving the image to increase the number of RGB pixels used on the display.
	Image has been dithered during processing	Do not enlarge or reduce the image until it is in your word processor.
	Non-integral ratio of printer resolution to image resolution at final size	The image is a pixel-map image. For best results, the number of printer-dots per image-dot should be an integer. For example, if the original image resolution is 72 dpi, reduced to 48% the final-size resolution is 72/.48 = 150 dpi. On a 600 dpi printer, each image pixel is exactly 4 printer-dots on a side.
Move/Draw PostScript output doesn’t look correct.	Primitives in Move/Draw PostScript output sometimes suffer from sorting problems. (This will be fixed in a subsequent release.)	Use Image Pixel type instead of Move/Draw.

(see How To Print/Save an Image)

2.11 Saving and Loading XY Plot Data

The xy data used for curves in EnSight's plotter can be saved to a file for future re-loading into EnSight or for use in other plotting packages.

The process is described below as well as in Section 7.11, Query/Plot

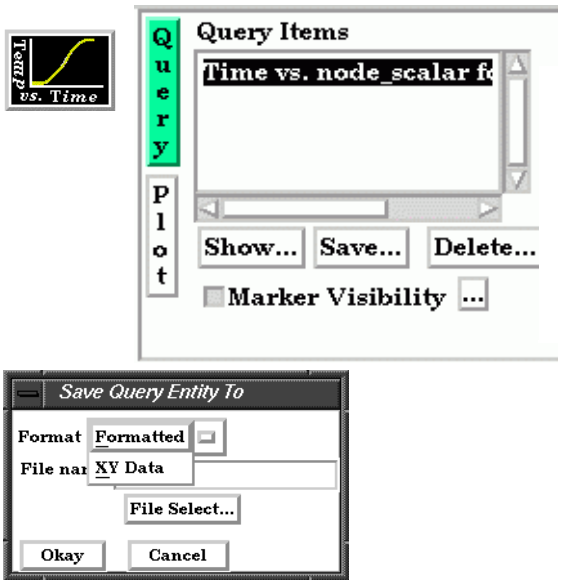


Figure 2-26
Saving or Loading XY Plot Data

`Access: Desktop > Query/Plot Feature`

Once the desired query item (curve) is selected in the list, the user can perform a Save operation by:

Save... Click this button in the Quick Interaction area of the Query/Plot feature to save the plotter curve data.

Format Select the Format of the data to save.

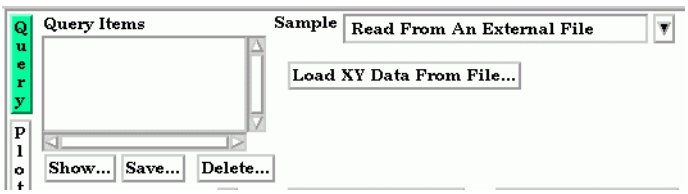
- ***Formatted*** is a table suitable for printing. (see Section 2.13, Saving Query Text Information)
- ***XY Data*** is the xy file format described in Section 11.10, XY Plot Data Format, which is suitable for re-loading into EnSight.

File Name Enter the desired filename for the xy data file, or click *File Select...* to be presented with the typical File Section dialog from which to perform the op8888eration.

Loading:

To Load a previously saved or externally generated xy data file (see Section 11.10, XY Plot Data Format) into EnSight you choose the "Read From An External File" Sample type (see How To Query/Plot). You will then be presented with the typical File Selection dialog from which to select the file. **Note: A MSC/Dytran .ths file is also a valid entry for this option.**

Figure 2-27
Query/Plot Area for Read From An External File.



2.12 Saving and Restoring Animation Frames

Both Flipbook and Keyframe Animation processes have save and restore capability. These are best described in the chapters devoted specifically to these features.

For Flipbook Animations, see Section 7.14, Flipbook Animation and How To Create a Flipbook Animation.

For Keyrame Animations, see Section 7.15, Keyframe Animation and How To Create a Keyframe Animation.

2.13 Saving Query Text Information

The data used for curves in EnSight's plotter and any other information from a query or otherwise which is presented in the EnSight Message Window can be saved to a file suitable for printing.

From Query/Plot Save... Formatted

One place this can occur is in the Query/Plot Quick Interaction area as described below as well as in Section 7.11, Query/Plot

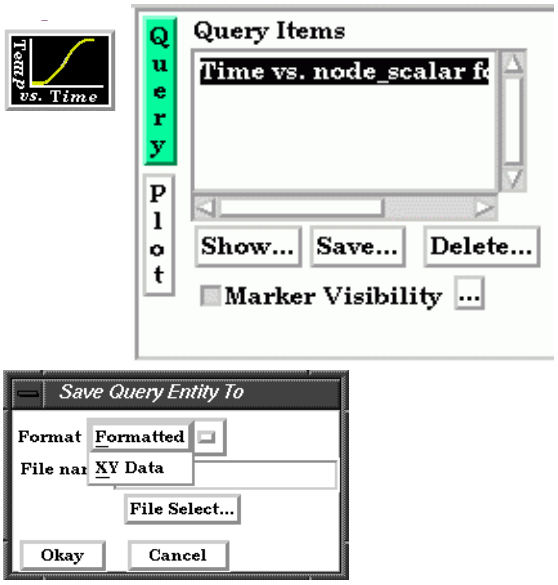


Figure 2-28
Saving or Loading XY Plot Data

```
Access: Desktop > Query/Plot Feature
```

Once the desired query item (curve) is selected in the list, the user can perform a Save operation by:

Save... Click this button in the Quick Interaction area of the Query/Plot feature to save the plotter curve data.

Format Select the Format of the data to save.

- *Formatted* is a table suitable for printing.
- *XY Data* is the xy file format described in Section 11.10, XY Plot Data Format, which is suitable for re-loading into EnSight.

File Name Enter the desired filename for the xy data file, or click *File Select...* to be presented with the typical File Section dialog from which to perform the operation.

From Query/Plot Show Text

Show ... Click this button to see the plotter curve information presented in the EnSight Message Window.

From EnSight Message Window

A file suitable for printing can be saved from any operation which places its information into the EnSight Message Window, such as Show Information queries and the Query/Plot Show Text... button described previously.

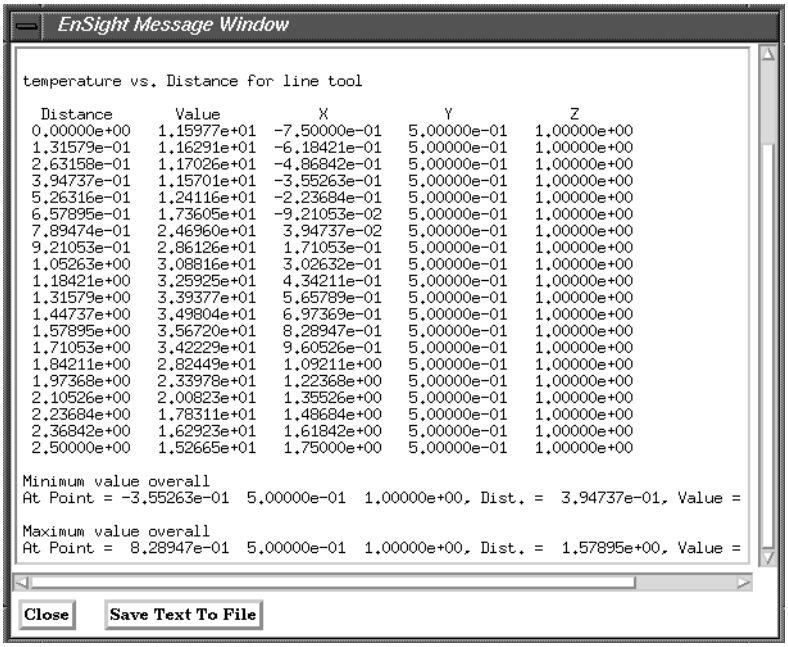


Figure 2-29

EnSight Message Window with Save Text To File Button

Save Text To File Brings up the typical File Selection dialog from which the information can be saved in the file of your choice.

2.14 Saving Your EnSight Environment

Every user has different postprocessing needs and personal preferences for how the EnSight windows should be positioned and sized. EnSight allows you to save dialog expandable section settings, and dialog size and position information to a file called “ensight7.winpos.default”. EnSight looks for this file at start up (in the current Client directory and if not there in the .ensight7 directory of the user’s home directory) and will bring the user interface dialogs up according to your saved settings (if the file is found).

Almost all major dialog windows are saved in the:

ensight7.winpos.default_XRESxYRES

file (where XRES and YRES are the resolution of the monitor when the preferences were saved). The only exception are minor prompt dialogs. There are also some dialogs for which you cannot save the size (such as the Tool Positions dialog).

The ensight7.winpos.default file also contains the size and location for all of the windows containing graphics.

A number of other settings, such as mouse and keyboard buttons and Icon Bar settings can also be saved to a user preferences file.

(see Preferences... in Section 6.2, Edit Menu Functions and How to Save GUI Settings)

2.15 Parallel Rendering Setup

EnSight Gold 7.4 supports general parallel rendering for increased performance, increased display resolution, and arbitrary screen orientations. This section describes the configuration file format and command-line parameters required for parallel rendering. Using hardware accelerated rendering, the features described are supported on SGI Irix 6.5, Sun Solaris 8, and HP-UX 11.0 at this time. Using software rendering (`ensight7 -X`) these features are supported on all platforms for which CEI supports threads.

In order to make use of parallel rendering with EnSight, the user must create a configuration file. This file is specified on the command line using the argument "`-dconfig <file>`". If `<file>` is not given as a full path, EnSight will check for the file in the directories `~/.ensight7/dconfig` and `$CEI_HOME/ensight74/site_preferences/dconfig` for user-level and site-level configurations, respectively.

There are two logical displays which can be configured in EnSight. The "GUI display" is always active, and consists of the main rendering window embedded in the user-interface. The "detached display" is external to the user-interface, and may consist of 1-16 smaller regions configured to form a large continuous display. The configuration file contains information about both the GUI display and the detached display, as well as tracking calibration information and options for using 6 DOF (degree-of-freedom) input devices. The next section will break down the format of a configuration file into distinct sections, followed by a section of example configurations. Sample configuration files can be found in the directory `$CEI_HOME/ensight74/user_defined_src/input/dconfig`.

CONFIGURATION FILE FORMAT

Configuration files are entirely text-based beginning with the line:

```
PRSd 2.0
# after the first line, anything following a '#' is a comment
```

The remainder of the file consists of one or more sections which describe how many graphics cards (referred to as 'pipes') to use and how they are to be configured.

Section: guidisplay

The default mode of EnSight uses only a single graphics pipe for rendering to the GUI display. When run on a multi-pipe X server (monster pipe), EnSight can be configured to use the additional pipes to accelerate the display through parallel rendering. The format of the configuration for the GUI display is as follows:

```
guidisplay
   worker <p1>
   worker <p2>
   ...
   worker <pn>
```

where:

```
  <pi> = an X display (i.e. b21:0.2)
```

Example:

```
guidisplay
   worker :0.1
   worker :0.2
   worker :0.3
```

In the example above, there is one X server (:0) which manages four graphics pipes. Note that the configuration file does not include the pipe to which the EnSight GUI is displayed. In this case, the GUI is assumed to be running on screen :0.0, and will be used as a worker pipe automatically.

For convenience, to instruct EnSight to use all graphics pipes which belong to the current X server you can use the command line option: "`-dconfig mpipe`". This will detect how many pipes are available and configure them appropriately.

Parallel software rendering is available on some platforms with the "`-X`" option. The same configuration file format is used in this case, although the displays themselves are not actually opened. For convenience, if the named file is not found and "`<file>`" is a number, this number will be interpreted as the number of parallel rendering workers. For example:

```
ensight7 -X -batch -dconfig 3 -p <cmdfile>
```

will run a batch session with 4 worker threads performing parallel rendering.

Section: wallresolution

There are two types of detached displays which can be configured. Flat, multi-screen displays can be configured by specifying the total resolution of the large display and configuring each screen as a subregion of the overall display. This section specifies the total resolution of the wall in pixels:

```
wallresolution
   <x-res> <y-res>
```

Section: view

For arbitrary screen positions and orientations, you will need to define a coordinate frame for your display environment. All of the positions and orientations that you give in the configuration file will be in this coordinate frame. Having chosen a frame, you can optionally add a '`view`' section to your configuration file in order to specify an initial transformation between your display frame and your desired viewing direction. If you are not using head-tracking this is a convenient way to specify an initial position for the viewer. The default is to position the viewer at the origin of your display frame with the orientation of the scene aligned with the orientation of your defined coordinate frame. You can change these defaults with the section:

```
view
   origin <x> <y> <z>
   zaxis  <x> <y> <z>
   yaxis  <x> <y> <z>
   center <x> <y <z>
   scale  <factor>
```

where '`zaxis`' and '`yaxis`' are unit vectors in your display coordinate frame. These parameters allow you to specify the orientation of your scene.

By default EnSight will center the scene at the center of the display environment, scaling the scene to fit within the region enclosed by the screens. The '`center`' and '`scale`' parameters allow the user to specify alternatives to these defaults. Specifying a scale factor of 1.0 may be useful if your display coordinates were designed to coincide with your model coordinates. This will allow you to view your models life-sized.

Section: tracker

EnSight supports tracking and input with 6DOF devices through a defined API. Libraries are provided to interface with trackd ((C) VRCO, Inc.) or the user may

write a custom interface to other devices or libraries. Which interface is used is determined by the ENSIGHT7_INPUT environment variable. For the trackd interface you will also need to set:

```
ENSIGHT7_TRACKER_KEY <num>
ENSIGHT7_CONTROLLER_KEY <num>
```

in order to specify the shared-memory keys for the input library to interact with trackd. You should be able to find these values in your trackd.conf configuration file. For information on connecting to other devices, please see the README file in

```
$CEI_HOME/ensight74/user_defined_src/input.
```

Once the EnSight client has been correctly interfaced to a tracking system you can add a section to the configuration file in order to calibrate the tracking with the display frame and customize the behavior of various interactions. The syntax for the section is:

```
tracker
   origin <x> <y> <z>
   zaxis  <x> <y> <z>
   yaxis  <x> <y> <z>
   headtracker <i>
   cursortracker <i>
   selectbutton <i>
   nextbutton <i>
   prevbutton <i>
   cancelbutton <i>
   rotatebutton <i>
   transbutton <i>
   zoombutton <i>
   transxval <i>
   transyval <i>
   transzval <i>
```

The ‘origin’, ‘zaxis’ and ‘yaxis’ parameters allow you to calibrate the tracker to your defined display coordinate frame. Many tracking libraries perform similar transformations, and you may omit these parameters if you have defined your display coordinate frame in terms of the native tracker coordinate frame. The ‘zaxis’ and ‘yaxis’ vectors need not be unit length. If your tracker coordinates are in inches but you find it more convenient to specify your display coordinate in centimeters, you might need use:

```
   zaxis 0.00 0.00 -2.54
   yaxis 0.00 2.54  0.00
```

The ‘headtracker’ and ‘cursortracker’ parameters allow you to specify which tracking device is tracking head position and which is tracking the controller. At this time only a single controller can be tracked in EnSight.

The remaining options allow you to customize the behavior of buttons and valuators on the input device.

The ‘select’, ‘next’, ‘prev’, and ‘cancel’ buttons are used for interacting with the 3D GUI items in the scene. Currently the 3D GUI includes a heads-up-macro (HUM) panel, a part list, and a part-value slider. Depending on the number of buttons that you have on your input device you may choose to interact with the 3D GUI in one of two ways.

Point and Click

The default mode is to define only a select button (default button 0). When the 3D GUI is visible, you can point at the 3D buttons and the item that you are pointing

at will be displayed in a highlight color. When you press the select button you will activate the current selection.

For the HUM panel, this means that you will activate the macro which is defined for the selected button. You will find example macro files and additional instructions in

`$CEI_HOME/ensight74/user_defined_src/input/README.`

Clicking on an item in the part list will select or unselect the item in the list. Combined with macros in the HUM, this will allow you to modify visibility or other attributes on a part or collection of parts. If there are many parts in the part list, you can also select the scrollbar and move the controller up and down to scroll through the list.

Similarly, the part-value slider can be used to modify part attributes for certain part types. For isosurfaces you can select the part slider and move left to right to change the isovalue.

Select and Navigate

If the '`next`' button is defined, interaction is performed in a different manner. It is often not easy to clearly point at a particular button and click without moving the controller too much. In this mode the '`select`' button will activate a menu for interaction. The '`next`' and '`previous`' buttons will scroll through the possible options, and pressing the '`select`' button again will activate a selection. The '`cancel`' button allows you to exit from the menu without selecting an option. If you only have two buttons, you can also hold down the '`next`' button and press '`select`' to cancel the operation.

Other Buttons

The '`rotatebutton`', '`transbutton`', and '`zoombutton`' allow you to link a button to each of these transformations. The '`xformbutton`' allows you to link a button to the current transformation mode, similar to the mouse button configurations for the main GUI interactions. You may want to add buttons on the heads-up-macro (HUM) panel to switch between modes.

Section: annot

Annotations in EnSight include the heads-up macro panel, text, lines, logos, legends, and plots. In the GUI display these items appear as an 'overlay' which is fixed in screen space. In a truly 3D display environment it is useful to be able to specify the locations of these objects. In EnSight 7.4, these items continue to occupy a plane in the 3D world. The user can specify the position and orientation of this plane, or allow it to float freely based on the head-tracked position of the user. To specify the position of the plane, use:

```
annot
   [
     center <x> <y> <z>
     zaxis  <x> <y> <z>
     yaxis  <x> <y> <z>
   ]
   OR
   [
     dist <float>
   ]
     xscale <float>
     yscale <float>
```

For example, to place the annotation plane at a fixed position and orientation in the world you might specify:

```
annot
   center  5 0 -5
   zaxis   1 0 0
   yaxis   0 1 0
   xscale  7.5
   yscale  7.5
```

Rather than specifying a fixed position, you may also allow the annotations to follow the viewer in a head-tracked environment:

```
annot
   dist    7.5
   xscale  7.5
   yscale  7.5
```

In this mode the annotation plane will be position 7.5 units from the viewer, with the zaxis and yaxis taken from the viewer orientation. Note that all coordinates are in the same units as the display environment.

The default mode is to fix the annotation plane on the first display screen found in the configuration file.

Section: stereo

This optional section specifies that the display should be initialized in stereo mode. This applies to the detached display pipes only, and consists of a single line:

```
stereo
```

Section: pipes

The remainder of the file specifies how many graphics pipes to use for the detached display and how they are arranged with respect to your defined space (either the '`wallresolution`' or the chosen display coordinate frame). The format of this section is:

```
numpipes
        <num>
pipe
   xserver <p1>
   resolution <x-res> <y-res>
   [ xorigin <xo> <yo> ]
   [ wallorigin
      <wall-x> <wall-y>
   ]
   [ bottomleft  <x> <y> <z>
     bottomright <x> <y> <z>
     topleft     <x> <y> <z>
   ]
   [ lefteye
       or
     righteye
   ]
   [ worker <p2>
     ...
     worker <pn> ]
```

[repeat '`pipe`' section (num-1) more times]

The '`numpipes`' parameter simply specifies how many separate regions will be configured. For each region, there will be a 'pipe' section which describes the position and orientation of the display surface. The '`xserver`' parameter specifies the X display (i.e. :0.1). The 'resolution' is the width and height of the window to

create on the given pipe. The 'xorigin' is an optional parameter to specify the origin of the window on the given pipe (default (0,0)).

If the detached display is configured as a flat display (wallresolution was used earlier), you will need to specify a 'wallorigin'. This parameter indicates the position of this pipe in the global display.

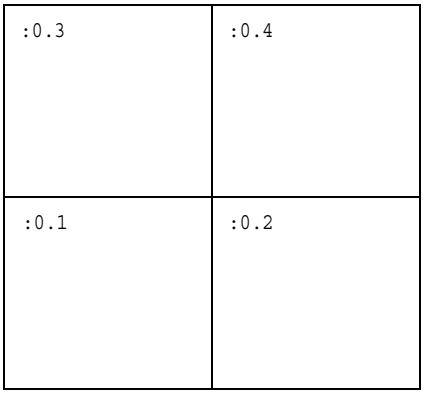
If the detached display is a more general display, you need to specify the positions of three of the corners in the display configuration frame. Note that 'bottom', 'top', 'left' and 'right' refer to the X and Y directions of the X screen. You may mount the projector upside-down or sideways, but the 'bottomleft' corner will always remain the same.

Passive stereo displays are supported by adding 'lefteye' and 'righteye' parameters for each pipe.

Finally, each region in a detached display can be rendered in parallel by adding additional 'worker' pipes, using a similar specification format as for the 'guidisplay' section.

Example 1

In this example there is one X server with multiple screens. The GUI is displayed on pipe :0.0, with the other four pipes used for the detached display. The projectors are configured in a 2x2 array:



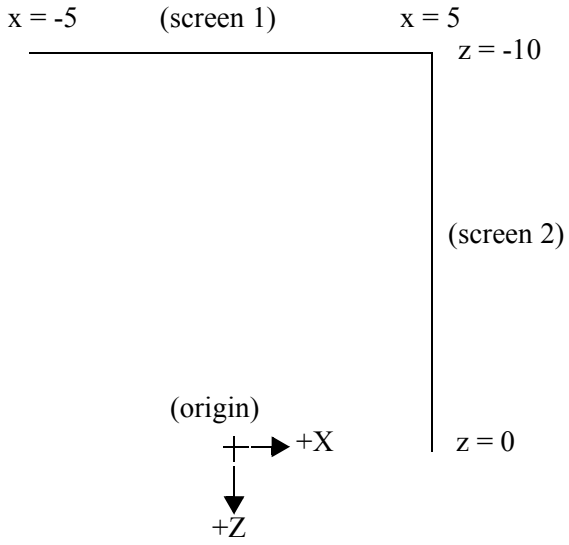
The configuration:

```
PRSd 2.0
#
# conference room display wall
#
wallresolution
   2560 2048
numpipes
   4
pipe # lower-left
   xserver :0.1
   resolution 1280 1024
   wallorigin    0    0
pipe # lower-right
   xserver :0.2
   resolution 1280 1024
   wallorigin 1280    0
pipe # upper-left
```

```
   xserver :0.3
   resolution 1280 1024
   wallorigin    0 1024
pipe # upper-right
   xserver :0.4
   resolution 1280 1024
   wallorigin 1280 1024
```

Example 2:

If the screens are in more general position and orientation it is necessary to define positions based on the corners of each screen. In this case you are free to choose a reference frame for describing your screens. In general it is less troublesome to use a coordinate system which aligns with the screens. In this abstract space, determine the locations of the bottom-left, bottom-right, and top-left corners of each screen. Consider an example with two walls at a 90 degree angle:



In this top-view illustration the y-axis is determined by the right-hand rule. The two walls are aligned with the X=5 and Z=-10 planes. Assuming a screen size of 1024x768 with square pixels, the proportional Y range (with square pixels) would be [-3.75, 3.75] (origin centered in both X and Y). This gives us a configuration file of:

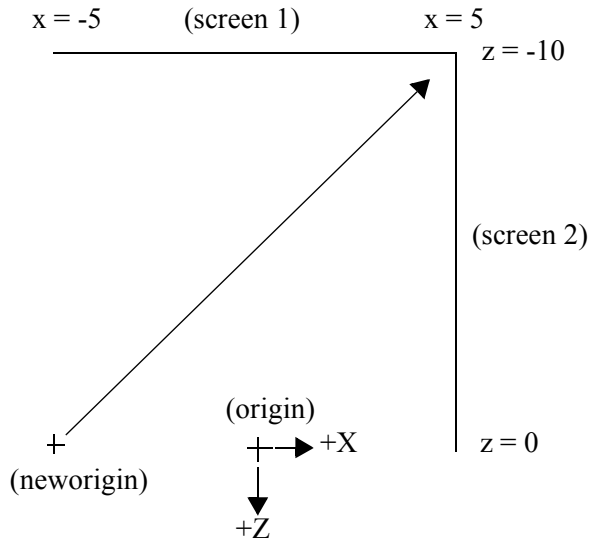
```
PRSd 2.0
numdisplays
   2
display
   xserver :0.2
   resolution 1024 768
   bottomleft -5 -3.75 -10
   bottomright 5 -3.75 -10
   topleft    -5  3.75 -10
display
   xserver :0.1
   resolution 1024 768
   bottomleft   5 -3.75 -10
   bottomright  5 -3.75   0
   topleft      5  3.75 -10
```

Example 3

The default viewing model of EnSight is to position the object along the -Z axis. One way to determine where your model is centered by default is to build your configuration file in the desired orientation. Because this can be cumbersome to modify, there is an optional section of the configuration file which allows you to specify an alternate coordinate frame. Once you have built a configuration file in any desired frame, you can add a section such as this:

```
view
   origin -5 0 0
   zaxis .707 0 -.707
   yaxis 0 1 0
```

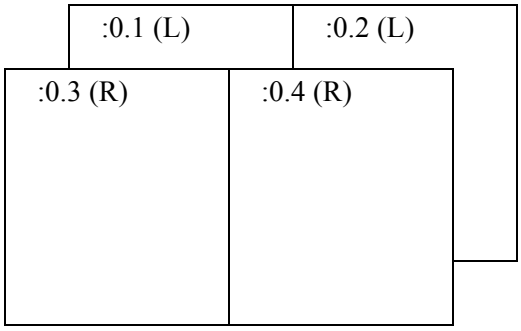
The default parameters are origin=(0,0,0), zaxis=(0,0,-1), and yaxis=(0,1,0). Adding this line to the above example we get as a result:



In the earlier example, the scene would originally center on the rear wall. With the transformed frame, the scene will center on the corner of the two walls. Using this optional feature is generally easier than computing the screen corners in the desired frame.

Example 4

Passive stereo displays achieve stereo by projecting overlapping polarized images from multiple projectors. This can be achieved using detached displays with a distinct rendering region for each screen and eye. Consider an example with 4 graphics pipes projecting to a 2-screen display wall:



```
PRSd 2.0
#
# passive stereo display (simulated on a single display)
#
wallresolution
   2560 1024
numpipes
   4
pipe # left-screen, left-eye
   xserver :0.1
   resolution 1280 1024
   wallorigin    0    0
   lefteye
pipe # right-screen, left-eye
   xserver :0.2
   resolution 1280 1024
   wallorigin 1280    0
   lefteye
pipe # left-screen, right-eye
   xserver :0.3
   resolution 1280 1024
   wallorigin    0    0
   righteye
pipe # right-screen, right-eye
   xserver :0.4
   resolution 1280 1024
   wallorigin 1280    0
   righteye
```

Note that the lefteye/righteye parameters are NOT necessary when using traditional quad-buffered stereo to drive the projectors. Some systems have a signal splitter which takes the frame-sequential stereo signal and generates separate signals for left and right eye. In this case a conventional configuration file without the “eye” designations will work fine. Passive stereo displays are always in stereo mode. If you would like to initialize a quad-buffered stereo display in stereo mode you can add the ‘stereo’ directive before defining your pipes.

Example 5

Note that it is fairly easy to test large displays and VR environments on a smaller system. The passive display above might be tested with a configuration file such as:

```
PRSd 2.0
#
# passive stereo display (simulated on a single display)
#
wallresolution
   512 256
numpipes
   4
pipe # left-screen, left-eye
   xserver :0.0
   resolution 256 256
   wallorigin   0   0
   xorigin      0   0
   lefteye
pipe # right-screen, left-eye
   xserver :0.0
   resolution 256 256
```

```
   wallorigin 256   0
   xorigin    256   0
   lefteye
pipe # left-screen, right-eye
   xserver :0.0
   resolution 256 256
   wallorigin   0   0
   xorigin      0 256
   righteye
pipe # right-screen, right-eye
   xserver :0.0
   resolution 256 256
   wallorigin 256   0
   xorigin    256 256
   righteye
```

The above configuration file simulates the four-pipe passive stereo example on a single-pipe machine, with each display represented by a 256x256 window. Note that these examples make use of the 'xorigin' parameters, which allow you to specify the origin location of each window on each X server.

Example 6

It is possible to combine multi-pipe and display wall rendering capabilities if you have enough graphics pipes. In the following example we have configured four graphics pipes for a two-screen display wall. Multiple pipes per screen will increase performance, in the same way that multiple pipes are used for the GUI window.

Configuration file:

```
PRSd 2.0
#
# display wall using 4 pipes
#
wallresolution
   2560 1024
numpipes
   2
pipe # left-screen
   xserver :0.1
   resolution 1280 1024
   wallorigin    0    0
   worker :0.3
pipe # right-screen
   xserver :0.2
   resolution 1280 1024
   wallorigin 1280    0
   worker :0.4
```