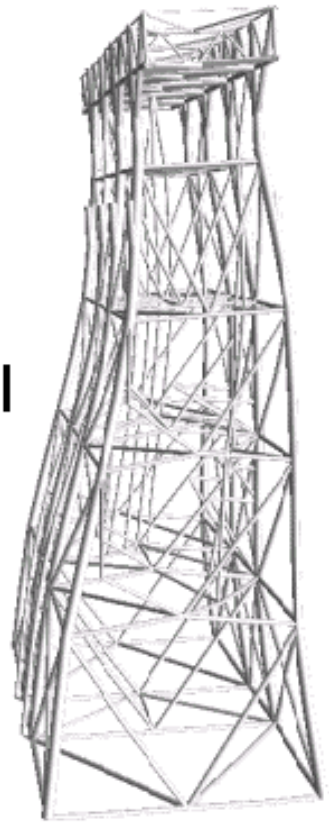
USFOS
Reality Engineering

USFOS Graphical User Interface



User's Manual

USFOS release 8-6

GUI Version 2.6

Table of Contents

1 Introduction

This document is the user manual for USFOS Graphical User Interface, GUI, (file name *xact.exe*). The document describes typical use of the tool, which is a mainly a post processor for USFOS, but also offers help in connection with analysis setup and run. The manual also explains how to navigate in the graphical user interface, what functionality the different buttons and menus contain. It also describes how to use the different dialogs that the GUI presents to the user.

The manual is divided into two main chapters:

- Tutorial : How to utilize the tool?
- Graphical User Interface : Detailed description of the interface

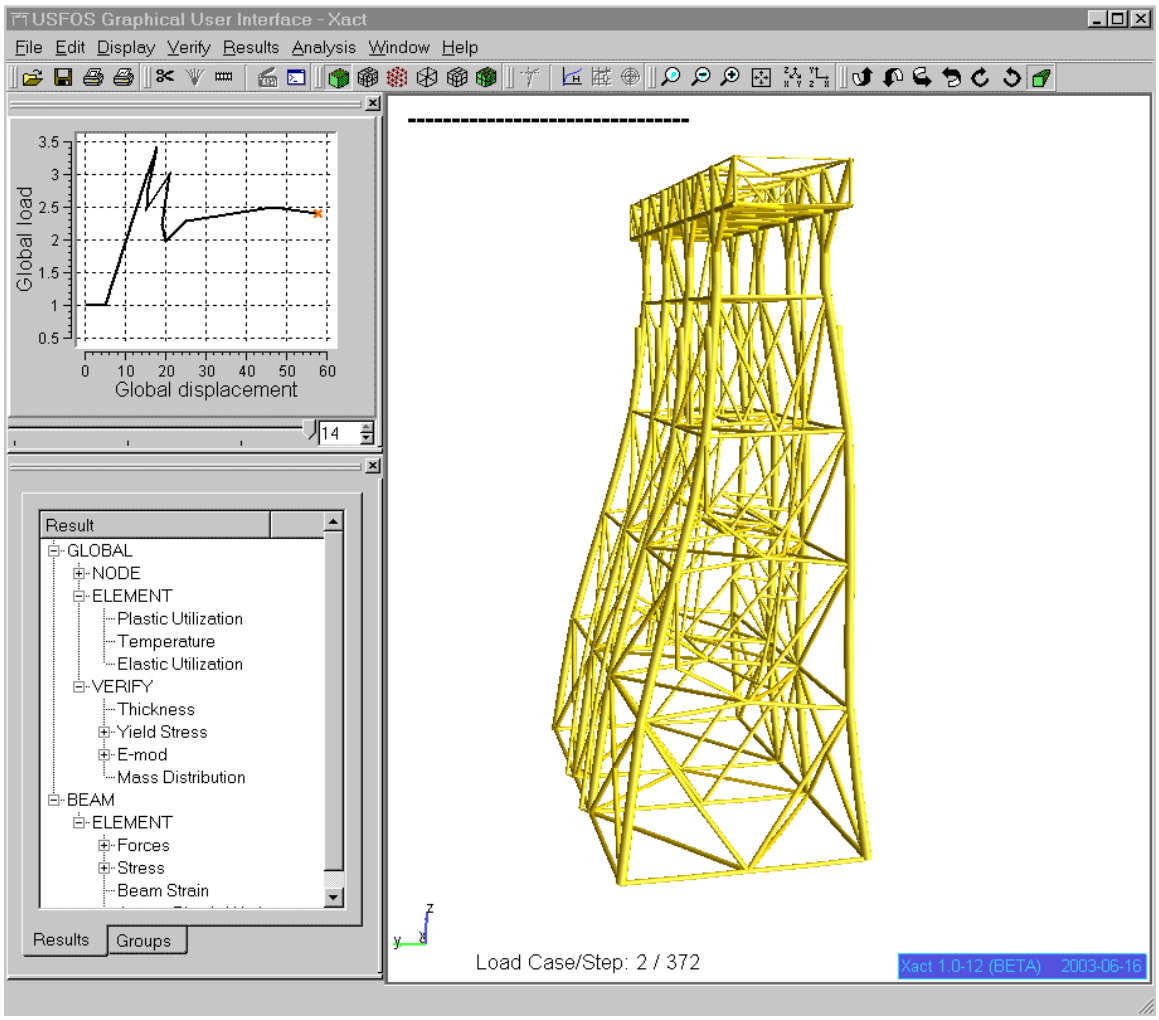


Figure 2.1-1 Welcome to USFOS Graphical User Interface

2 Getting Started

2.1 Introduction

THE GUI replaces the previous USFOS post processor, (XFOS), and could be used similar. The GUI is compatible with older USFOS versions, (the current USFOS version is 8-2), and will normally handle cases where the simulation "engine" (usfos.exe) and the GUI are running on different computer platforms (f ex USFOS running on UNIX and XACT on Windows). The GUI is available on both Windows (NT, 2000 and XP) and Linux (RedHat 7.2). The "internal name" of the GUI is "*xact*", (hence the *xact.exe*), and this name is referred to (for convenience) in some of the following examples.

2.2 Activating USFOS – Graphical User Interface on Windows Computers

Both USFOS and XACT, (as most codes), could be activated in several ways on windows computers, from typing commands to "double clicking". (The "best" method is the one preferred by the user).

Starting the Graphical User Interface XACT:

- "DoubleClick" on shortcut or from the Start/Programs menu

- "Double click" on ***raf*** file from Explorer.
- Type ***xact*** from a command window

Starting the analysis module: USFOS:

- Run the analysis module, USFOS from XACT
- Type ***usfos*** from a command window
- Execute USFOS from script file(s)

It should be noted that the ***default directory*** in XACT (the directory where XACT will search for and save files) is depending on start method, see Table 2.2-1:

Start Method	*Default Directory*
Shortcut/start menu	USFOS example folder, (typical C:\Program Files\USFOS\examples)
From Explorer	The directory where the raf file is located.
Typing from command window	"Current directory" of the actual command window, (f ex where the USFOS files are located).

Table 2.2-1 Default Directory vs. Start Method on Windows

In order to avoid tedious "browsing" to the correct directory, it is recommended to start both USFOS and XACT from the directory where the USFOS files are located.

2.3 Activating USFOS – XACT on Linux computers

On UNIX (Linux is a UNIX dialect), programs are executed by typing it's name in a command window, (the PATH variable is updated during installation).

Startingteh Graphicla User Interface XACT:

- Type ***xact*** from a command window

Startin the naalysis module: USFOS:

- Run USFOS from XACT
- Type ***usfos*** from a command window
- Execute USFOS from script file(s)

The ***default directory*** in XACT (the directory where XACT will search for and save files) is "Current directory" of the actual command window, (fex where the USFOS files are located).

In order to avoid tedious "browsing" to the correct directory, it is recommended to start both USFOS and XACT from the directory where the USFOS files are located.

3 Tutorial

3.1 Introduction

Typical use of USFOS-XACT will be described in the following without explaining how to operate the different menus/buttons, (Description of the interface is given in Graphical User Interface section).

The examples should give the user a brief overview of the possibilities, and does not cover all available functions. It is therefore recommended to take a look in the Graphical User Interface section of this manual.

USFOS-XACT has identical behavior on Windows and Linux.[1]

3.2 What is XACT?

XACT is the graphical User interface for USFOS with following main functions:

- Preparing USFOS analysis input
- Starting USFOS analysis
- Inspecting USFOS analysis results

3.3 Preparing USFOS input Files

3.3.1 Introduction

XACT offers input preparation aids for USFOS:

- Visualization of the structural model in advance of the analysis
- "Wizard" for first time users for creation of a minimum input file
- Specialized USFOS editor, (context sensitive)
- Communication between the 3D graphical image and the USFOS editor in connection with selection of nodes and elements.
- Testing the input by running directly from the Analysis Control Center.

[1] This is not the case for USFOS release 8-3/ USFOS GUI version 2.3, where USFOS GUI version 2.3 is available for Windows only

3.3.2 Opening of Analysis Model

If the complete structural model is collected in *one* file, it could be opened and displayed prior to the analysis. Node- and element numbers are available in the 3D image, which will ease the further USFOS model setup.

Use "Open USFOS Model File" located under the File menu, see Figure 3.3-1. The File open dialogue box appears (Figure 3.3-2), and files with extension ***.fem*** will be displayed. In this demo example, the file *t1.fem* is selected.

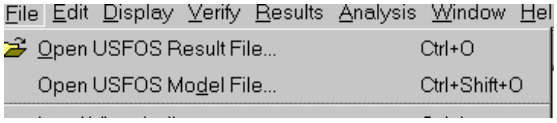


Figure 3.3-1 Open USFOS Model File

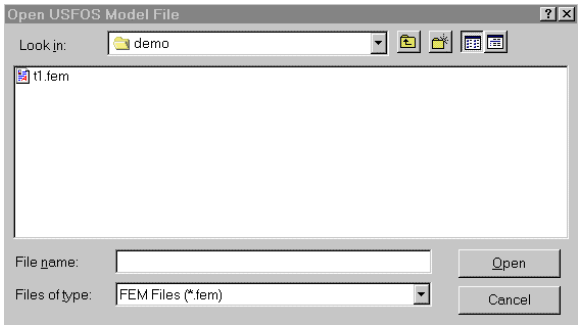


Figure 3.3-2 Open Model File Dialogue box

If the model file is complete, (and correct), the 3D image of the model will appear without any results available, see Figure 3.3-3. In this mode, following options are available:

- Hide/Show Groups (ref to *set* in SESAM)
- Node and Element number visualization, (On/off switches under Verify)
- Verification of Node coordinates and boundary conditions (Verify/Verification)

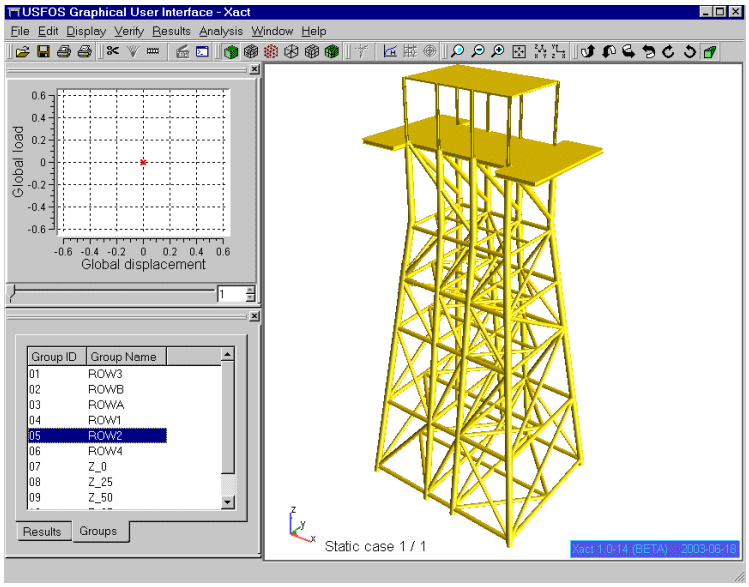


Figure 3.3-3 Successfully Opening of Model File

3.3.3 Analysis Setup, (Wizard).

When an input is modeled from scratch, the "Wizard" could give some help in connection with defining the mandatory (minimum) input for the analysis. Under *Analysis* on the menu bar, the "USFOS Analysis Setup" is selected.

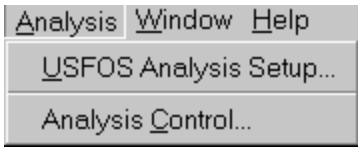


Figure 3.3-4 Analysis Setup / Analysis Control

Three different "sheets" are available:

- Static analysis setup
- Cyclic analysis setup
- Dynamic analysis setup

The different choices are shown on the tabs on the upper left corner of the setup sheet, (see Figure 3.3-5 for static analysis setup sheet and **Figure 3.3-6** for dynamic analysis setup).

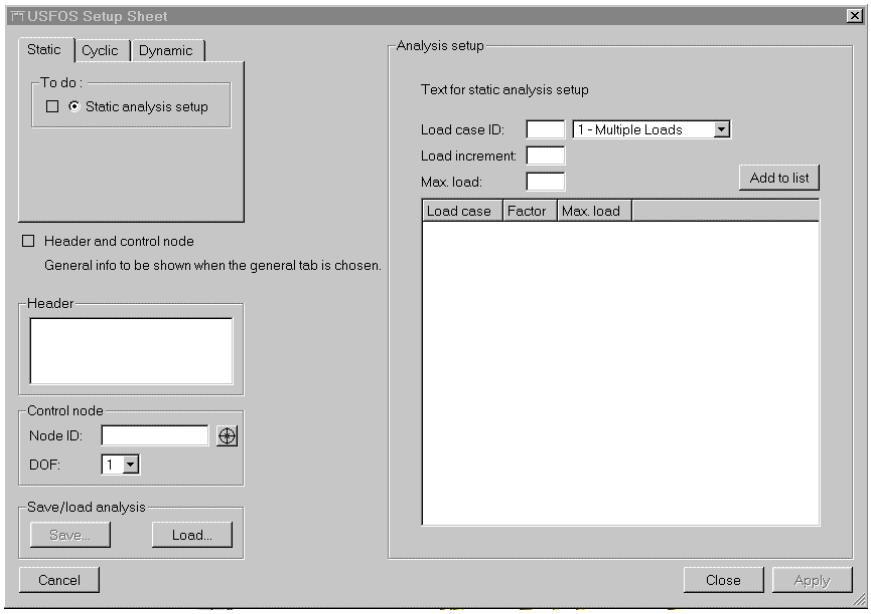


Figure 3.3-5 Setup sheet for static analysis. Empty

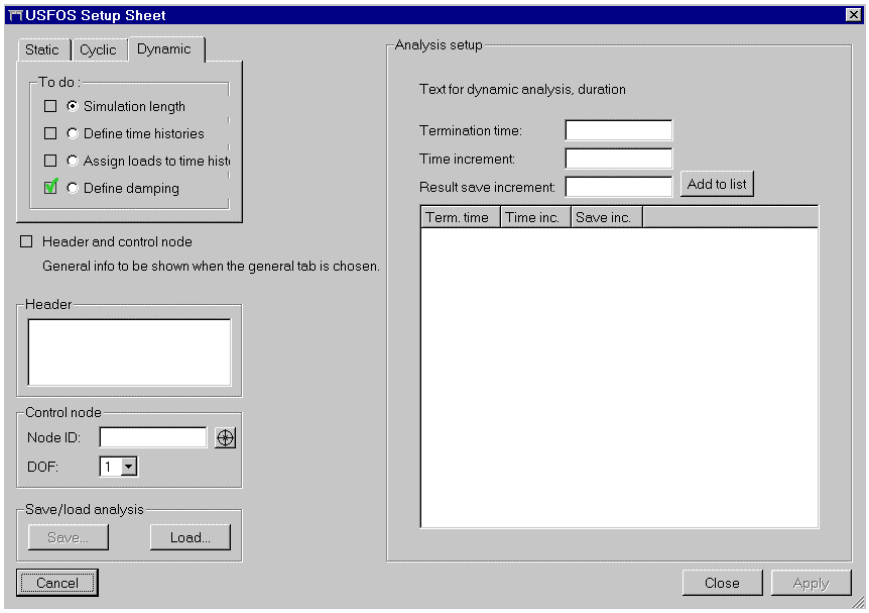


Figure 3.3-6 Setup sheet for dynamic analysis. Empty

The use of the Wizard is demonstrated in the following:

Header:
Up to three lines of text to identify the analysis. The heading will appear on the result output.

Control Node:
Selection of a Node/Dof to monitor during the analysis. This displacement is the "Global Displacement", and is the X-axis in the global Force-Displacement plot, which appears automatically when a USFOS result file is opened.

The node is selected either by typing in the Node ID or by pointing on the 3D Image of the structure. (Press the target button, and use ctrl+left mouse button to select). The actual degree of freedom, (typically the direction of the main loading), to monitor has to be specified.

Loading sequence:
The actual load cases to apply, (in a specific sequence) are defined by selection of load case ID, scaling factor per step and maximum load. Available load cases, (found on the model file), are listed.

Select load case, type in load increment, max load and add to list.

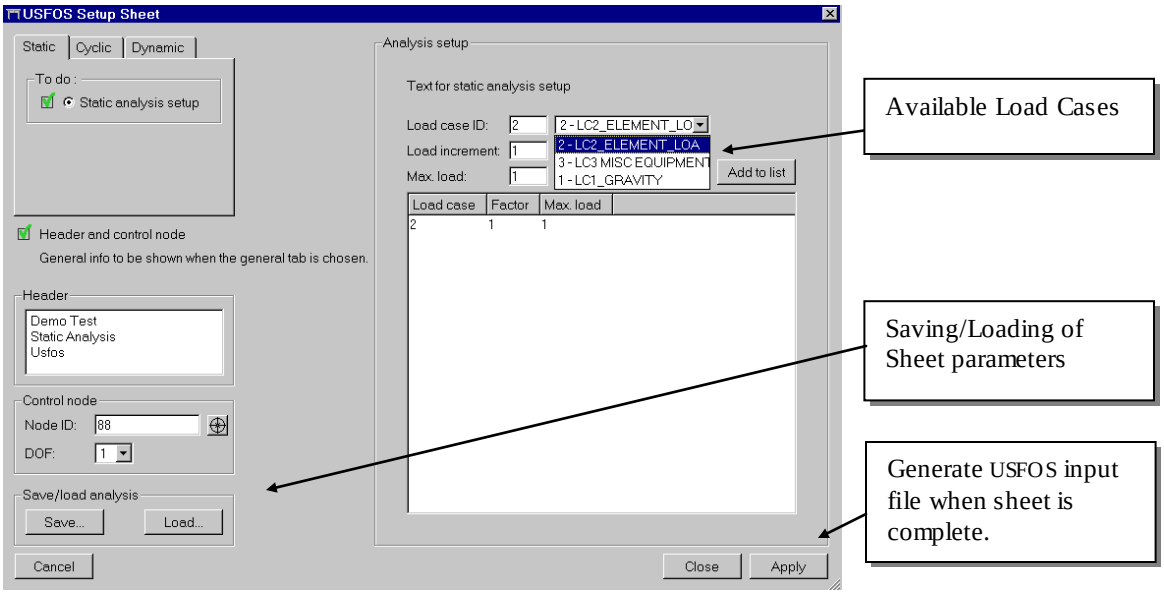


Figure 3.3-7 Setup sheet for static analysis. Completed

It is possible to save the sheet parameters, and the file extension is ***.uss***, (**US**FOS **S**etup file), using the Save button, see Figure 3.3-8. This setup file could be loaded and for example used as a starting point for later analyses, (Figure 3.3-9).

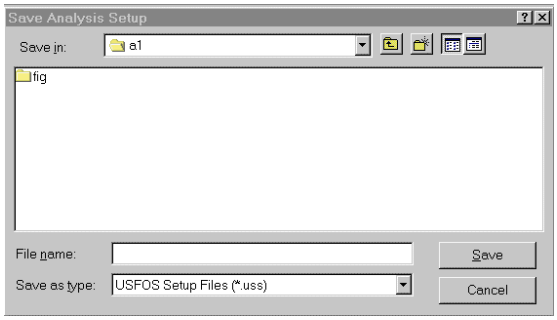


Figure 3.3-8 Saving of Setup Sheet Parameters

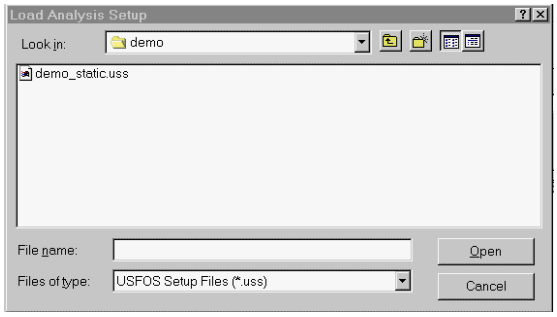


Figure 3.3-9 Loading parameters for Setup sheet

As the different items are correctly filled in, the actual check box gets a green tag. When the sheet is completely filled in, the apply button becomes active.

When the APPLY button is activated, the Sheet parameters are translated to USFOS commands, and the commands appear in the control file window of the "Analysis Control" dialogue box, (see 3.3.4).

3.3.4 Analysis Control, (Editor and Control Center).

With the automatically created USFOS control file, Figure 3.3-10, the model is ready for the analysis, (just press the "run" button). Output from the analysis module appears in the lower window, Figure 3.3-11.

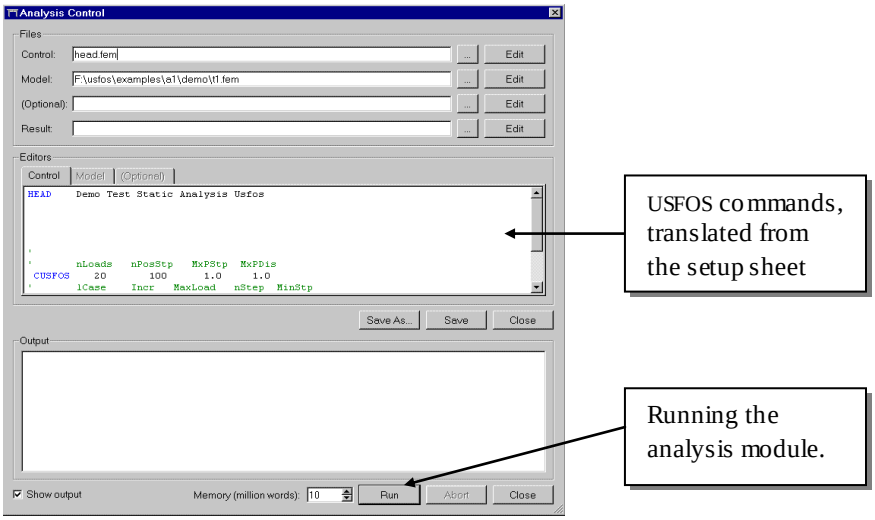


Figure 3.3-10 Analysis Control automatically filled in by the Wizard

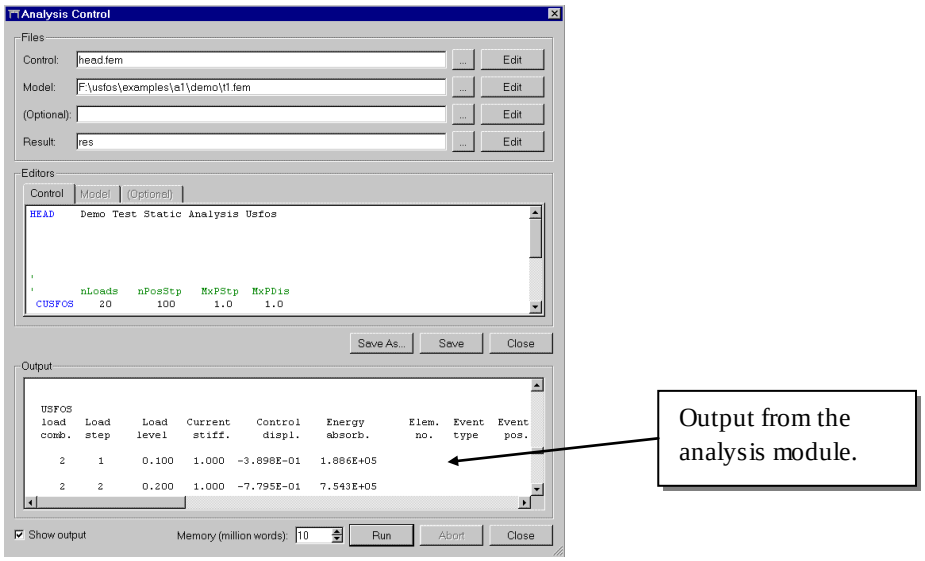


Figure 3.3-11 Running USFOS from the Analysis Control

Further commands to USFOS have to be written 100% manually in the "USFOS editor", (other text editors could also be used), or semi automatic utilizing the command menu selection. One example of use of the latter is given below:

By pressing the function key ***F2***, the main command headlines appear, Figure 3.3-12. In this example, the "Joint Modeling" is selected, and all available USFOS commands for joints appear, Figure 3.3-13. The *Joint Capacity Check* is selected, and the actual USFOS command is automatically inserted in the control file (Figure 3.3-14) ready for specification of the actual joint (node ID) and the two chord members. The parameters are either typed in directly or selected from the 3D image by just pointing, (ctrl + left mouse button), at the node and elements, see **Figure 3.3-15**. Note that by pressing the ***F1*** key when the cursor is in a line starting with an USFOS command, the parameter description is inserted as a comment line above the line containing the command.

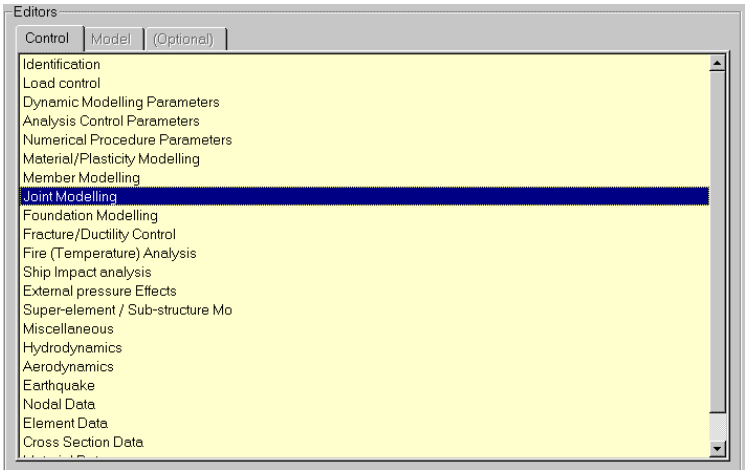


Figure 3.3-12 Adding USFOS commands. Selection from the command menu, (F2 key)

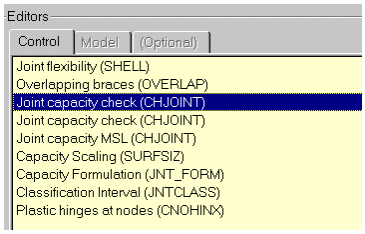


Figure 3.3-13 Selection of Joint Modeling Command: CHJOINT

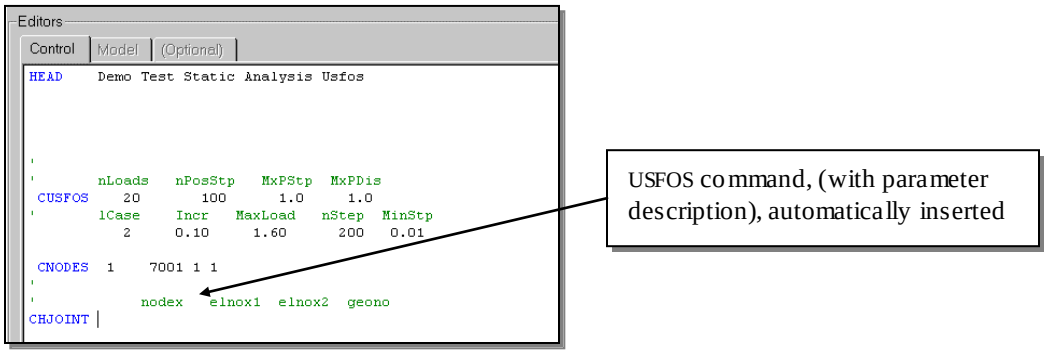


Figure 3.3-14 Automatic inserting of the CHJOINT Command in the USFOS editor.

Figure 3.3-15 shows an image of the structure with node- and element ID's applied and how the ID's are automatically filled into the control file.

If USFOS is analyzed with this joint option applied, restriction on load transfer between brace and chord is set automatically. **Figure 3.3-16** shows the model with the joint transfer elements automatically inserted, and the utilization of the different components (image to the right). In this case, the red color of the joint transfer elements indicates that the connection is weaker than the brace itself for the actual loading situation.

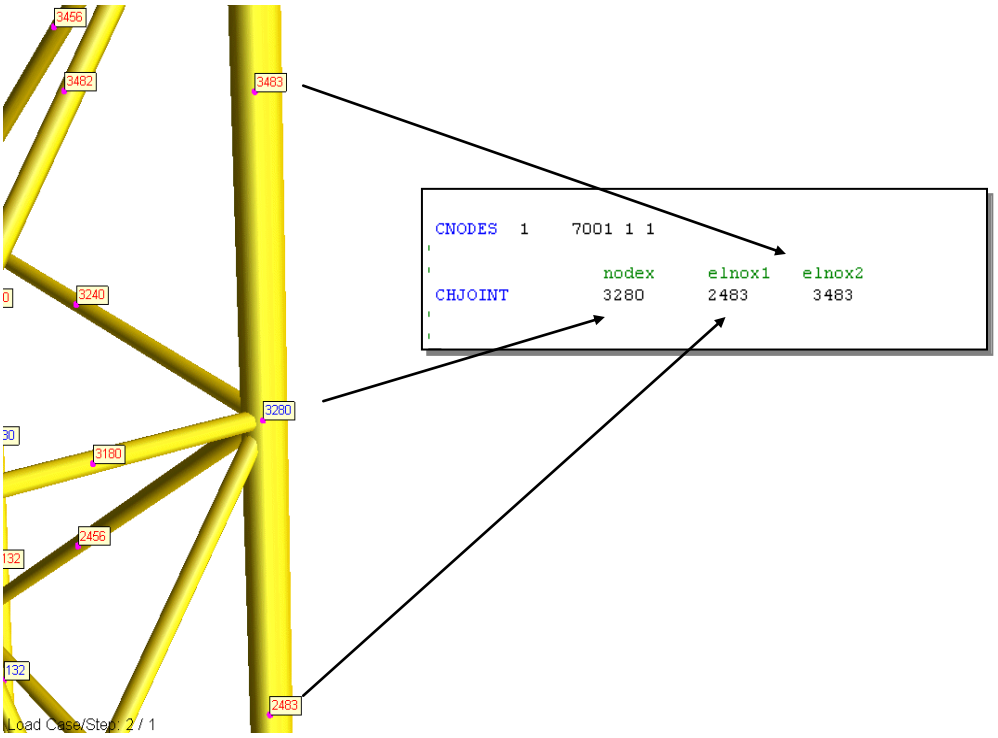


Figure 3.3-15 Selecting node and element ID's by pointing on the 3D image.

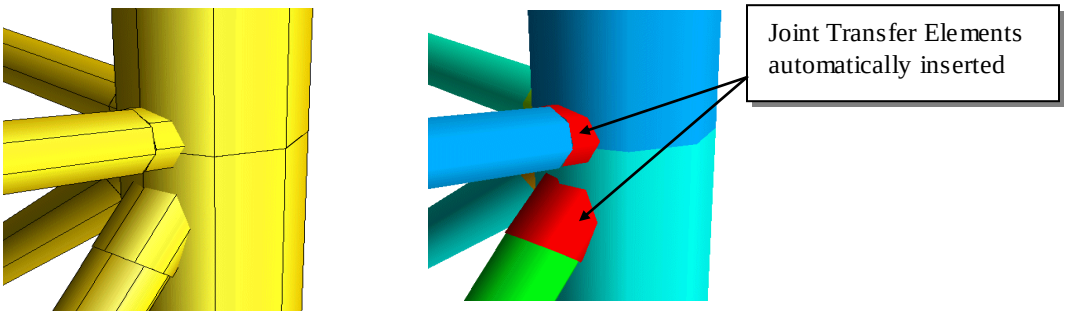


Figure 3.3-16 USFOS analysis with Joint Resistance Limitation applied.

3.4 Post Processor

In the examples below, it is assumed that USFOS has been run, and that USFOS result file (the "***raf***" file) exists.

3.4.1 Checking USFOS analysis model

In order to verify that the analysis model is correct, it is recommended to inspect the properties graphically. In Figure 3.4-1, the available verification items are shown in the results dialogue box, and in the example, the marine growth is visualized with colors. Similar images are available for:

- Wall thickness (see Figure 3.4-2, left)
- Yield Stress (both initial and eventually degraded)
- E-mod (both initial and eventually degraded)
- Mass distribution (Density of material multiplied by wall thickness)
- Hydrodynamic Drag- and Mass Coefficients
- Flooded Elements (Flooded members are red, see Figure 3.4-2, right)
- Buoyancy Diameter (Flooded members become blue, and the air filled member with largest diameter become red. If history dependent buoyancy is used, the buoyancy at current load step (time) is visualized)

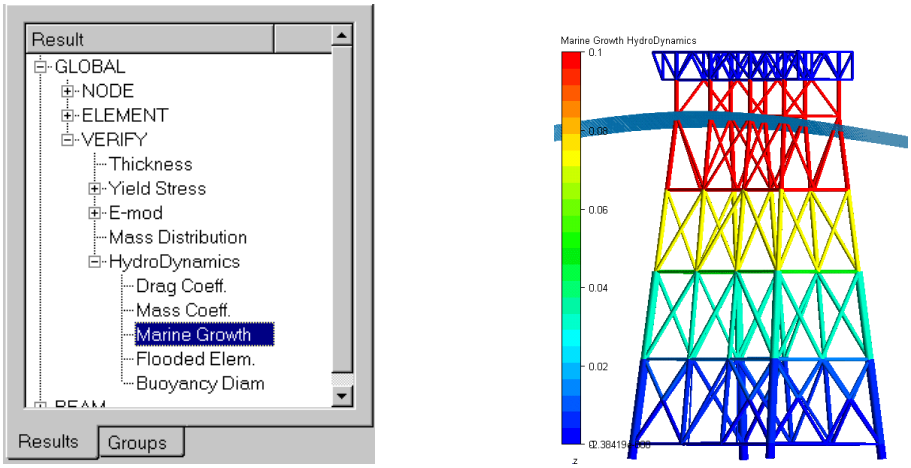


Figure 3.4-1 Visualizing the Marine Growth on the individual members.

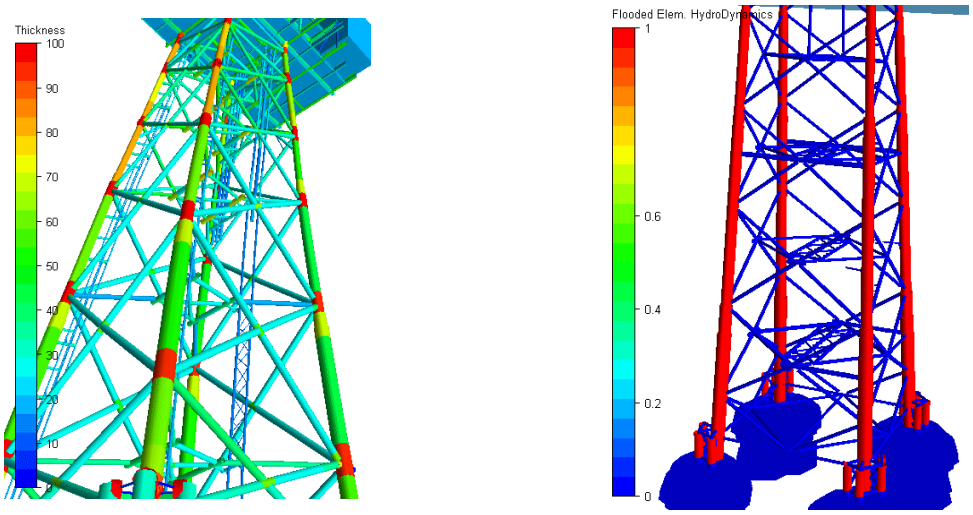


Figure 3.4-2 Visualizing the Wall thickness (left) and Flooded Members (right)

Detailed information of the model is given using the "Verify/Verification" option, and in Figure 3.4-3, node and element information is presented. If the model contains a foundation model, key information about the soil properties are presented by pointing on actual soil disc elements. (If the element(s) in question are hidden, use the clip command, f ex "pick and remove element").

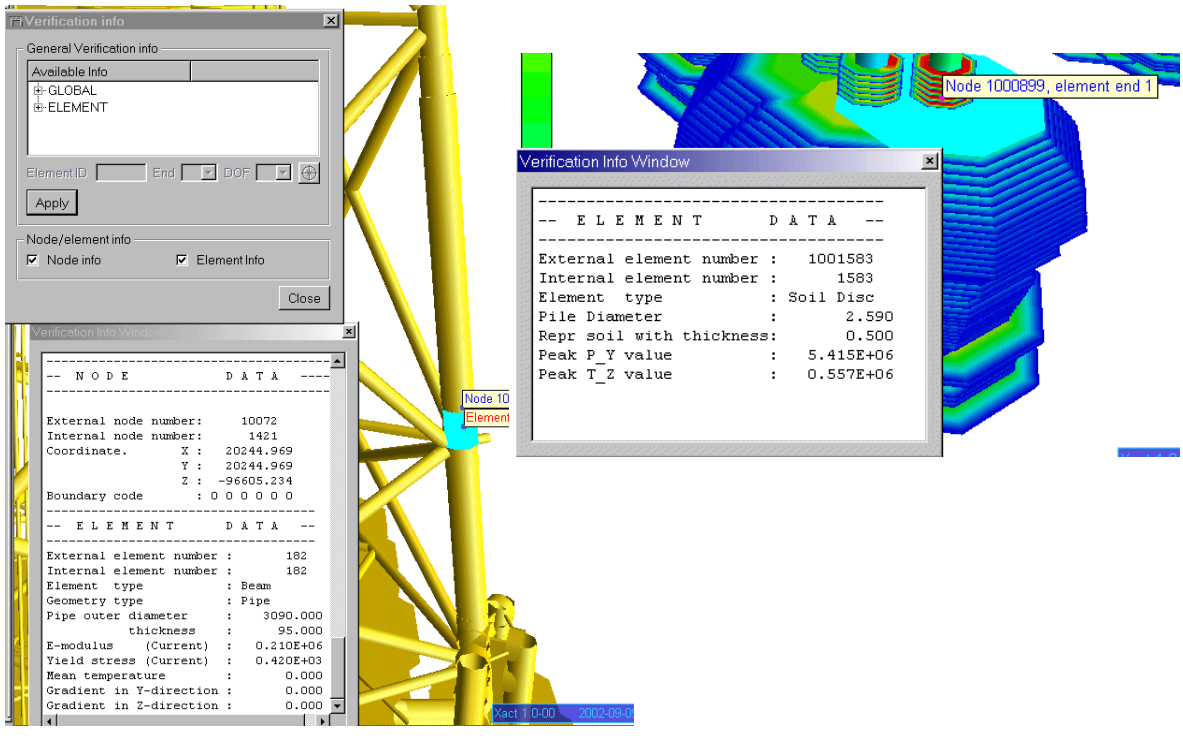


Figure 3.4-3 Print of Node- and Element Information.

3.4.2 Inspecting USFOS results

In the following, results from a typical jacket "push over" analysis are presented. In Figure 3.4-4, the USFOS response curve (Global displacement vs Global Load) is by default presented in the plot. By default, the deformed shape is visualized with displacement scaling factor=1.0, (could be modified by the user).

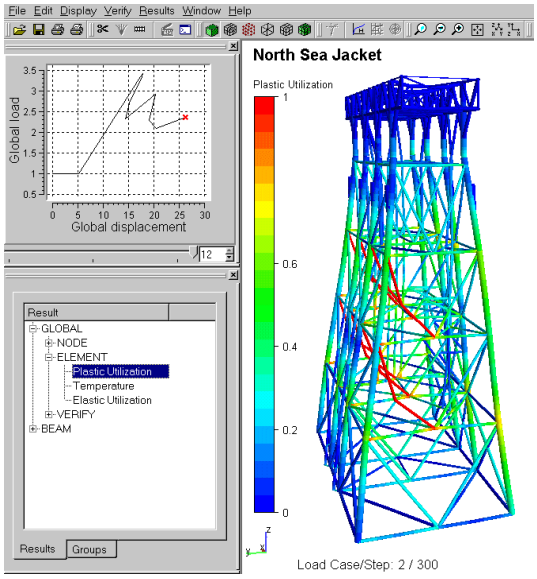


Figure 3.4-4 Jacket "Push Over" Analysis Results

A "Snapshot" of the plastic utilization of the structural members for the *final step* (note the red cross in the curve, which indicates the actual load step) is visualized in the image window. (Actual Load Case/Step is printed in the image window). To select an analysis step to inspect, either point on the curve at actual state, or step forward/backwards using the slide or the up/down arrows.

If a foundation model (piles or spud can) is included in the simulation, the utilization of the soil is presented together with the structural model, see Figure 3.4-5.

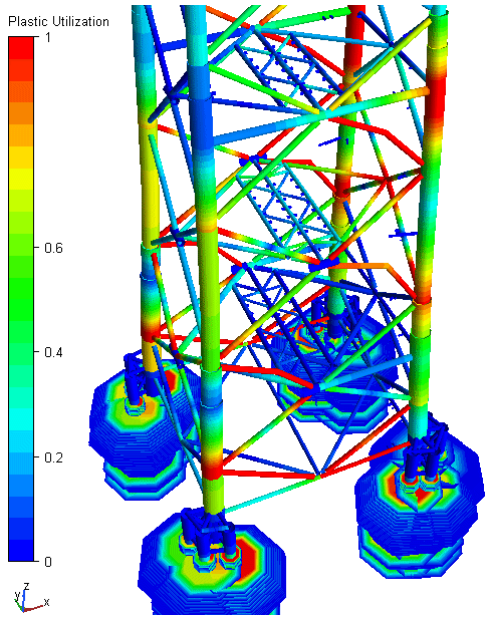


Figure 3.4-5 Plastic Utilization of structure and pile/soil foundation

In order to get a better overview of the individual components of a complex jacket structure, the *clipping function* (see Clip…), will together with the Element Numbers ease the documentation of the analysis, (see Figure 3.4-6).

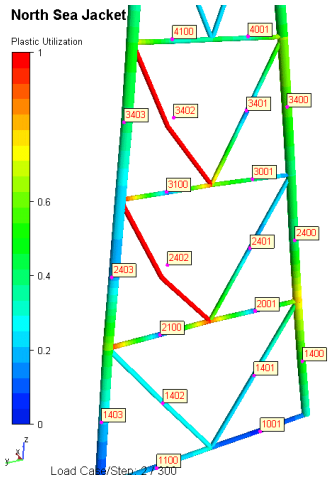


Figure 3.4-6 Visualization of the utilization of one Row with element numbers applied.

In Figure 3.4-7, plot of the axial force of one member, (element 2402, which has buckled) is included, and it should be noted that the red cross in the main plot window (upper left corner of the interface) and all history plots are synchronized helping the user to interpret the analysis results. Any history plot could be used for selecting of actual load step to inspect, and an "unlimited" number of history plots could be defined, ("New Plot" gives a new plot, while "Update Plot" updates the current active plot).

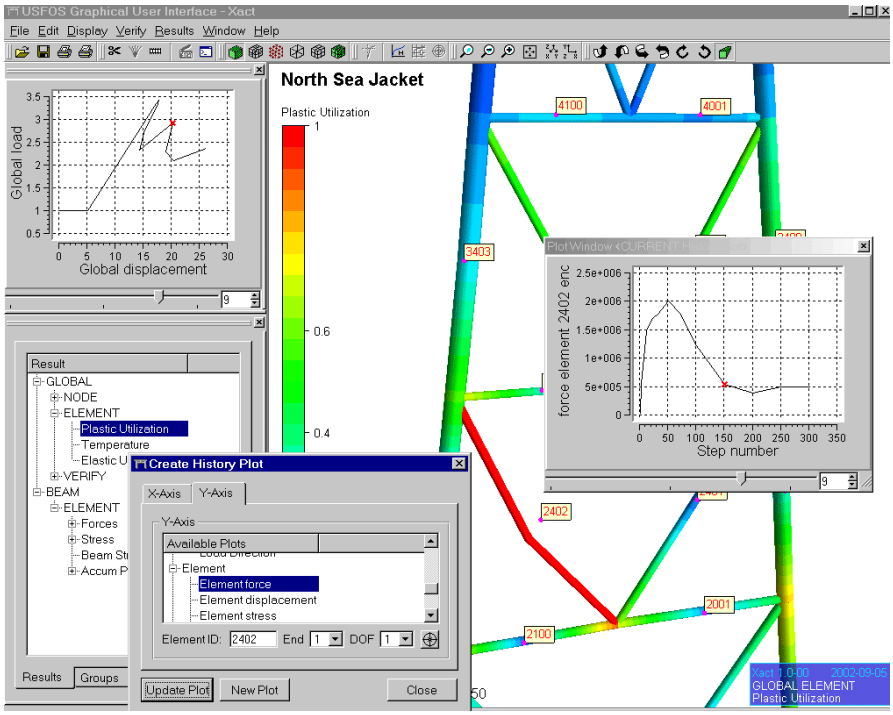


Figure 3.4-7 Visualization of the utilization of one Row with element numbers applied.

3.4.3 Extract Results for documentation. Images

Images contain valuable information. As structural analyses typically are documented in written reports, the transfer of data from the User Interface to the Word Processor is important for the user's efficiency. If a general image tool is available, it is always possible to "grab" any image from the screen (as used for this document), but its also possible to save the images directly from the interface. Several file formats are available (tif, gif, bmp …), and the screen resolution (default is 480 x 640) could be modified by the user.

In addition to use the menu, it is possible to access the "Save scene to image" using short key.

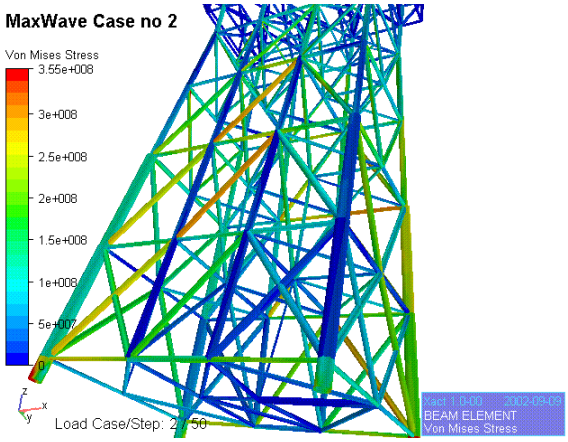


Figure 3.4-8 "Save Scene to Image" with default resolution

Some time saving hints:

- Use Ctrl M to get the dialogue box
- The interface "remembers" last used image format
- Default file names are suggested, and the name is automatic incremented, (first image gets "image0.gif", next get "image1,gif", etc)
- Hit return to save.

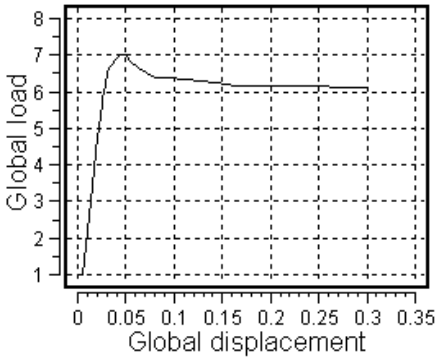
This means that by typing: ***Ctrl M*** followed by ***C/R C/R*** saves the image on the ***default directory*** (NOTE that default directory is dependent on start method, see page 7).

3.4.4 Extract Results for documentation. Curves

In addition to images, 2D curves contain valuable information, and the user may need the plot information for different purposes. The available options are:

- Print (default printer)
- Copy (send to clipboard for import in f ex Word or Powerpoint)
- Save plot as a file (***bmp*** format)
- Copy the plot data into f ex Word, Excel etc
- Save the plot data on a file.

Figure 3.4-9 shows the history plot together with the plot data. Axis labels are written on the top.




```
#
#Global displacement
#Global load
0.00438014  1
0.0131412   3
0.0210948   4.8153
0.021916    5
0.0226429   5.15826
0.0229894   5.23253
```

Figure 3.4-9 History plot and the plot data

Similar to the saving of images, several short keys exist for fast import to f ex Word:

- Define plot
- Press ***ctrl c***
- Activate Word and "paste" (***ctrl v***)

Unfortunately, some word processors (f ex Word) will auto scale the plot, (small plots become bigger), and it is therefore necessary to adjust the plot size manually after import to obtain same size on paper as in the graphical user interface.

3.4.5 Extract Results for documentation. Facts (text)

In addition to images and plots, facts are required in the documentation of an analysis. Some facts are available from the User Interface, and this type of information is organized under "Verify/Verification Info".

Most of the items are dependent on the load step, and results for the current step are printed, (The Print ***is not*** automatically updated if another step is selected).

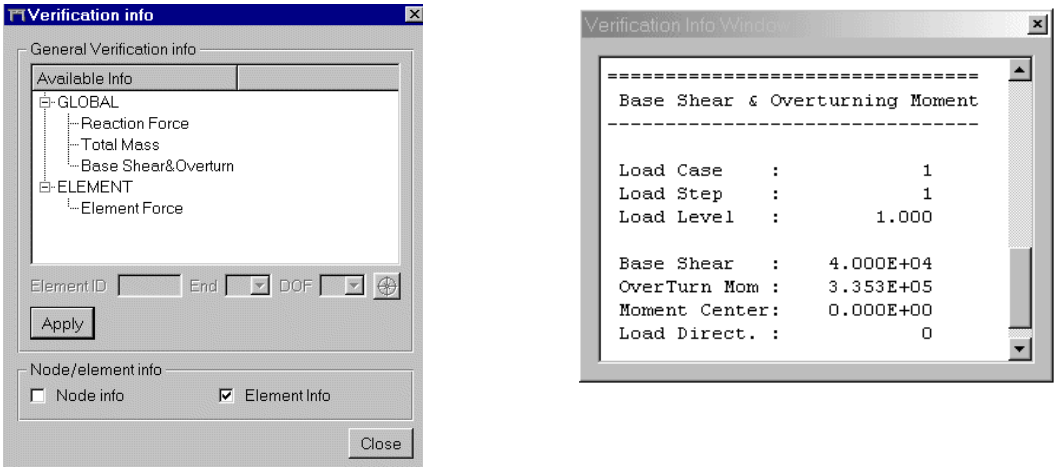


Figure 3.4-10 Verification Info and print of Base Shear etc for a given load step.

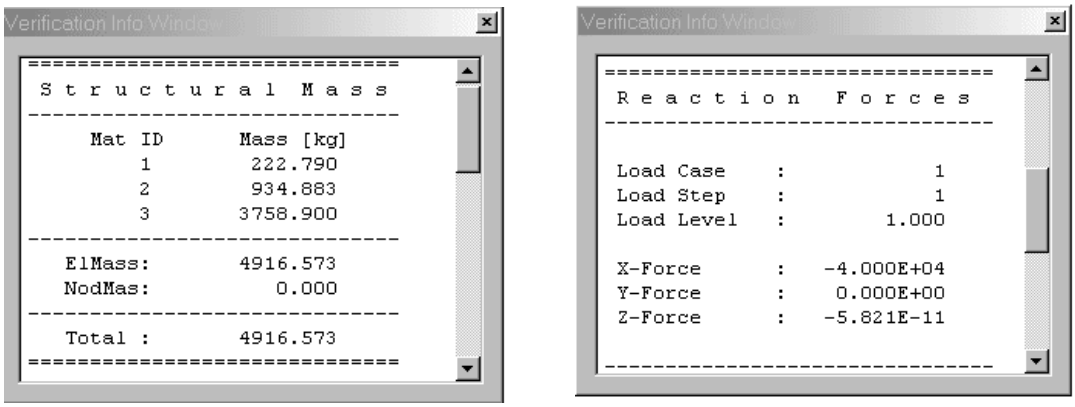


Figure 3.4-11 Print of Structural Mass and Reaction Forces .

The information in the "Verification Info Window" could be selected and imported to f ex Word (mark the actual information using left mouse button, and use the right to copy).

3.4.6 Extract Results for documentation. Hydrodynamics.

If a dynamic wave analysis has been performed in USFOS, plots of wave kinematics could be generated. The user selects where (in the X-Y space) kinematics should be calculated, and a depth profile of actual wave kinematics component (either Acceleration or Velocity, X- Y- or Z- component) is generated. The user defines the X-Y position by pointing at a node on the structure.

In Figure 3.4-12, two depth profiles are selected, one at the left hand side of the jacket, and one at the right. As seen from the wave surface position, the right hand side of the platform has approx. maximum velocity while the left hand side is between max and min. Eventually current profile is included in the velocity profile. Zero on the vertical axis is the MWL, and minimum Z-value is the depth specified in the USFOS Wave Data input.

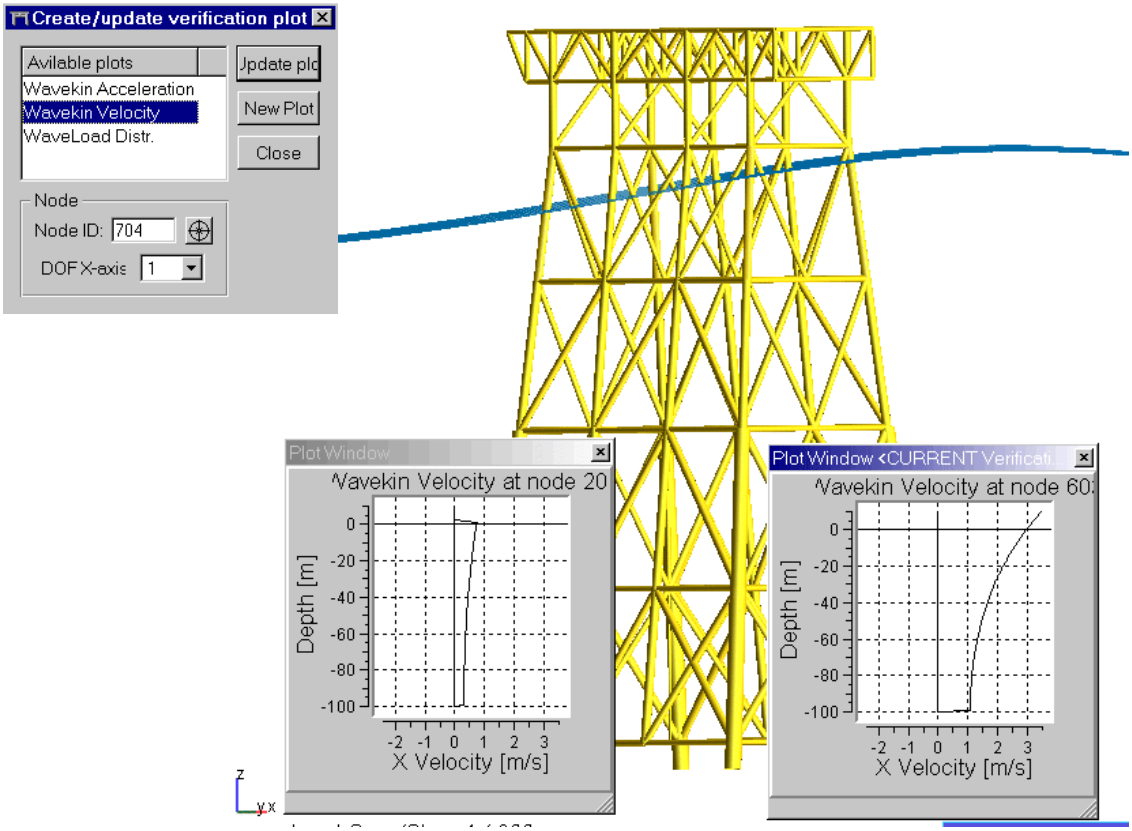


Figure 3.4-12 Plot of Wave Kinematics at different positions on the Jacket. .

4 Graphical User Interface

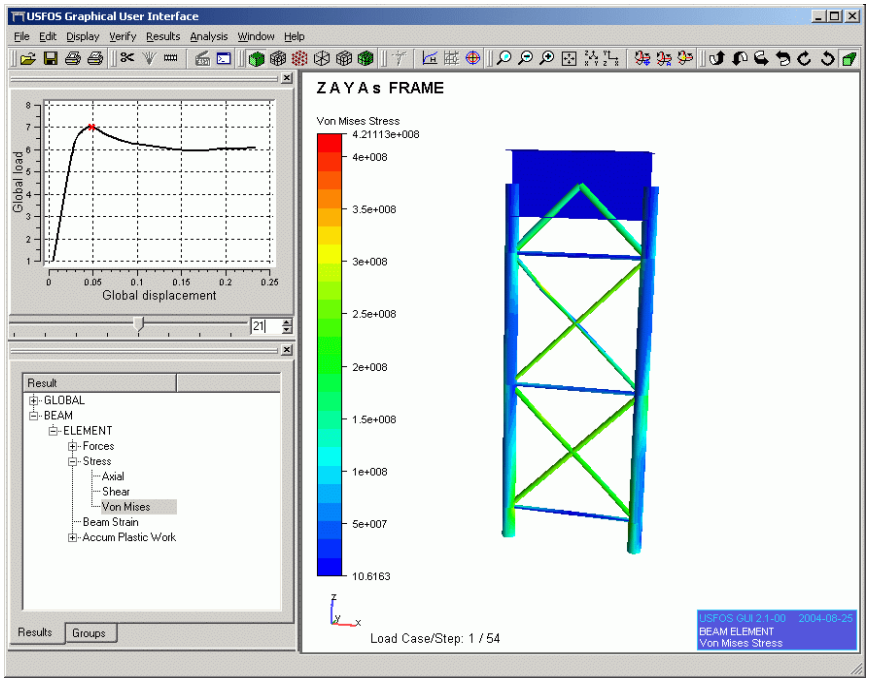
4.1 Introduction

This chapter describes the graphical user interface and the functionality of USFOS/Xact. It is a walkthrough of the contents in the user interface, which includes the main window with the docking windows it may contain, menus in the menu bar, the toolbar, and a number of dialogs that are available through the menus and toolbars.

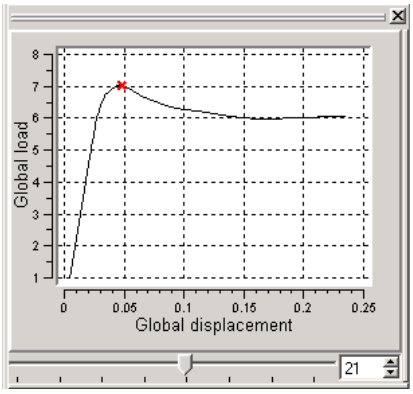
4.2 Main Window

When Xact is started, the main window will be empty, and only a few actions are available for the user. The rest of the actions will become available when an analysis is opened. Some actions may still be unavailable after you have opened an analysis file if the file does not contain data to complete the action. I.e. if you have an analysis with no eigenmodes, the eigenmodes button will not be available.

Once a raf-file is loaded, the graphics window will show the model. On the left side are two dockable windows. The top window is the main control plot. The lower window is the result and groups window.



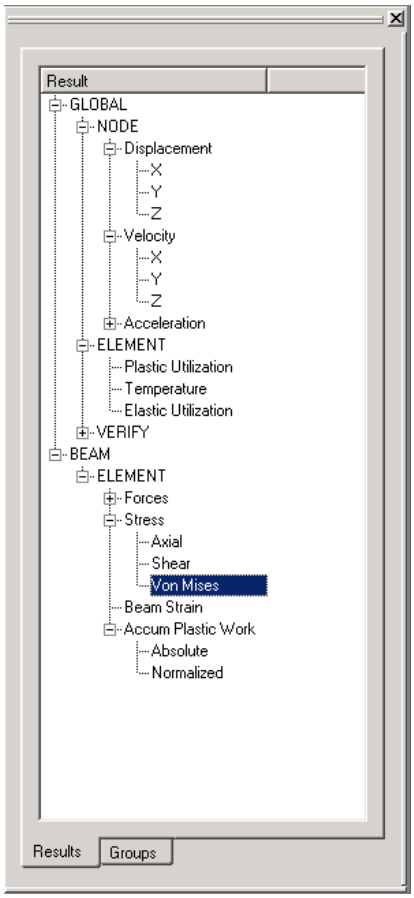
4.2.1 Control Plot



The main plot window will show the default plot for the analysis. Once a model is opened, you can set the load step from this plot window. There are several ways to set the load step. Either by clicking on the point in the plot you want to go to, by dragging the bottom slider, or by using the spin box in the lower right corner. Read more about plots in the "Plots" chapter.

4.2.2 Results/Groups/Parts window

The results/groups/parts window will show what results are on the current analysis and able you to choose what result to show.



4.2.2.1 Results Tab

All results are organized in a tree view. If you double click a result, the graphical window will immediately be updated to show the result on the model. A fringe range will automatically be set to fit the result. Results can also be applied by moving the blue selection up and down in the list and pressing enter.

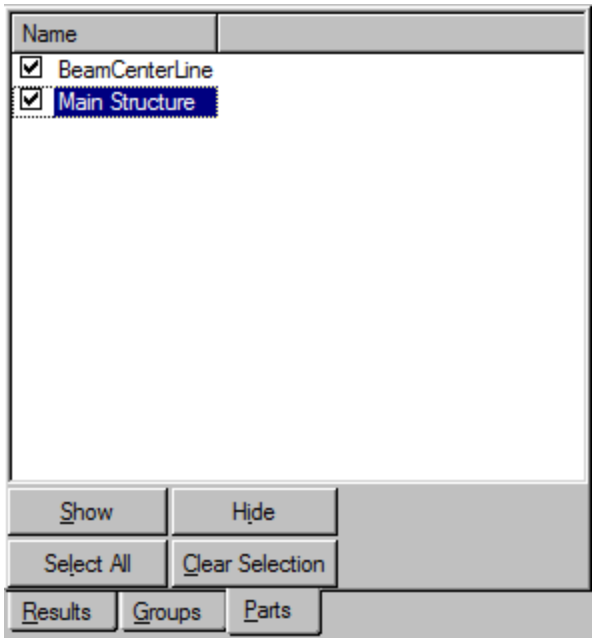
4.2.2.2 Groups Tab

If any predefined groups are available for the analysis, they will turn up in this list. To show a group, just right click on it and pick "Show selected groups", from the menu that pops up. To display several groups, select what groups you want to show, right click them, and select "Show selected groups". All selected groups will be displayed in addition to groups already displayed. To hide groups, just right click in the group tab, and choose "Hide selected groups". This hides all groups you had selected.

With no selection you can also right click and choose “Show all groups”, “Hide all groups” and “Show complete model”.
The “Highlight selected groups” checkbox at the bottom is checked by default, which means that the currently selected groups are highlighted in the graphics window. The elements of the highlighted groups are drawn in magenta color.

4.2.2.3 Parts Tab

The parts tab contains a list of the parts in the model. Each part name has a check box on the left side. Checked part names indicate that the corresponding part is visible, and an unchecked check box means that the part is hidden. The following picture shows and example with two parts:



One or more parts can be selected at once (use the Ctrl or Shift button while selecting). The “Show” and “Hide” buttons show and hide all currently selected parts, respectively. “Select All” selects all parts, while “Clear Selection” deselects all parts.

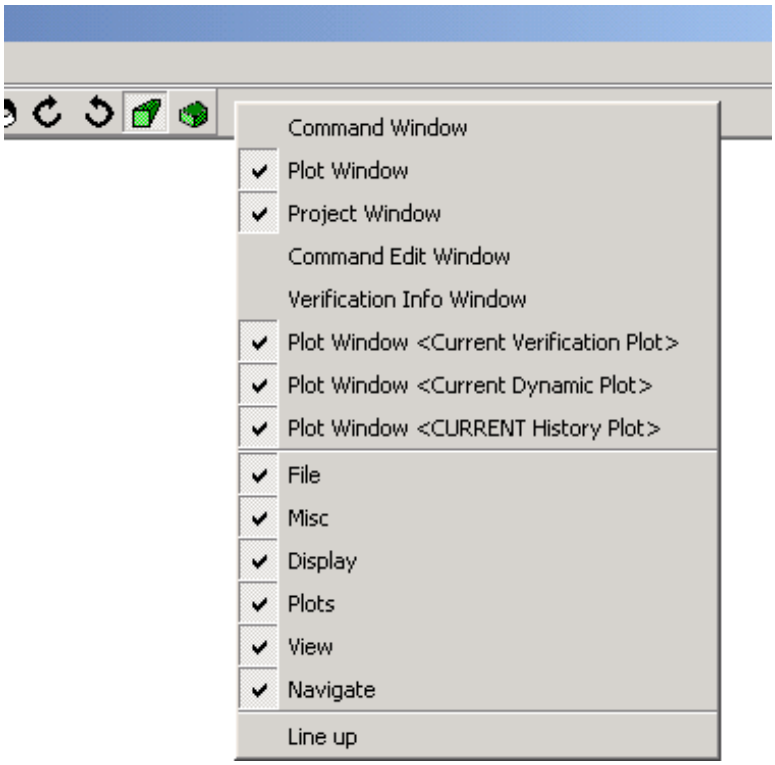
4.2.3 Command Window

All menu commands can be accessed directly by using the command window. For a complete command index, see Appendix 1. When you start up Xact, the command window will be hidden. To display it go to the Window menu and choose “Command Window”

4.2.4 The Toolbar

If you right click anywhere on the toolbar, a menu will pop up,. The menu shows all the windows in the application. Windows that are currently displayed in the application is shown with a small check sign on the left. All toolbars in the application is also listed, and if they are displayed, they too will have a small

check sign to the left. If you want to hide a plot, window or toolbar just click on it in this menu, and it will be hidden from the application. To bring it back again, right click the toolbar again to bring up the menu, and check the windows you want to display.
In the menu below we see that a number of plots has been created and that they all are displayed. We also see that the command window is not shown in the application, and that all toolbars are displayed.



4.2.5 Navigating in the Graphics Window

You can rotate, zoom and pan the model in the graphics window.
Pan – To pan the model, press the left mouse button and drag the mouse in the direction you want the model moved. Your cursor will change to a hand when you left-click in the graphics window.
Zoom - To zoom the model, press both left and right button and drag the mouse towards you to zoom in, and push it away to zoom out. The cursor will change to an arrow pointing inwards the screen when you zoom.
Rotate – Press and hold the right mouse button to rotate the model. Your cursor will change, and when you move the mouse the model will rotate accordingly. It may take a bit of practice to get used to how the rotate function behaves.

In addition to these navigational functions there is a toolbar with functions for stepwise rotation, and a "navigation cube" for navigation to specific views. This will be explained later.

4.2.6 Picking

You can pick one element by holding down CTRL and left clicking an element with your mouse. Element number, closest Node, and End will be displayed for the picked element. The picked element will also be colored light blue. To clear any picking, hold CTRL key down and left click on an empty space on the graphics window.

4.2.7 Plots

There are four different plot-types in Xact: History Plots, Dynamic Plots, Material Model Plots and Verification Plots.
All plots have some common features. By right clicking in the plot, you will bring up a menu. From this menu, you can print the plot, copy it to the clipboard, export it to a image file, copy it as text to the clipboard (pairs of x and y-values), save the x and y values to a text file and you can destroy the plot. When you destroy a plot it will be removed from the application. It cannot be retrieved from the menu-bar.

4.2.7.1 History Plots

All history plots can control what load step you want to display. Load step can be selected by clicking in the plot on the point you want to display, by dragging the slider at the bottom, by increasing or decreasing the value in the spin box, or by typing a value in the box.

4.2.7.2 Dynamic Plots

Dynamic plots show data that are the same for all steps. No interaction is possible with dynamic plots.

4.2.7.3 Material Model Plots

Material Model Plots show a different plot for each step. When you change load step via a history plot, the data in the material plot will be updated accordingly.

4.2.7.4 Verification Plots

Verification Plots act in the same way as Material Model Plots. The plots are updated when the load step is changed.

4.2.7.5 Current Plot

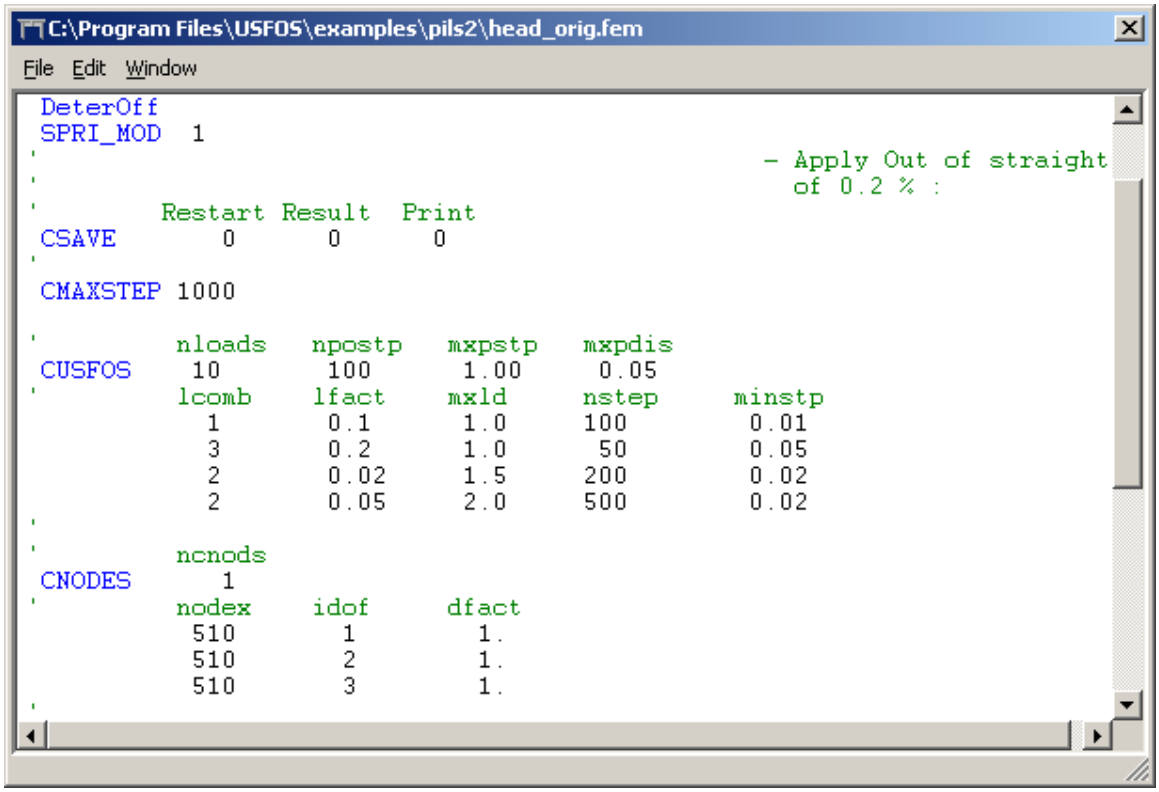
There are two types of current plots. For all four plot-types there exists a current plot. A current History Plot, A current, Material Plot and so forth. In addition there is a current plot for all types. That plot is always the last one selected. You can se what plot is the current plot for all types by looking at the plot title. The plot title of such a plot, will have CURRENT written in capital letters.

4.2.8 The USFOS Editor

The application has an internal text editor. You may open any ASCII text file in this editor and edit it (see sections 4.3.3 and 4.3.4), even though it is originally meant for USFOS input files (see section 4.8.2).

The editor does syntax highlighting of USFOS commands, comments and command parameters, has a command browser, and performs automatic insertion of element and node IDs as command parameters by CTRL + left mouse button clicks in the 3D model. See section 3.3.4 for a brief description of how to use the editor, illustrated with an example. A more detailed description is given here.

The USFOS editor looks like illustrated in the following figure:



The USFOS editor uses syntax highlighting. If a line starts with a word that is recognized as an USFOS command, the word is written in blue color. Comments are written in green color and all other text, which should be command parameters, is written in black color.

The editor has a “File” menu, an “Edit” menu, and a “Window” menu. The items in the “File” menu are:

New:
Create a new file.

Open:
Open an existing file.

Save:
Save the current file.

Save As…:
Save the current file to a given filename.

Close:
Closes the current file.

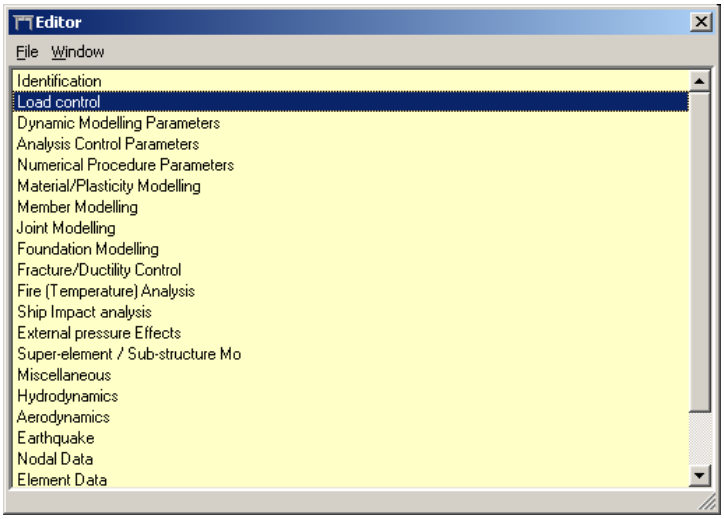
Close Editor:
Closes the editor.

The "Edit" menu has an item called "Find…" which lets you search for a text string and highlight it if found. "Find Again" repeats the last search.

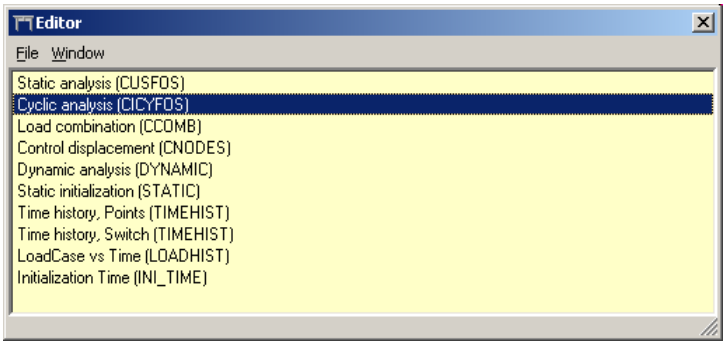
The "Window" menu contains an item for each file being open in the editor and lets you switch between the different files.

The editor gives you help in writing commands. By pressing the ***F1*** key when the text cursor is in a line starting with an USFOS command, the parameter description is inserted as a comment line above the line containing the command.

If you press the ***F2*** key, the "command browser" appears. The command browser contains a list of the command categories. You may select a category by using the up/down arrow keys and pressing the RETURN key, or by double-clicking a category in the list with the left mouse button key.

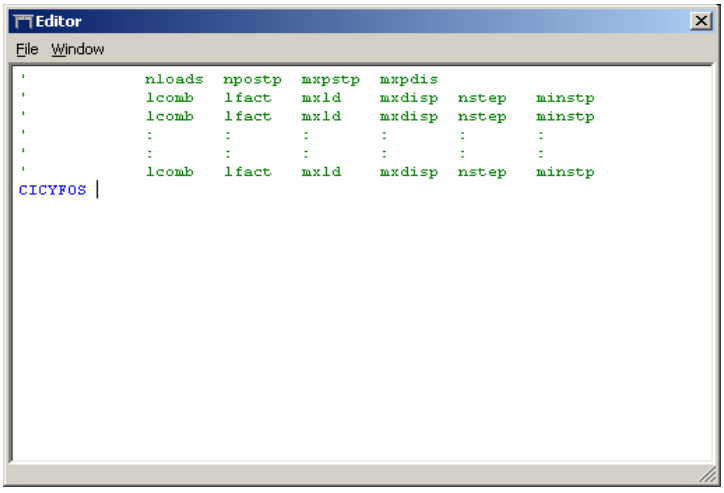


When a category is selected, the command browser shows a list of the commands in the category. To get back to the category list, just press the BACKSPACE



key. A command is selected the same way as you select a category.

When a command is selected, the command browser disappears and the command is inserted in the file you are editing. One or more comment lines with parameter descriptions will be inserted automatically above the line containing the command. Note that the command browser may be closed at any time by pressing the ESC key.



A lot of the USFOS commands have node IDs or element IDs as parameters. Those may be written manually of course, but you may also insert them by pressing the CTRL button and clicking the left mouse button while pointing in the 3D model. This will only work if the text cursor is located in a line containing a command (or a following line with parameters) and the next expected parameter is a node ID or an element ID. Also note that the USFOS editors in the Analysis Control Dialog (see section 4.8.2) have priority over this editor. This means that the node ID or element ID will be inserted in the currently active editor in the Analysis Control Dialog if the dialog is visible and the next parameter to be written is a node ID or an element ID.

4.3 The File Menu

4.3.1 Open USFOS Result File…

Shows a file dialog where you can browse to .raf-files that can be opened in Xact. Once a raf-file is opened, the graphics window will be updated to show the model, and the control plot and result/group window will contain relevant data from the analysis.

4.3.2 Open USFOS Model File…

Opens a file dialog from which model files may be opened. Once a model file has been opened successfully, the graphics window will be updated to show the model. The control plot and the result/group window will be empty since a model file contains only the model, not any results. Xact is able to read structural files supported by USFOS (SESAM and UFO).

4.3.3 Open Text File…

Opens a file dialog for selection of files to be opened as text files. Any file can be opened here, and is displayed in the internal text editor when it has been opened. Multiple files can be open simultaneously, and the Window menu of the editor lets you select the file you want to edit.

4.3.4 New Text File…

Displays the internal editor and creates a new file ready for editing.

4.3.5 Load View Attributes…

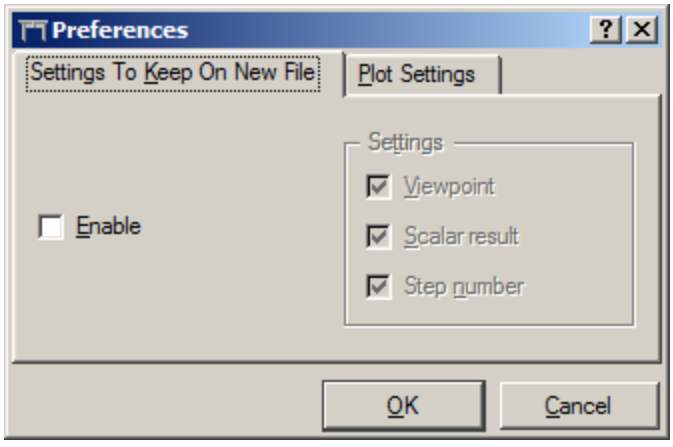
Enables users to load view attributes from an USFOS Attribute File (.usa). For convenience the default file that pops up in the file dialog is usfos.usa. View attributes can be saved with the "Save view attributes" action.

4.3.6 Save View Attributes…

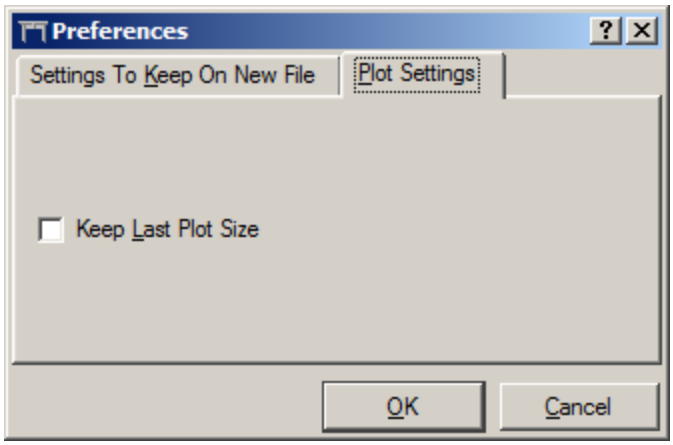
Saves a view attribute file to disk. This file contains the rotation and viewpoint you have in your graphics view at the moment. You can save your current view, and load it later. The view attribute file is global, so it can be used for other models as well as the one you saved it with. By default the name for the saved view attribute file is usfos.usa. This name can be changed at the users convenience.

4.3.7 Preferences…

Opens the preferences dialog, which contains two tab windows: "Settings To Keep On New File", and "Plot Settings". The former is used to select settings to keep when a new file is opened, which may be useful for comparison of the outputs of different analysis runs. There's also a check box for toggling of this functionality:

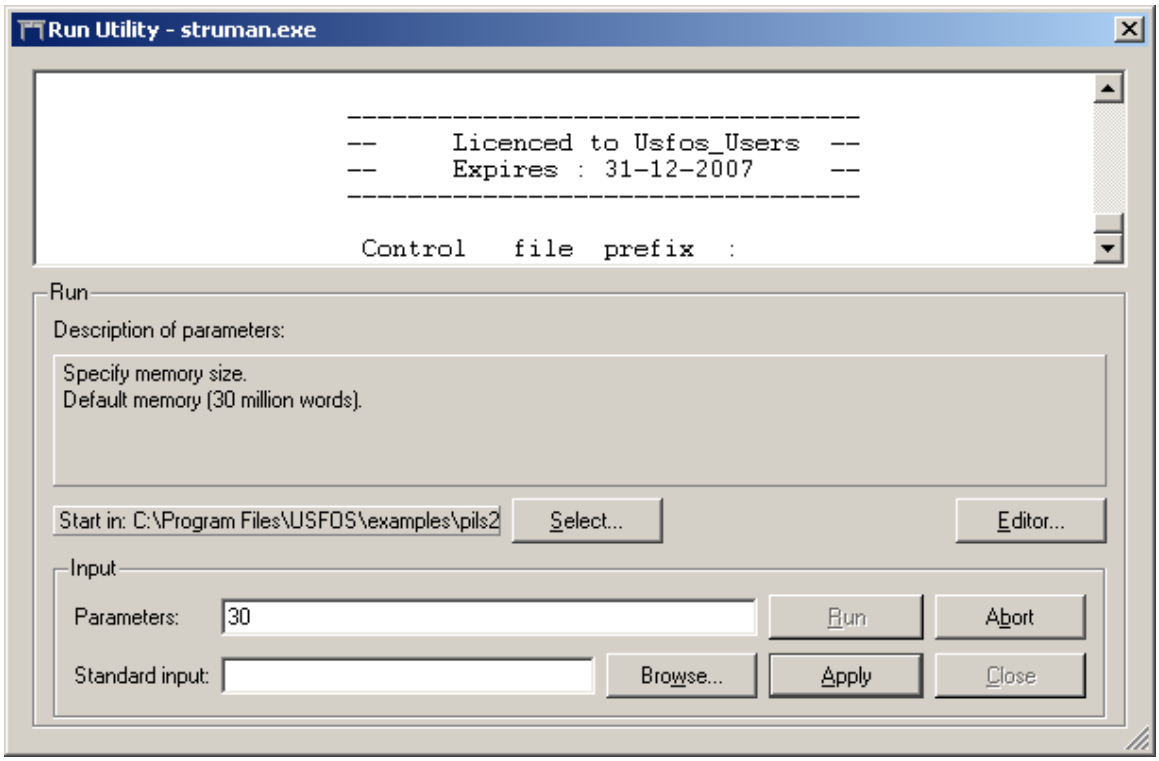


The “Plot Settings” tab has a check box that can enable usage of the size of the last plot that was created when a new plot is created:



4.3.8 Run Utility

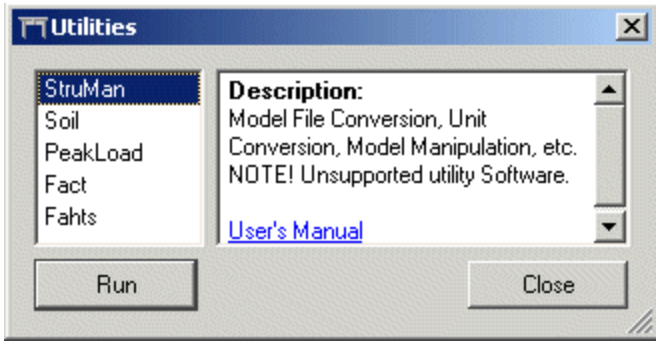
This sub menu is available only if a configuration file named “Xact_Utils.usc” was present in the same directory as the USFOS GUI executable when it was launched, and if the file actually contained valid information about available utilities. The sub menu contains the names of the utilities according to the specification found in the configuration file. A “run utility dialog” is opened if one of the utilities is selected. The dialog presents the standard output from the utility and lets you give standard input to it. The utility is launched when the “Run” buttons is pressed, starting with the given parameters. The following figure shows what it looks like when StruMan is launched from this menu.



The "Output" window on top shows the standard output and the "Standard input" control lets you type standard input. Standard input is given to the utility when RETURN is pressed or when the "Apply" button is clicked. Note that the "Browse…" button lets you select a filename from a file dialog. The filename is appended to the current text in the "Standard input" control. The "Abort" button terminates the current run and the utility is launched again if you click the "Run" button. The "Close" button closes the dialog. See Appendix B for a description of the format of the configuration file "Xact_Utils.usc".

4.3.9 Utilities…

This menu item is available only if a configuration file named "Xact_Utils.usc" was present in the same directory as the USFOS GUI executable when it was launched, and if the file actually contained valid information about available utilities. The menu item opens a dialog that may look like the following figure.



The list on the left side shows the names of the available utilities as defined in the configuration file. When a utility is selected, a description of the utility is shown on the right side of the dialog if a description is given in the configuration file. A hyperlink to a manual for the utility is given after the description, if present. The manual file will be launched if the hyperlink is clicked. This will work only if the manual's file type (typically .pdf, .html, or .txt) is associated with an application that is able to read it.

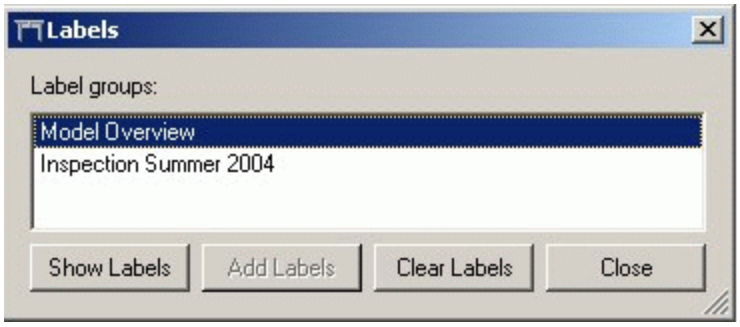
The "Run" button launches the selected utility, which will be run through a dialog. This is analog to launching a utility through the"Run Utility" sub menu. See previous section. See Appendix B for a description of the format of the configuration file "Xact_Utils.usc".

4.3.10 Read Labels From File…

Opens a file dialog where you may select and open an USFOS Label File (.usl). This file type is an ASCII text file containing user-defined labels for nodes and/or elements. This is useful if you want to add textual information to specific parts of the model. User-defined labels are attached to nodes or elements in the 3D model. A dialog will appear when a file containing labels has been loaded successfully. The next section describes this dialog. See Appendix B for a description of the USFOS Label File Format.

4.3.11 Labels…

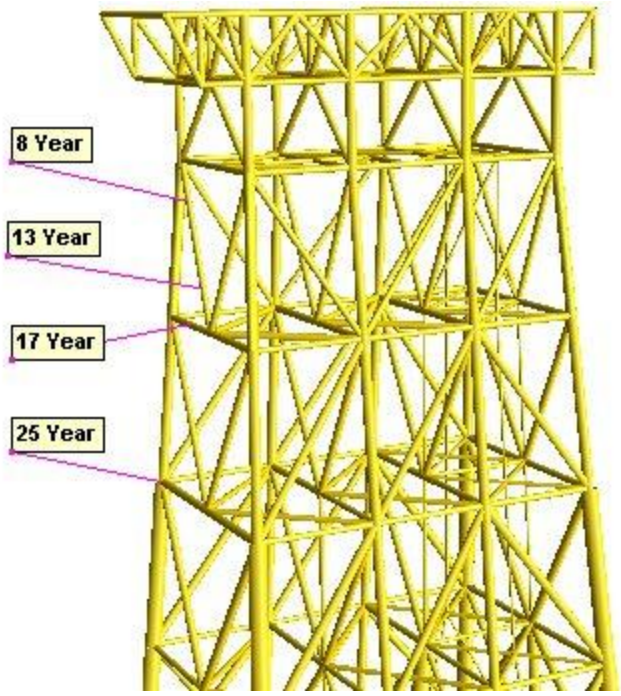
This menu item is disabled unless you have read a valid USFOS Label File (.usl) (see previous section). The menu item opens a dialog as shown in the following figure.



The dialog contains a list of the label group names. A label group is selected by pointing at its name and clicking it, or by using the arrow up/down buttons. "Show Labels" shows the selected group of labels. To show more than one group of labels simultaneously, click the "Add Labels" button. "Clear Labels" will remove all user-defined labels from the 3D view. The "Close" button closes the dialog.

Note that you may manipulate visible user-defined labels in the 3D graphics window. If you click the right mouse button within the window, a popup menu

will appear. Selection of the menu item “Attach all labels to 2D view” will make all visible user-defined labels “float in 2D”. This means that the labels are not attached to nodes or elements any more and will not follow the model when you navigate in the 3D graphics window. Lines are drawn from the labels to the nodes or elements the labels belong to, as shown in the following figure.



Selection of the menu item “Reattach all labels to 3D model” will reattach all visible and floating user-defined labels to the 3D model.

Labels that are floating in the 2D view may be moved by moving the mouse while pressing the Ctrl key and clicking and holding down the left mouse button.

Also note that you may manipulate the user-defined labels individually by pressing the Ctrl key while right-clicking the node or element a label belongs to. This will bring up a context menu saying “Attach label to 2D view” or “Reattach label to 3D model”. Selection of one of those brings up another level in the menu conaining the label text, and selection of this item will perform the action.

4.3.12 Export to GLview Pro…

Exports the current model and its states to a VTF-file. This file can be opened with GLview Pro from Ceetron for further post-processing (see http://www.ceetron.com). You can choose to export the current state/load step, or to export all available states.

4.3.13 Export to Image…

Saves the current view as a picture. The first dialog lets you set the resolution for the picture. Once you have set the right resolution, you will be able to choose a filename and format for the picture. Current supported formats are; .tif, .bmp, .gif, .eps, .ppm, or .rgb. Note that if a region has been selected, the region will specify the area that is exported as an image to file. See also 4.3.15.

4.3.14 Print…

Print Scene – Prints the model as it's seen on the screen. First dialog lets you set the resolution of the print, and the second one lets you choose a printer.
Print Plot – Prints CURRENT plot. See Plots on definition on CURRENT plot.

4.3.15 Select Region

When this menu item is selected, clicking and dragging inside the graphics window lets you select an area (a rectangle). This area specifies the area that is exported when an image is exported (see 4.3.13). No selected region means that the entire view is exported.

4.3.16 Recently Opened Files

After the print item in the file menu a list of the four last opened models will appear. If you are using Xact for the first time, this list will be empty.

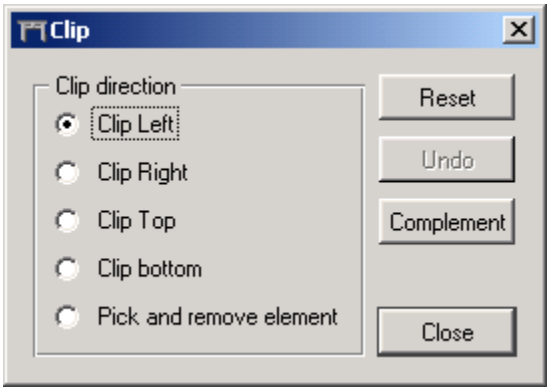
4.3.17 Xit Xact

Closes the current raf-file, and exits Xact.

4.4 The Edit Menu

4.4.1 Clip…

The clip action brings up the clip dialog. From there you can start clipping your model. The cursor takes the form of a pair of scissors to show that you are in clip-mode. To clip the model, simply press and hold the left mouse button. A line is drawn, and depending on what clip mode you are in, the model will be cut on either left, right, top or bottom of the line that you drew. You can change clip mode at any time.



If you choose pick and remove element, you will enter a clip mode where you can clip one and one element of the model. Hold down the Ctrl-key on your keyboard while clicking on the element you want to remove. The element will be removed from the model. If you removed the wrong element, click "Undo" in the clip dialog. You can only undo one step back.

The "Complement" button shows all parts that are clipped. When you press the "Complement" button, the text on it will change to "Clipped". If you press the button now, it will show the original clipped model.

Pressing the "Reset" button resets all clipping. All parts are made visible. The complete model is shown.

You exit the clip mode by closing the clip dialog.

4.4.2 Clip Left

Lets you do a one shot clip. Press and hold the left mouse button and drag the mouse. A line will be drawn up. When you release the mouse button, everything on the left side of the line will be cut.

4.4.3 Clip Right

Same as Clip Left but clips everything on the right side of the line.

4.4.4 Clip Top

Same as Clip Left but clips everything over the line.

4.4.5 Clip Bottom

Same as Clip Left but clips everything under the line.

4.4.6 Show Clipped

Shows the clipped model. If you have no clipped model, this choice will be grayed out.

4.4.7 Show Complement

Shows the complement of the clipped model. If you have no clipped model, this choice will be unavailable

4.4.8 Reset Clipping

Resets the clipping and shows the complete model.

4.5 The Display Menu

4.5.1 Show as Surface

Updates the view to show the model with a solid surface. This is the standard display mode.

4.5.2 Show as Lines

Model in view is show as lines/grid

4.5.3 Show as Points

The model is represented with a number of points at element ends.

4.5.4 Show as Hidden Lines

Same as “show as lines” but lines behind elements are hidden.

4.5.5 Mesh

Displays a mesh over the model, if you are looking at it in “show as surface” mode.

4.5.6 Show Legend

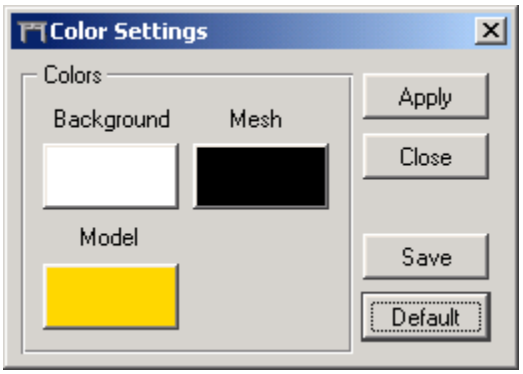
Toggles display of a the legend. The legend shows the colors and names of the parts the model consists of, unless a scalar result has been selected and display of fringes is turned on. In the latter case the legend will show how the model is colored according to the result.

4.5.7 Show Navigation Cube

Toggles display of a “navigation cube” in the lower left corner of the graphics view. Clicking on the faces, edges or corners of the cube or the arrow symbols surrounding the cube navigates the model to special views, typically so that the view direction is parallel with one of the coordinate system axes.

4.5.8 Color Settings…

Brings up a dialog that lets you choose what colors to use for your view. Click on the colored field under Background, Mesh or Model to bring up a color picker one of them. Once the desired color is picked and you exit the color picker, you can press "Apply" to apply the new colors.

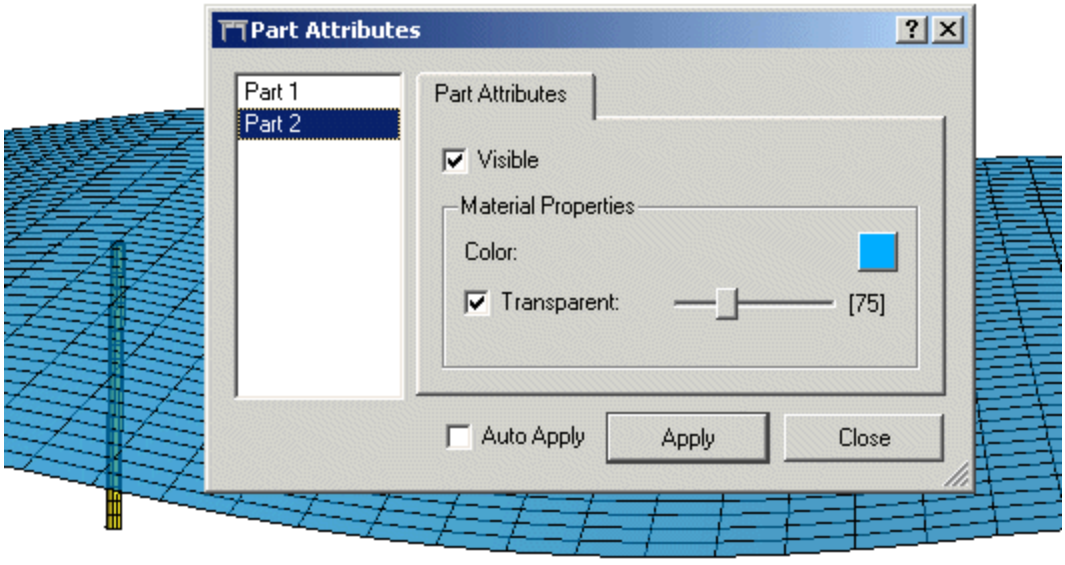


The "Save" button stores away the currently applied colors to the registry, and uses these colors to draw the current and future models.

If you want to restore the colors to the default USFOS/Xact colors, just press the "Default" button, and "Apply" thescolors to the model.

4.5.9 Part Attributes…

Brings up a dialog that lets you change the visibility, color, and transparency of parts. It looks as shown in the following figure.



The parts are listed in the list to the left. The controls will manipulate the currently selected part. Clicking the "Visible" check box will show or hide the part, the "Color:" button brings up a dialog for selection of part color, the "Transparent:" check box turns on or off the transparency of the part, and the transparency slider control sets the transparency of the part. This is a value from 0 to 255, where 255 means total transparency. Clicking the "Apply" button will apply the current settings. You don't have to click this button if "Auto Apply" is checked. "Close" will close the dialog.

4.6 The Verify Menu

4.6.1 Node Numbers

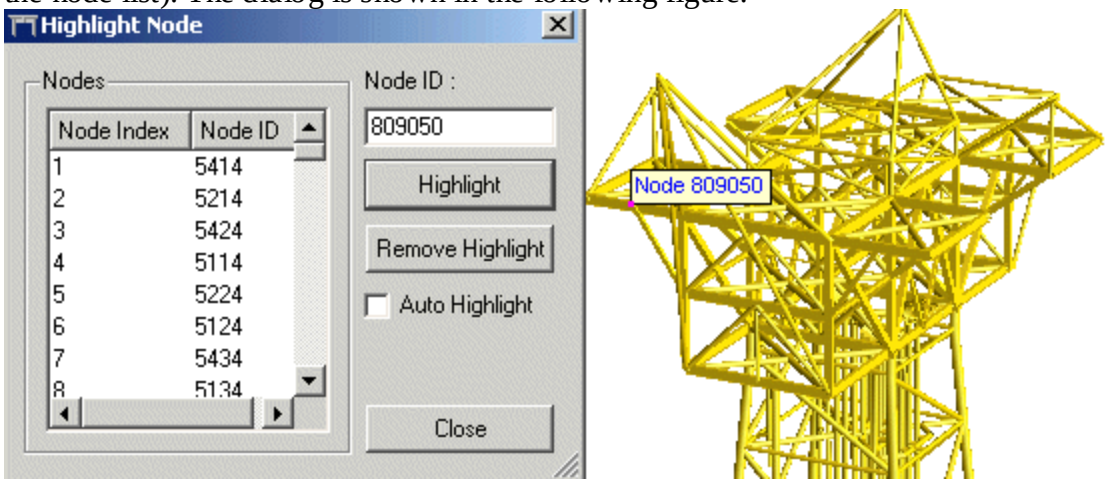
Turns the display of node numbers on/off

4.6.2 Element Numbers

Turns display of element numbers on/off

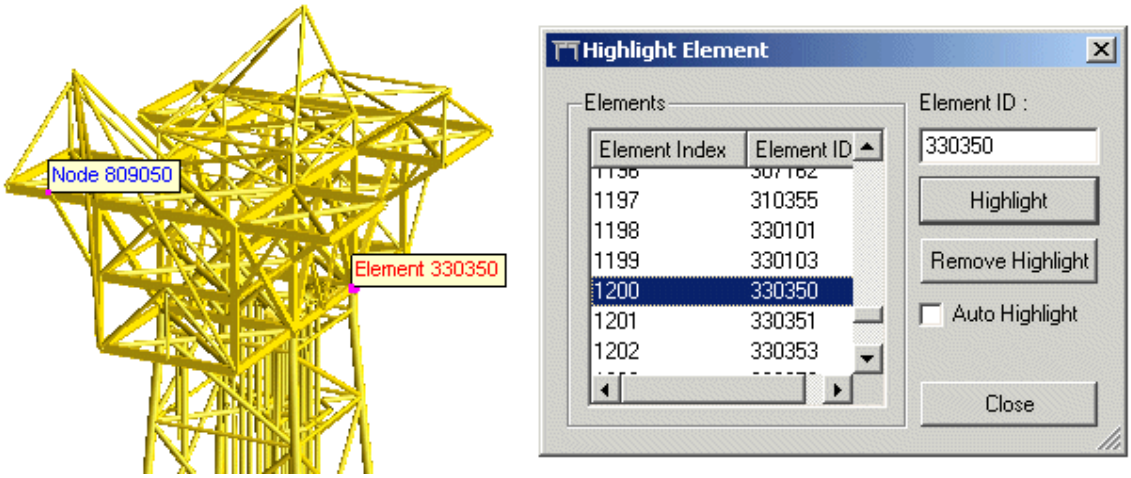
4.6.3 Highlight Node…

Opens a dialog for highlighting of nodes. It contains a list of all nodes and the node selected in the list is highlighted by clicking the "Highlight" button or by pressing the RETURN key. The node ID may be typed in manually as well. If "Auto Highlight" is checked, a node is highlighted as the node selection changes (by pointing and clicking using the mouse or by using the arrow up/down keys in the node list). The dialog is shown in the following figure.



4.6.4 Highlight Element…

Opens a dialog for highlighting of elements. It contains a list of all elements and the element selected in the list is highlighted by clicking the "Highlight" button or by pressing the RETURN key. The element ID may be typed in manually as well. If "Auto Highlight" is checked, an element is highlighted as the element selection changes (by pointing and clicking using the mouse or by using the arrow up/down keys in the element list). The following figure shows what this dialog looks like.

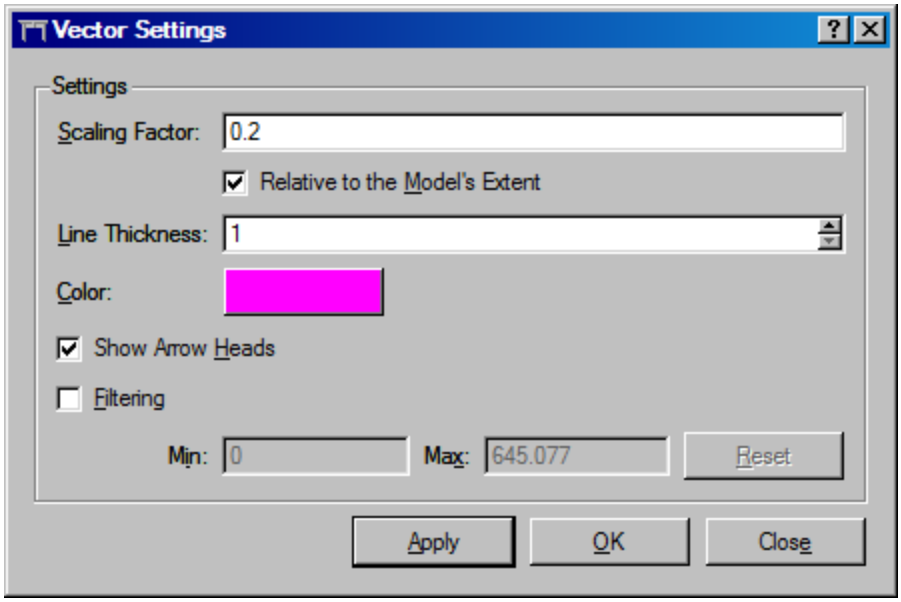


4.6.5 Vectors

Toggles display of vectors. The menu item is disabled unless a vector result or boundary condition like nodal forces has been selected.

4.6.6 Vector Settings…

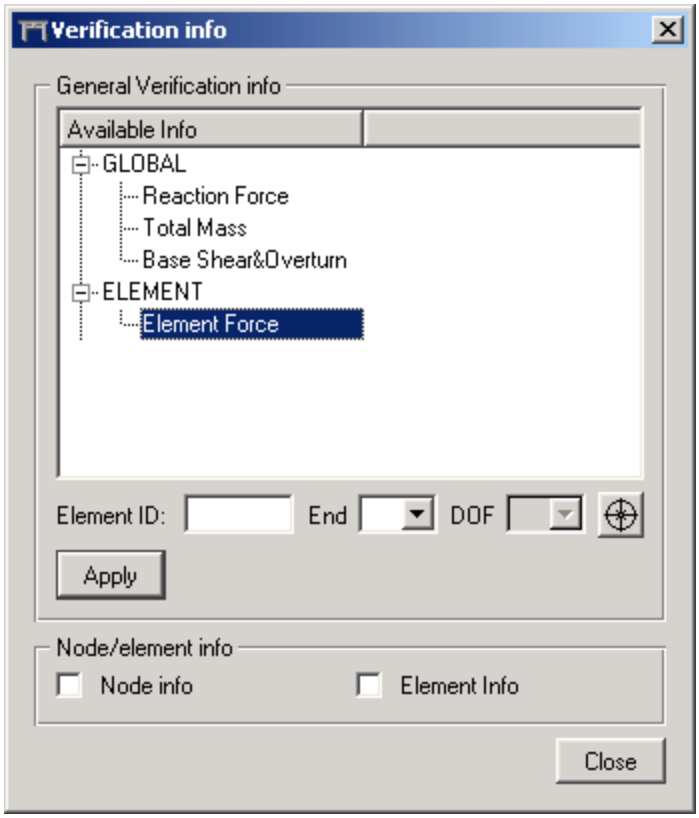
This menu item is enabled when a vector result is selected in the "Results tab window", and selecting it will open a dialog for settings of the selected vector result. Vectors are drawn as arrows with or without arrow heads with a given scaling factor, absolute or relative to the model's extent. This dialog also lets you set the color and line thickness of the vectors. It is also possible to apply a filter so that only vectors with lengths within a given range are visible. The vector settings dialog looks as shown in the following figure.



4.6.7 Info Window…

Displays the Info Dialog box. This box will be filled with information when you use the Verification dialog explained next. The text in the info dialog can be cut out, if you want to use it in other applications or reports.

4.6.8 Verification…



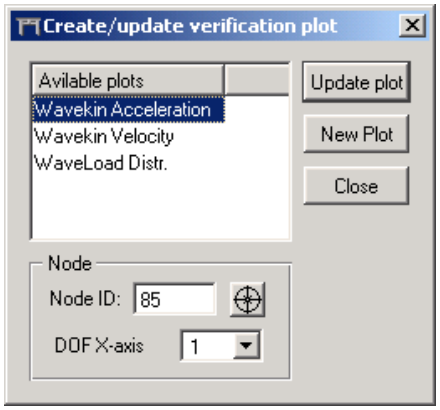
The verification dialog lets you retrieve information about the model. All info is printed in the Info Dialog.
The dialog is split into two parts: one for General Verification info and one for node/element-info.

If any general verification info is available on the opened file, it will show up in the available info-list. Some information might require additional input, like a element ID, End and DOF. This information can be typed in by hand, or picked directly from the model. To pick an element or node, push the "pick" button. It will be toggled and you can do a regular pick on the model (CTRL + Left mouse click). The Element/node ID and End will be filled out automatically from the pick. If you press "Apply", the information for the picked node/element will appear in the Info Dialog. If the info dialog isn't visible on the screen, it will automatically pop up when you do this.

The Node/element info has two checkboxes. If you check the Node Info box, the Info Dialog will display info about nodes when you do picking in your model.

Checking the Element box will do the same for elements. When you close the verification dialog, no info will pop up in the Info Dialog when you do picking.

4.6.9 Verification Plot…



Brings up a dialog that lets you create a verification plot. A list of available plots is shown. In addition to what plot you want to show, you have to choose what node you want to show it for, and the DOF for the x-axis.
Update Plot updates the current verification plot. If you haven't made any verification plots yet, this button will create a new one.
New Plot creates a new verification plot with the data you have selected. All verification plots will be updated when you change the load step.

4.7 The Result Menu

4.7.1 Fringes

Turns the display of fringes on/off. If you haven't chosen any results from the results window, this item will be grayed out.

4.7.2 Fringe range…

Shows the dialog for setting the fringe range. The fringe range is automatically set when you choose a result, but if you want to override those settings, you can do it in this dialog. The fringe range will be reset to an automatic value when you chose a new result.

4.7.3 Displacements

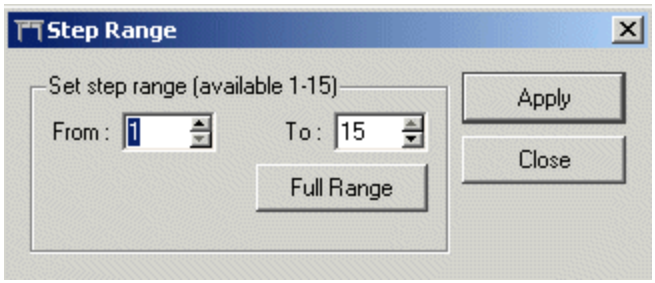
Turns displacements on/off. Displacements are turned on by default.

4.7.4 Displacement Scaling…

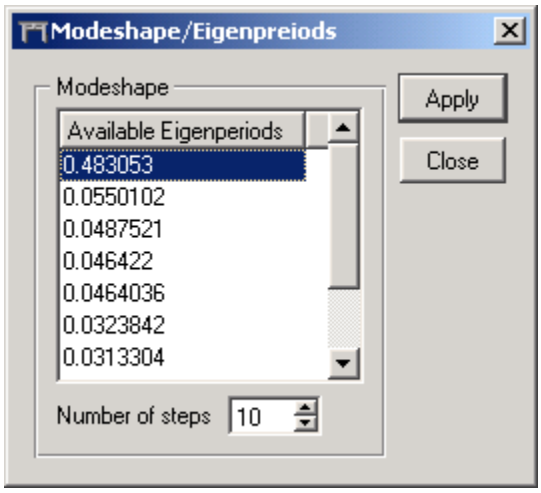
This dialog lets you exaggerate or understate the displacements. A scaling factor of 1 means normal displacements. A value less than 1 will make the displacements smaller, and a value over 1 will make the displacements larger.

4.7.5 Step Range…

Opens a dialog that lets you set the step range. By default, all steps are available, but this dialog gives you the opportunity to focus on a sub range of steps. When a sub range is selected, only steps from the sub range are shown in the plots and included in animations.



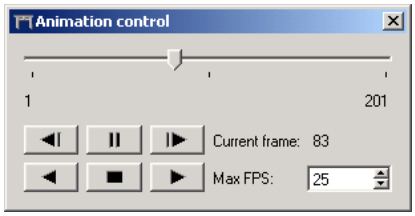
4.7.6 Eigenmodes…



In this dialog you can set up a simple animation of a model's eigenmodes. A list of available eigenmodes will be displayed. Just chose what eigenperiod you want to animate, and the number of steps you want to animate over. The graphics window will show an animation with the input you chose. To end an eigenmode-animation, just chose another load step. If no eigenmodes are available on your analysis file, the menu item will be unavailable.

4.7.7 Animate

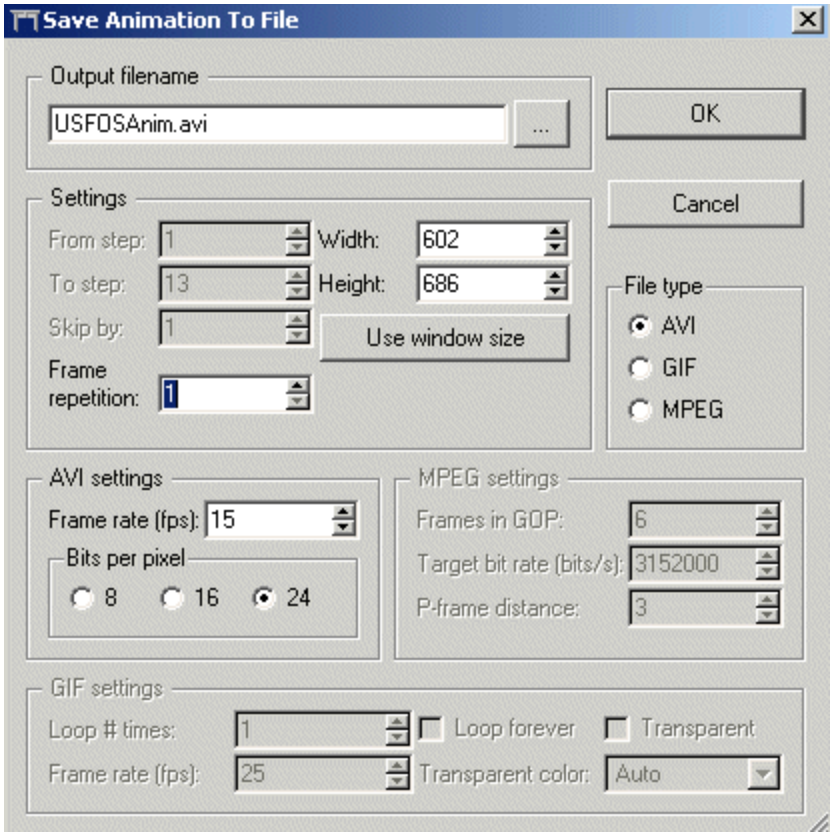
Sets up a simple animation over all steps in the analysis file. The current result, if any, will be shown for all steps. An animation control appears, and from there you can play the animation forwards and backwards. Step forwards and backwards and set the current frame rate. To stop the animation, just choose a



step in the control plot. The animation control will only be available when an animation is running.

4.7.8 Save Animation to File…

If an animation has been set up, this menu item is available and selecting it opens a dialog that lets you save the current animation to an AVI, MPEG, or Animated GIF file. The dialog is shown in the following figure.



The dialog may seem a bit complex at first glance, but there are different settings for the different file formats. Settings that are irrelevant for the selected format are disabled. In the figure you can see that AVI settings are enabled but MPEG settings and Animated GIF settings are disabled.

AVI settings

Note that you must select a compression format ("codec") when you export to an AVI file. Some compression formats come with Windows but you may also have installed 3rd party codecs explicitly or indirectly via other software. Note that if you are using a non-standard codec you may be unable to play the AVI file on a different computer (unless the codec is installed on it). If you want to be safe, export the frames uncompressed. This results in large files but you will be able to play them no matter what, and the quality is perfect since there is no compression. Such an AVI file can usually be compressed to a much smaller .zip file.

Frame rate may be set to specify the rate to use when playing back the saved AVI file. Color depth may be specified to vary between 8, 16, and 24 bits implying increased image quality but also increased storage requirements, loading time etc.

MPEG settings

Compression and quality parameters. The options Frames in GOP, P-frame distance and Target bit rate are expert options. If experimenting, first try to change the setting of Target bit rate.

Frames in GOP Frames in each MPEG group of pictures (encoding pattern).

Target bit rate Target bit rate of MPEG stream in bits/sec. A low bit rate implies high compression (and possible lower quality).

P-frame distance Distance between P-frames in MPEG encoding pattern.

GIF settings

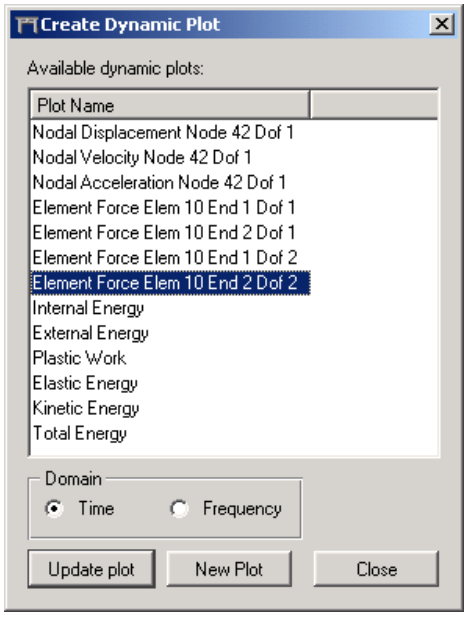
Settings for animated GIF output.

Loop # of times specifies the number of times the saved animated GIF sequence is to be repeated. Checking the "Loop forever" box causes the GIF sequence to repeat indefinitely.

Frame rate may be set to specify the rate to use when running the resulting animation from the saved file.

Transparent: When this box is checked the GIF is generated with transparent background color, selectable via the "Transparent color" combo box.

4.7.9 Dynamic Plot…



Shows a dialog that lets you display the dynamic plots available on the current model. Pick the plot you want to show, and chose if you want to show it in the time-domain or frequency domain.

Update Plot updates your current dynamic plot if you have any. If you don't it will create a new one for you.

New Plot creates a new plot with the input you have selected.

If no dynamic plots are available on your analysis, this item will be unavailable.

4.7.10 History Plot…

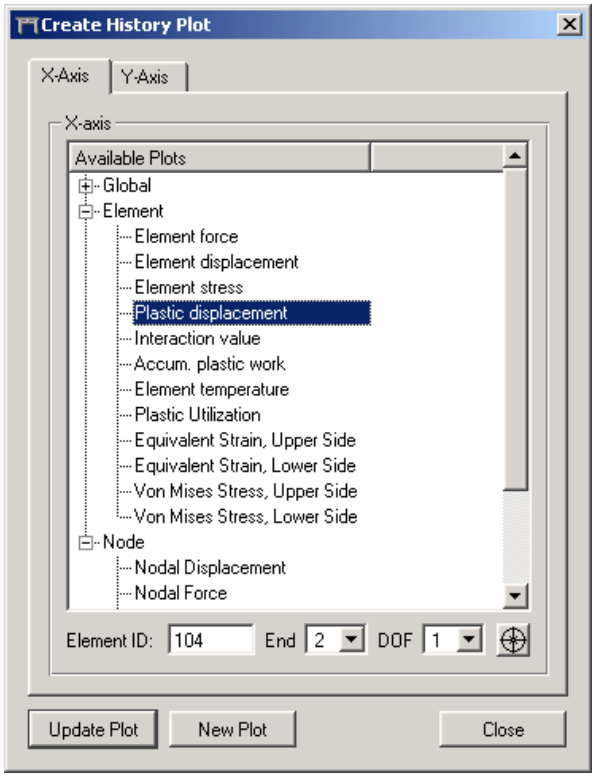
Lets you create a history plot. You have to choose parameters for both x-axis and y-axis. There are two tabs in the dialog, one for both axes. The manner in witch input values are selected is identical for the two axes.

A list of available plots is shown. The list is organized in Global, Element and Node plots. For each plot in the list, there may be a number of additional input parameters you need to supply.

If a plot needs an element or node id, the input box for this will be opened. The same goes for End and DOF.

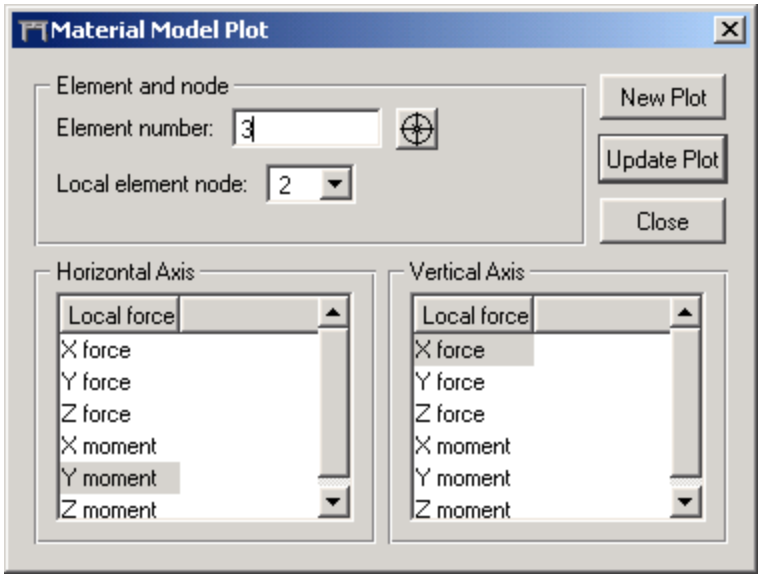
Element and Node Ids can be typed in manually, or you can pick them from the model. Press the Aim-icon in the dialog, and you can pick element id and end, or node id from the model by holding down CTRL and left clicking on the model.

Chose input parameters for both axis and press either Update Plot or New Plot.



Update Plot updates the current history plot. NB! If you haven't created any history plot, your control plot will be updated.
New Plot creates a new history plot with the input parameters chosen.

4.7.11 Material Plot…



The material Plot dialog lets you create a Material Model plot. You can type the Element ID and Element End you want the plot to represent. As in all dialogs requiring Element IDs, you can press the Aim-button and pick an Element from the model by pressing CTRL and left clicking with your mouse on the element you want.
You also need to specify the force of both the x- and y-axis. The default selection is Y Moment for the x-axis, and X Force for the y-axis.
New plot creates a new plot with the chosen input parameters.
Update Plot updates the current Material Plot. If no Material Plot exists, it creates a new one.

4.7.12 Start Logging Plot Commands…

Opens a file dialog where you may specify the name of an USFOS Script File (.usf). All commands that have been run manually (through the command window) or indirectly via actions in the graphical user interface are logged to the file until you stop logging commands. This is very useful if you want to create plots for a number of result files. Just open the first file and start logging commands before performing the plot actions. When you're done, stop the command logging. Then you may open the next result file and open and run the USFOS Script File you created. Then repeat this for the remaining result files. See the next section.

4.7.13 Stop Logging Plot Commands

See the previous section. Stops logging commands to file if command logging is activated. The USFOS Script File being logged to will be closed and can now be opened so that the commands in it will be run in sequence. See the next section.

4.7.14 Run Script File…

Opens a file dialog where you may select and open an USFOS Script File (.usf). All commands in the file will be run in sequence. See the previous section.

4.8 The Analysis Menu

4.8.1 USFOS Analysis Setup…

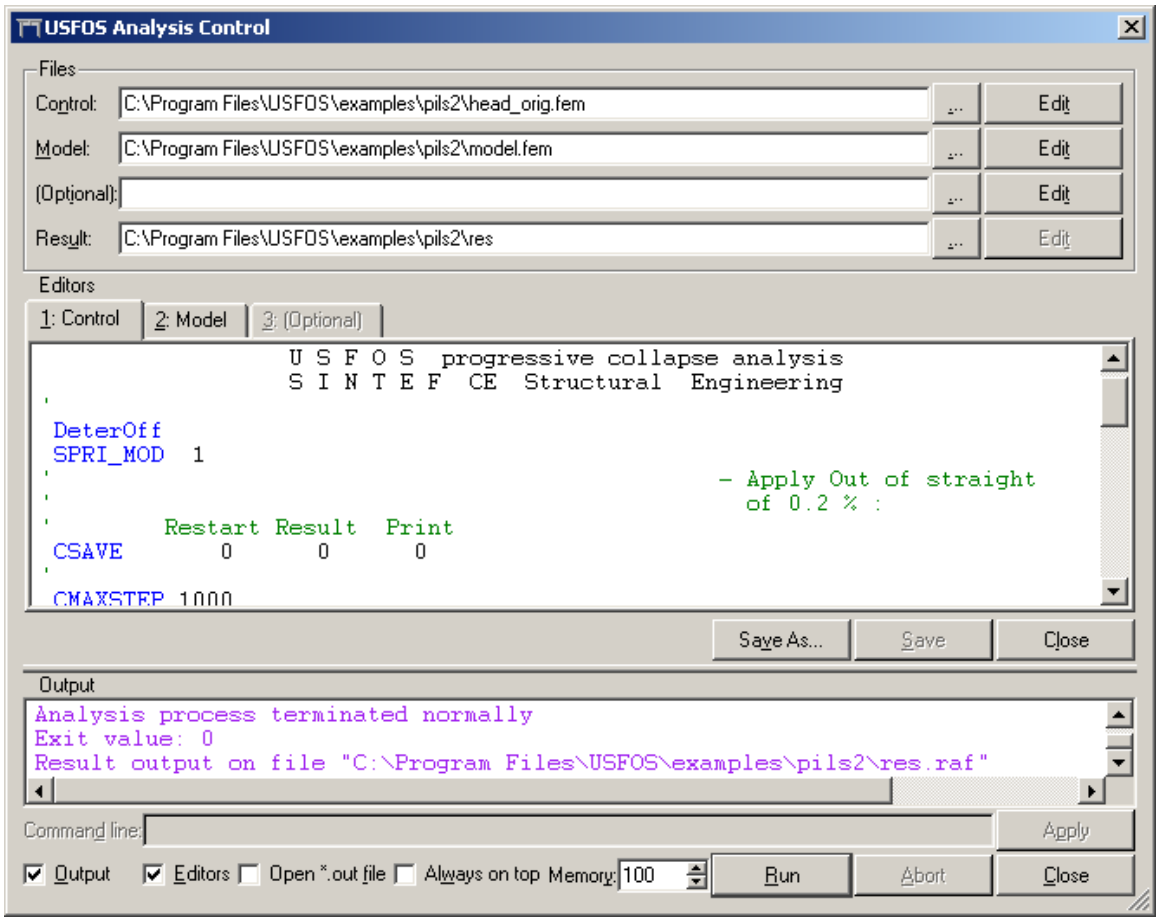
Opens the USFOS Analysis Setup Dialog. This dialog makes it easier to start the definition of input for the analysis from scratch by helping to provide a minimum of what is needed. The contents of the dialog may be stored in an USFOS Setup File (.uss) so that it can be used in a later session.

For more details, see section 3.3.3.

4.8.2 USFOS Analysis Control…

Opens the USFOS Analysis Control Dialog. This dialog makes it possible to edit input files, and to run an USFOS analysis. For a brief introduction illustrated by an example, see section 3.3.4. A more detailed description is given here.

The Analysis Control Dialog looks as illustrated by the following figure.



The upper part of the dialog contains edit controls where names of the input files and the result file should be given. They may be given manually or the corresponding browse buttons ([…]) may be clicked to bring up file dialogs for specification of file names. The name "res" will be suggested as name of the result file by default, which means that the results will be written to a file called "res.raf".

The middle part of the dialog contains USFOS editors (see section 4.2.8) for the input files (control file, model file and optional file). Once a filename has been given, clicking the corresponding "Edit" button will open the input file in the corresponding USFOS editor, and the input file may be edited and saved, perhaps to a different filename, before running an analysis. The "Save As…", "Save", and "Close" buttons act on the currently active editor, which is determined by the selected tab.

If there are no input files yet, they may be created from scratch with the USFOS editors. Just click "Edit", and the corresponding editor will be activated. Just click "Save" when the input file is ready. If no name has been given for the input file, a file dialog will ask for a filename and the filename will appear in the corresponding edit control after the file has been saved.

When the input file(s) have been specified, an analysis may be run. Just click the "Run" button, and USFOS will be started with the given input file(s) as input. The amount of memory (in number of million words) to be used by USFOS for the analysis may be specified using the spin button in the bottom.

Standard output from the analysis process is shown in the output window in the lower part of the dialog. This window may be hidden to make more space for the USFOS editors by unchecking the "Output" check button. This is also the case for the editors, which may be hidden by unchecking the "Editors" check button. Also note that the splitter control (thin horizontal bar) between the editors and the output window can be clicked and dragged vertically to change the heights of the editors and the output window.

If the analysis needs additional input from the user, the contents of the "Command line" control is sent to the standard input of the analysis process when "Apply" is clicked or the RETURN key is pressed.

The analysis process may be aborted by clicking the "Abort" button. When the analysis process is finished, Xact will load the result file automatically.

There is a check button called "Always on top". If it is checked, this dialog will always be on top of the main window. If it is unchecked, clicking in the main window will bring the main window in front of this dialog. If the "Open *.out file" check button is checked, the .out file corresponding to the .res file will be opened in the internal editor and displayed when the analysis process is finished. This may be useful if something goes wrong during the analysis. The .out file may indicate what the problem is.

4.8.3 FAHTS Analysis Control…

Opens the FAHTS Analysis Control Dialog. This dialog makes it possible to edit input files, and to run a FAHTS analysis. The dialog is identical to the USFOS Analysis Control Dialog, except for the fact that it runs FAHTS instead of USFOS, and that "optional" files are called "KFX files". See section 4.8.2 for more information.

4.8.4 STRUMAN Analysis Control…

Opens the STRUMAN Analysis Control Dialog. This dialog lets you edit input files and run a STRUMAN analysis. The dialog is identical to the USFOS Analysis Control Dialog, except for the fact that it runs STRUMAN instead of USFOS. See section 4.8.2 for more information.

4.9 The Window Menu

4.9.1 Animation Control

Here you can turn the display of the animation control on or off. Remember that the animation control is unavailable when no current animation is running.

4.9.2 Command Window

Turns the display of the Xact Command Window on/off. For a listing of available commands, see Appendix A, Quick Command Reference.

4.10 The Help Menu

4.10.1 About

Shows information like version numbers etc.

4.10.2 USFOS Input Commands…

Shows the on-line documentation of the USFOS input commands.

4.10.3 Graphical User Interface…

Shows the on-line documentation of the graphical user interface (this document).

4.11 The File Toolbar

4.11.1 Open

See Open File in The File Menu section.

4.11.2 Save Scene

See Save Scene in The File Menu section.

4.11.3 Print Scene

See Print in The File Menu section.

4.11.4 Print Current Plot

See Print in The File Menu section.

4.11.5 Select Region

Activates/deactivates “Select Region” mode. See 4.3.15.

4.12 The Misc. Toolbar

4.12.1 Clip

See Clip… in The Edit Menu section

4.12.2 Eigenmodes

See Eigenmodes…in The Result Menu section

4.12.3 Set up Animation

See Animate… in The Result Menu section

4.12.4 Animation Control

See Animation Control… in The Window Menu section

4.12.5 Command Window

See Command Window… in The Window Menu section

4.13 The Display Toolbar

4.13.1 Show as Surface

See Show as surface in The Display Menu Section

4.13.2 Show as Lines

See Show as lines in The Display Menu Section

4.13.3 Show as Points

See Show as points in The Display Menu Section

4.13.4 Show as Outline

See Show as outline in The Display Menu section

4.13.5 Show as Hidden Lines

See Show as hidden lines in The Display Menu section

4.14 The Plots Toolbar

4.14.1 Verification

See Verification Plot in The Verify Menu section.

4.14.2 History

See History Plot in The Result Menu section

4.14.3 Dynamic

See Dynamic Plot in The Result Menu section

4.14.4 Material Model

See Material Plot in The Result Menu section.

4.15 The View Toolbar

4.15.1 Zoom

The rubberband zoom. When you toggle this button, you can zoom in on a part of the model. Just left click and hold the mouse button down in the graphics view. You will begin to draw a rectangle. When you release the mouse button, the view will zoom in to the area in the rectangle. A small rectangle zooms with a great magnitude, and a large rectangle zooms in with less magnitude.

4.15.2 Zoom Out

Zooms out with a preset factor.

4.15.3 Zoom In

Zooms in with a preset factor.

4.15.4 Frame

Zooms in or out to fit the whole model in the view.

4.15.5 Isometric View

Sets the view to look at the model with origo in the center of the screen.

4.15.6 View From Z-axis

Views the model from the Z-axis. Straight on the XY-plane.

4.15.7 Set Rotation Point

Sets the rotation point once you have clicked the left mouse button while pointing at a position of your choice in the model. The model will now rotate around this point when you navigate.

4.15.8 Automatic Rotation Point

Sets the rotation point automatically. This means that the model will rotate around the center of the model's bounding box when you navigate. This is default.

4.15.9 Show Rotation Point

Toggles display of the rotation point, which is shown as a light gray sphere. Display of the rotation point is switched off by default.

4.16 The Navigate Toolbar

4.16.1 Rotate Up

Rotates the model 22,5 degrees upwards

4.16.2 Rotate Down

Rotates the model 22,5 degrees downwards

4.16.3 Rotate Right

Rotates the model 22,5 degrees right

4.16.4 Rotate Left

Rotates the model 22,5 degrees left

4.16.5 Rotate Clockwise

Rotates the model 22,5 degrees clockwise

4.16.6 Rotate Counter-clockwise

Rotates the model 22,5 degrees counter clockwise

4.16.7 Perspective

Toggles between perspective and parallel projection. Perspective is default.

Appendix A

Quick Command Reference

Required parameters are given inside brackets like this: <*parameter*>. Optional parameters given inside square brackets, like this [optional parameter]. When you need to choose one of several options for a parameter the options are divided by a pipe like this: [on|off].

Commands	**Parameters**	**Description**
File commands		
open	<*file name*>	Open .raf file, .fem file, or open as text file in editor
open raf	<*file name*>	Open .raf file
open fem	<*file name*>	Open .fem file
print plot		Print current plot
save viewattributes	<*file name*>	Save view attributes to file
load viewattributes	<*file name*>	Load view attributes from file
load labels	<*file name*>	Load user-defined labels from file
Display commands		
display hiddenlines	[<*from part*>, <*to part*>]	Show parts as hidden lines
display surface	[<*from part*>, <*to part*>]	Show parts as surface
display lines	[<*from part*>, <*to part*>]	Show parts as lines
display points	[<*from part*>, <*to part*>]	Show parts as points
display outline	[<*from part*>, <*to part*>]	Show parts as outline
display hide	[<*from part*>, <*to part*>]	Hide parts
display legend	[*on*	*off*]	Toggle or set display of legend
display elementnumbers	[*on*	*off*]	Toggle or set display of master element numbers

display nodenumbers	[*on* | *off*]	Toggle or set display of master node numbers
display highlightelement	<element ID> | none	Highlight master element or turn off highlighting
display highlightnode	<node ID> | none	Highlight master node or turn off highlighting
display mesh	[*on* | *off*]	Toggle or set display of mesh
display bgcolor	<*r, g, b*>	Set background color
display meshcolor	<*r, g, b*>	Set mesh color
display partcolor	<*r, g, b*> [<*from part*>, <*to part*>]	Set part color

Navigation commands

rotate right	[<*degrees*>]	Rotate view to the right
rotate left	[<*degrees*>]	Rotate view to the left
rotate up	[<*degrees*>]	Rotate view up
rotate down	[<*degrees*>]	Rotate view down
rotate clockwise	[<*degrees*>]	Rotate view clockwise
rotate counter clockwise	[<*degrees*>]	Rotate view counter clockwise
rotate setdefault	<*degrees*>	Set default rotation angle
zoom in	[<*factor*>]	Zoom in
zoom out	[<*factor*>]	Zoom out
zoom frame		Zoom model to fit frame
zoom setdefault	<*factor*>	Set default zoom factor
view from x		View model from x-axis
view from –x		View model from negative x-axis
view from y		View model from y-axis
view from –y		View model from negative y-axis
view from z		View model from z-axis
view from –z		View model from negative z-axis
view isometric 1		View model from 1st octant
view isometric 2		View model from 2nd octant
view isometric 3		View model from 3rd octant
view isometric 4		View model from 4th octant

view isometric 5		View model from 5th octant
view isometric 6		View model from 6th octant
view isometric 7		View model from 7th octant
view isometric 8		View model from 8th octant
perspective on		Use perspective projection
perspective off		Use parallel projection

Results commands

results step	<*step number*>	Select step
results steprange	<*from*>, <*to*>	Set step range
results showfringes	[*on* | *off*]	Toggle or set display of fringes
results fringe	<*1st name*>, <*2nd name*>, <*3rd name*> [,<*4th name*>]	Select fringe result
results fringerange	<*from*>, <*to*>	Set fringe range
results bar	<1st name>, <2nd name>, <3rd name> [,<4th name>]	Select bar result
results showbars	[on | off]	Toggle or set display of bars
results vector	<1st name>, <2nd name>, <3rd name> [,<4th name>]	Select vector result
results showvectors	[on | off]	Toggle or set display of vectors
results displacements	[*on* | *off*]	Toggle or set displacements
results scaledisplacements	<*factor*>	Scale displacements
results list eigenperiods		List eigen periods
results modeshape	<*eigen period index*> [*steps*]	Set up mode shape animation for eigenperiod
results animate		

Animation commands

anim play	Play animation

anim stop		Stop animation
anim pause		Pause animation
anim playbackward		Play animation backward
anim stepforward		Step animation forward
anim stepbackward		Step animation backward
anim maxfps	<*fps*>	Set maximum number of frames per second

Plot commands

plot history	<*x-name* | *x-number*>, <*x-item*>, <x-end | x-int.sec>, <x-dof | x-int.pnt>, <*y-name* | *y-number*>, <*y-item*>, <y-end | y-int.sec>, <y-dof | y-int.pnt>	Create history plot
plot dynamic	<*name* | *number*> [*freq*]	Create dynamic plot
plot materialmodel	<*name* | *number*>, <*element ID*>, <*element end*>, <*x-dof*>, <*y-dof*>	Create material model plot
plot verification	<*name* | *number*>, <*node ID*>, <*x-dof*>	Create verification plot
plot list history		List available history plots
plot list dynamic		List available dynamic plots
plot list materialmodel		List available material model plots
plot list verification		List available verification plots
plot save	<filename>, <file type>	Save plot as image to file
plot savedata	<filename>	Save plot data to text file

Set commands

set colors	[*on* | *off*]	Turn display of set colors on or off
set list		List available sets

set show	[*ID*]	Show set given by ID. -1 means all
set hide	[*ID*]	Hide set given by ID. -1 means all
set entiremodel		Show the entire model

Clip commands

clip add	<*element ID*>	Add element to clipped model
clip remove	<*element ID*>	Clip away element
clip undo elementremove		Undo last element removal
clip reset		Reset clipping
clip showcomplement		Show the complement of the clipped model
clip showclipped		Show the clipped model

Export commands

export vtf	<*filename*>, [*current* | *all*]	Export current or all steps to VTF file. Default is all

Misc commands

verify node	<*ID*>	Display verification info for node
verify element	<*ID*>	Display verification info for element
verify general	<*text info ID*>, <*element ID*>, <*end*>, <*node ID*>, <*DOF*>, <*group ID*>	Display general verification info
quit		Close the application

Appendix B

USFOS Label File Format (.usl)

This file type is an ASCII text file containing user-defined labels for nodes and/or elements. This is useful if you want to add textual information to specific parts of the model. The sections Read Labels From File… and Labels… describe how to read user-defined labels from file and how to manipulate the labels.

The file contains one block per label group. A label group may contain any number of element labels and/or any number of node labels. The format is as follows:

```
BeginLabel

Heading        <name of label group>

ElementLabel   <element ID>   <label text>
ElementLabel   <element ID>   <label text>
[..]
ElementLabel   <element ID>   <label text>

NodeLabel      <node ID>      <label text>
NodeLabel      <node ID>      <label text>
[..]
NodeLabel      <node ID>      <label text>

EndLabel
```

Example:

```
#                                        - First Label Set
#
BeginLabel
Heading    Model Overview
#
#             ElemID      Label Text
ElementLabel     1         Guide Pin
ElementLabel 20573         South
ElementLabel 21120         North
ElementLabel 20846         East
ElementLabel 21387         West
#
"             NodeID      Label Text
NodeLabel        1         Elevation  4.520
NodeLabel    13955         Elevation -1.710

EndLabel

#                                        - 2'nd  Label Set
#
BeginLabel
Heading    Inspection Summer 2004
#
#             ElemID      Label Text
ElementLabel 20738         Excellent Weld
ElementLabel 20788         Normal Weld
#
#             NodeID      Label Text
NodeLabel    52286       Gasket Renewed

EndLabel
```

USFOS Utility Configuration File ("Xact_Utils.usc")

If the file "Xact_Utils.usc" is present in the same directory as the USFOS GUI executable when USFOS GUI is launched, information about available utilities will be read on startup. If the file contains valid information about utilities, the File menu will have a sub menu called "Run Utility" from which you may launch utilities, and a menu item called "Utilities…", which brings up a dialog containing a list of the utilities and optionally descriptions of the utilities and hyperlinks to user manuals.

The file contains one block per utility. The format is as follows:

```
BeginUtilityTool
ProgramName     <program name>
ProgramFile     <executable>
Manual          <name of manual file>
Description     <description line 1>
Description     <description line 2>
[…]
Description     <description line N>
Parameter       <default parameter>
ParameterInfo   <parameter description line 1>
ParameterInfo   <parameter description line 2>
[…]
ParameterInfo   <parameter description line N>
```

Example:

```
BeginUtilityTool
ProgramName             StruMan
ProgramFile             struman.exe
Manual                  struman.pdf
Description             Model File Conversion,
Description             Unit Conversion, Model Manipulation, etc.
Description             NOTE! Unsupported utility Software.
ParameterInfo           Specify memory size.
ParameterInfo           Default memory (30 million words).
EndUtilityTool

BeginUtilityTool
ProgramName             Soil
ProgramFile             soil.exe
Manual
Description             Gensod data to Usfos.
Description             NOTE! Unsupported utility Software.
EndUtilityTool
```

USFOS Script File (.usf)

Files of this type are simply ASCII files containing commands (see Quick Command Reference). One line can contain only one command (with parameters) and a command can't be distributed over several lines.

Such files can be written manually or generated automatically through command logging (see Start Logging Plot Commands…). USFOS Script Files can be opened and run. For more information, see Run Script File….