Phase^2

2D finite element program for calculating stresses and estimating support around underground excavations

Phase^2 Interpret Program
Reference Manual

© 1998 - 2001 Rocscience Inc.

Phase^2

2D finite element program for calculating stresses and estimating support around underground excavations

Phase^2 Interpret Program
Reference Manual

© 1998 - 2001 Rocscience Inc.

Phase^2

2D finite element program for calculating stresses and estimating support around underground excavations

Phase^2 Interpret Program
Reference Manual

© 1998 - 2001 Rocscience Inc.

Table of Contents

Phase2 Interpreter 5
Analysis Results 6

File Menu

Opening a File 7
Opening Multiple Files 7
Opening the Modeler from Interpret 8
Opening the Log File 8
Info Viewer 9
Exporting Images 10

Edit Menu

Copy 11
Picture Format 12
AVI Movie Creation 13

View Menu

Zoom All 14
Zoom In 15
Zoom Out 16
Zoom Window 17
Zoom Mouse 18
Zoom Excavation 19
Pan 20
Quick Zoom 21
Data Tips 23
Grayscale 24
Display Options 25
Element Numbers 27
Node Numbers 27
Bolt Numbers 28
Liner Numbers 29
Joint Numbers 29
Discretizations 30
Boundaries 30
Bolts 30
Liners 30
Restraints 31
Tractions 31
Forces 31
Stress Trajectories 32
Elements 32
Shrink Elements 32
Shrink Contours 33
Bolt Elements 33
Deform Mesh 33
Deform Contours 34
Deform Boundaries 34
Deformation Vectors 35
Scale Factor 36
Contour Options 37
Contour Range 38

Contour Format....39
Contour Mode....40
Auto Options....41
Legend Options....42
Ruler....43
Stress Block....43
Ground Level....43
Snap....44
Grid....46

Data Menu

Overview of Data....47
Sigma 1....49
Sigma 3....49
Sigma Z....49
Strength Factor....50
Displacement Contours....51
Ubiquitous Joints....52
Strain Contours....53
Yielded Element Contours....54
User Data....55
Defining User Data....56
Editing User Data....57
Deleting User Data....57
User Data Configuration File....58
User Data Variables, Functions and Properties....59
Select Composite Component....60
Stage Settings....61
Differential Data....62
Animate Stages....63

Analysis Menu

Show Values....64
Show Yield....65
Add Yield Line....67
Add Iso-Line....68
Delete Iso-Line....69

Query Overview

Query Overview....70
Add Material Query....71
Query Boundary....72
Query Excavations....73
Editing Queries....74
Deleting Material Queries....75
Writing Query Files....76
Reading Query Files....77

Graph Menu

Graph Material Queries....78
Graph Bolt Data....80
Graph Liner Data....81
Graph Joint Data....82
Plot in Excel....83
Chart Properties....84
Copy Data to Clipboard....85

Tools Menu

Tools Overview 87
Contour Label 88
Measure 89
Axes 90
Line 91
Arrow 92
Rectangle 93
Circle 94
Text Box 95
Picture from file 96
Scale Picture 97
Save Tools File 98
Open Tools File 99
Delete Drawings 100
Delete All Drawings 101
Move, Re-size, Format Drawing Tools 102
New Window 104
Right-Click Menus 105
Selecting 106
Shortcut Keys 107

Index

Index 108

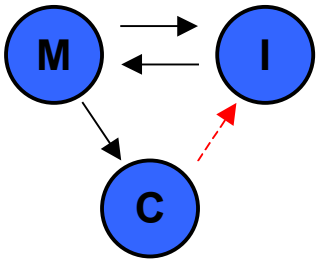
Phase2 Interpreter

The PHASE2 program consists of 3 program modules:

- MODEL
- COMPUTE
- INTERPRET

You are now using the INTERPRET module. INTERPRET is the post-processing module used for data visualization and interpretation of the PHASE2 analysis results .

MODEL, COMPUTE and INTERPRET will each run as standalone programs. They also interact with each other as illustrated in the schematic below:



- COMPUTE and INTERPRET can both be started from within MODEL.
- COMPUTE must be run on a file before results can be analyzed with INTERPRET (red arrow)
- MODEL can be started from INTERPRET .

Analysis Results

INTERPRET allows you to contour analysis results directly on the model, including:

- SIGMA 1
- SIGMA 3
- SIGMA Z
- STRENGTH FACTOR
- HORIZONTAL DISPLACEMENT
- VERTICAL DISPLACEMENT
- TOTAL DISPLACEMENT
- STRENGTH FACTOR WITH UBIQUITOUS JOINTS
- USER-DEFINED DATA

In addition to the contour plots, data can be displayed on the model or graphed for:

- USER-DEFINED LINES (MATERIAL QUERIES) WITHIN THE ROCK MASS
- BOLTS
- LINERS
- JOINTS

Opening a File

To view the results of a PHASE2 analysis, select **Open** from the **File** menu or the Standard toolbar. PHASE2 files have an .FEA filename extension.

Before results can be displayed in INTERPRET, the finite element analysis must have been performed using COMPUTE. If not, an error message will be displayed when you attempt to open the file in INTERPRET.

INTERPRET will always open a file by displaying a **Sigma 1** contour plot.

Opening Multiple Files

As PHASE2 is a fully MDI (multiple document interface) application, any number of files can be simultaneously open for viewing.

Furthermore, any number of windows (views) can be opened for each file, using the New Window option in the **Window** menu.

Opening the Modeler from Interpret

While in INTERPRET, you may return to or start the PHASE2 MODELER by selecting **Open Modeler** from the **File** menu, or by selecting the **Open Modeler** button available on the Standard toolbar.

This will start the MODELER and automatically open the PHASE2 file which was active in INTERPRET. If the MODELER was already open, then you will simply be transferred to the MODELER and the PHASE2 file which was active in INTERPRET will be active in MODEL.

Since MODEL can be activated/started from INTERPRET, and INTERPRET can be activated/started from MODEL, this allows the user to quickly make changes to a model, re-compute results, and view the new results.

Opening the Log File

Each time a PHASE2 analysis is run, a LOG file is created which summarizes a few key analysis parameters from the finite element analysis. While in INTERPRET, this file can be viewed by selecting **Log File** from the **File** menu.

This will display the LOG file in its own view, in a similar format to the **Info Viewer**. The LOG file information can then be copied to the clipboard, saved to a file etc, if desired. See the Info Viewer topic for more information.

Info Viewer

The **Info Viewer** option provides a convenient summary of model and analysis information in its own view. Use the scroll bar on the right to scroll through the information. The **Info Viewer** behaves like any other view in PHASE2 – ie. it can be re-sized, tiled, minimized, maximized and closed.

The **Info Viewer** option is available in both the PHASE2 MODEL and PHASE2 INTERPRET programs.

- In the PHASE2 MODEL program, only model information is listed.
- In the PHASE2 INTERPRET program, model information is listed, as well as analysis information.
- In both PHASE2 MODEL and PHASE2 INTERPRET, the **Info Viewer** is available in the toolbar or the **File** menu.

The **Info Viewer** information can be saved by the user in a variety of ways, for including in reports, documents, etc. **Info Viewer** text can easily be added to views of the model, with the **Text Box** option.

Copy to Clipboard

The **Info Viewer** information can be copied to the clipboard using the **Copy** option in the toolbar or the **Edit** menu, or by right-clicking in the view and selecting **Copy**. From the clipboard, the information can be pasted into word processing programs for report writing.

Save to File

The **Info Viewer** information can be saved to a text file (*.TXT) or rich-text format file (*.RTF), by right-clicking in the view and selecting the desired **Save As** option, or by selecting the desired **Save As** option from the File menu, while the **Info Viewer** is the active view.

Note: a rich-text format file preserves the text formatting that you see in the Info Viewer as it is displayed in PHASE2. A plain text file saves the text only, with no formatting.

Printing

The **Info Viewer** information can also be sent directly to your printer using the **Print** option in the **File** menu.

Adding Info Viewer Text to a View

In the PHASE2 INTERPRET program, any information in the **Info Viewer** (for example, Material Properties) can easily be added to a view of the model, using the *Auto-Text* feature of the **Text Box** option in the **Tools** menu. See the Text Box topic for details.

Exporting Images

Various options are available for exporting image files in PHASE2.

Export Image File

The **Export Image File** option in the **File** menu allows the user to save the current view directly to one of four image file formats:

JPEG (*.jpg)

Windows Bitmap (*.bmp)

Windows Enhanced Metafile (*.emf)

Windows Metafile (*.wmf)

NOTE: the **Export Image File** option is also available in the default right-click menu on all model or graph views.

Copy to Clipboard

The current view can also be copied to the Windows clipboard using the **Copy** option in the **Edit** menu. From the clipboard, images can be pasted directly into word or image processing applications. See the Copy topic for details.

NOTE:

- To capture the entire screen to the Windows clipboard, you can use the *Print Screen* key on your keyboard. This is useful if you wish to capture multiple views in a single image, for example after tiling the views.
- Pressing the *Alt* + *Print Screen* keys together will capture the application window only to the clipboard (useful if the PHASE2 INTERPRET program window is not maximized on your desktop).

Black and White Images (Grayscale)

The **Grayscale** option in the toolbar or the **View** menu, automatically converts the current view to Grayscale, suitable for black and white image requirements. This is useful when sending images to a black and white printer, or for capturing black and white image files. The Grayscale option works as a toggle, and all previous colour settings of the current view will be restored when **Grayscale** is toggled off.

Copy

The **Copy** option in the **Edit** menu or the toolbar, allows you to copy the contents of the active view to the Windows clipboard. This can then be pasted into word-processing or paint programs, for further editing, report writing etc.

Images can be saved in four different formats, available in the Picture Format option in the **Edit** menu:

1. Bitmap, original size, colour
2. Bitmap, size x 2, monochrome
3. Bitmap, size x 4, monochrome
4. Enhanced metafile (vector based format)

You should set the desired picture format BEFORE using **Copy**. Changing the picture format AFTER using **Copy** will have no effect on the clipboard contents.

Note:

- The window border and title bar will NOT be copied, only the contents of the window.
- The highlighted dark blue title bar indicates the active view.
- If the Info Viewer is the active view, then the **Copy** option will copy the **Info Viewer** text to the clipboard. By default, ALL text in the **Info Viewer** will be copied to the clipboard. If you only want selected text, then first highlight the desired text with the mouse, and then select **Copy.**
- See the Exporting Images topic for further information about saving images with PHASE2.

Picture Format

The **Picture Format** option in the Edit menu determines what image format will be used with the Copy option.

Four options are available:

1. Bitmap, original size, colour
2. Bitmap, size x 2, monochrome
3. Bitmap, size x 4, monochrome
4. Enhanced metafile (vector based format)

Select the desired picture format BEFORE you use **Copy**. Changing the picture format AFTER using **Copy** will have no effect on the clipboard contents.

AVI Movie Creation

You may create AVI format movie files in PHASE2 as follows:

1. Select the **Start Recording** option from the **Edit** menu or the toolbar.
2. You will see a dialog. Follow the instructions in the dialog, and select *OK*.
3. You will see another dialog. Select a video compressor and select *OK*.
4. Select **Add Frame** from the toolbar or the **Edit** menu, for each view that you would like to add to the AVI movie.
   - For example, to create a movie of results at different stages, select each stage tab, and select **Add Frame** at each stage.
5. When you are finished recording, select **Stop Recording** from the toolbar or the **Edit** menu. The AVI file will then be saved.

Zoom All

The **Zoom All** option will automatically scale the model in the active view, so that all parts of the model are visible.

Zoom All is useful for quickly restoring the whole model to view, if you have been using the other Zoom options (eg. Zoom In , Zoom Out , Zoom Window , Zoom Mouse , Zoom Excavation), or the Pan option.

Zoom All is available in:

- The **Zoom** sub-menu of the **View** menu,
- The Zoom toolbar,
- The default right-click menu,
- The F2 function key.

Zoom In

Zoom In will increase the magnification of the model in the active view, so that the new view area is 90% of the old view area.

Zoom In is useful when you quickly want to zoom in on the model. You can keep selecting **Zoom In** as many times as necessary. If you want to zoom in on a specific area of interest, then use the Zoom Window option, rather than **Zoom In**.

Zoom In is available in:

- The **Zoom** sub-menu of the **View** menu,
- The Zoom toolbar,
- The F5 function key,
- The *Home* key.

Zoom Out

Zoom Out decreases the magnification of the model in the active view, so that the new view area is ~111% of the old view area.

Zoom Out is useful when you quickly want to zoom out on the model. You can keep selecting **Zoom Out** as many times as necessary. If you want to restore the entire model to view, then you may want to use the Zoom All option, rather than **Zoom Out**.

Zoom Out is available in:

- The **Zoom** sub-menu of the **View** menu,
- The Zoom toolbar,
- The F4 function key,
- The *End* key.

Zoom Window

Use **Zoom Window** to quickly zoom in to any specific area of the model. To use **Zoom Window**:

1. Select the **Zoom Window** option.
2. Use the left mouse button to select one corner of the area you wish to zoom in on. When you move the mouse, you will notice a rectangular window opening up.
3. Use the left mouse button to select the opposite corner of the area to be zoomed.
4. The view will then be updated, scaling the area that you have just selected to fit in the view. The view will always be updated to show the ENTIRE window you have selected, even if it is not the same "shape" as the view.

Tips

- The corners of the zoom window can be entered as x,y coordinate pairs in the prompt line at the bottom right of the screen. This is useful if you wish to zoom into exactly the same region at different times, or on different models.
- If you are not satisfied with the result of using **Zoom Window**, you can always try to improve the view by using Zoom In , Zoom Out , Zoom Mouse , Pan , or using **Zoom Window** again. If all else fails, select Zoom All and start over!
- You may find Zoom Excavation a useful alternative to **Zoom Window**, for automatically zooming in to your excavation(s).
- To cancel the **Zoom Window** option (before you have entered the second window corner) use the **Escape** key, or right-click and select **Cancel**.

Zoom Window is available in:

- The **Zoom** sub-menu of the **View** menu,
- The Zoom toolbar,
- The default right-click menu.

Zoom Mouse

The **Zoom Mouse** option (a.k.a. Mouse Zooming, dynamic zooming) is another convenient method for zooming in or out. To use **Zoom Mouse**:

1. Select the **Zoom Mouse** option.
2. Press and hold the left mouse button.
3. Move the mouse up (to zoom in) or down (to zoom out) while keeping the left mouse button pressed.
4. Release the left mouse button when the zoom is satisfactory.
5. To exit the **Zoom Mouse** option, use the **Escape** key, or right-click and select **Done**.

Tips

- If you have a mouse with a wheel, you can quickly zoom in or out by moving the wheel forward or backward.
- Note that **Zoom Mouse** always zooms the model about the center of the view. To zoom in to a specific area of interest, you may find it necessary to use Zoom Window rather than **Zoom Mouse**, unless the area you wish to zoom is already at the center of the view.

Zoom Mouse is available in:

- The **Zoom** sub-menu of the **View** menu,
- The Zoom toolbar,
- The default right-click menu.

Zoom Excavation

Zoom Excavation is a useful shortcut for quickly zooming in to the excavation(s) in your model, without having to use the Zoom Window option.

If your model consists of multiple excavations, **Zoom Excavation** will take into account the combined extent of all excavations.

Zoom Excavation is available in:

- The **Zoom** sub-menu of the **View** menu,
- The Zoom toolbar,
- The F8 function key.

Pan

The **Pan** option allows you to move the model in any direction within the view. To **Pan**:

1. Select the **Pan** option.
2. Press and hold the left mouse button.
3. Move the mouse around while keeping the left mouse button pressed. You will see that the model moves around with the cursor.
4. Release the left mouse button when the view is satisfactory.
5. You can repeat steps 2 – 4 as many times as desired, until you are satisfied with the results.
6. To exit the **Pan** option, use the **Escape** key, or right-click and select **Done**.

Tips

- If you have a 3-button mouse (or a mouse wheel) you can quickly pan by pressing and holding the middle button of the mouse, while moving the mouse.
- You will find **Pan** a very useful and frequently used option. For example, it is very handy for making adjustments to the model position in the view after using the Zoom Window option, if the view is not quite where you intended.
- Since the **Pan** option operates as a "mode", you can use Zoom In , Zoom Out , Zoom All , or Zoom Excavation , without leaving **Pan** mode. This is a useful shortcut for manipulating the view.

Pan is available in:

- The **View** menu,
- The Zoom toolbar,
- The default right-click menu.

The arrow keys (up / down / left / right) can also be used to pan.

Quick Zoom

The **Quick Zoom** option, available in the toolbar or the **Zoom** sub-menu of the **View** menu, allows the user to:

1. Save zoom coordinates, so that specific views can easily be re-created for screen captures, demonstration purposes, etc.
2. Automatically zoom to the same coordinates in multiple views.

Saving Zoom Coordinates

To save zoom coordinates with the **Quick Zoom** dialog:

1. First zoom the view to the desired coordinates.
2. Select the **Quick Zoom** option. NOTE:
   - while the Quick Zoom dialog is active, you may still use the regular Zoom options (ie. Zoom All, Zoom In, Zoom Out, the mouse wheel, etc), to adjust the view further if necessary)
   - you may also enter coordinates in the **Quick Zoom** dialog, and select the *Zoom* button in the dialog.
3. When you have zoomed to the desired location, select the *Save* button in the **Quick Zoom** dialog.
4. You will then see another dialog which allows you to enter a *Name* for the saved coordinates. NOTE: by default, the filename will appear as the zoom name, however, you may enter any name. Also, a filename extension is NOT necessary, since the coordinates are actually saved in the registry on your computer.
5. Select *OK* in this dialog, and the *Name* you entered will now appear in the *Zoom To* list in the **Quick Zoom** dialog.
6. Select *Done* in the **Quick Zoom** dialog.
7. The zoom coordinates are now permanently saved. You may now zoom to these coordinates at any time, by selecting **Quick Zoom**, and selecting the *Name* of your zoom coordinates, from the *Zoom To* list in the dialog.

Zooming on Multiple Views

The **Quick Zoom** settings can be applied to multiple views, as follows:

1. In the Quick Zoom dialog, select either:
   - *All Views (This File)* or
   - *All Views (All Files)*

   from the *Apply To* list in the **Quick Zoom** dialog.
2. Select a *Name* from the *Zoom To* list, or enter the desired zoom coordinates.

3. Select the *Zoom* button in the **Quick Zoom** dialog, and the selected zoom coordinates will be applied to all views of the current file, or all views of all files, according to the *Apply To* selection.

The *All Views (This File)* option will apply the **Quick Zoom** settings to all views of the current file, if you have multiple views open for a file.

The *All Views (All Files)* option will apply the **Quick Zoom** settings to all views of all files.

This capability is very useful if for example, you have multiple views of different stages of a staged model, or if you have multiple views of different files, and you wish to zoom in to the same area of each view.

Data Tips

Data Tips allow the user to quickly access information about model input parameters, and analysis results, by simply hovering the mouse over the desired model entities or locations. This includes the following information:

- material properties
- support properties
- field stress
- contour data values at any point
- and other information (eg. iso-line values)

By default, the **Data Tips** display mode is OFF (ie. data tips are not displayed). To enable **Data Tips**, select the desired option from the **Data Tips** sub-menu in the **View** menu, or select an option from the status bar at the bottom of the application window. The following options are available.

Data Tips Max

The **Data Tips Max** option will provide the maximum amount of information about the desired entities. For example, when the mouse is hovered over any material region, a complete list of material properties for that material will appear. This also applies to support properties (ie. hover the mouse over bolts or liners to obtain support properties).

Data Tips Min

The **Data Tips Min** option will provide a minimal amount of information, for example, material names, excavation staging information, and support installation information.

Data Tips Query

The **Data Tips Query** option will display exact interpolated values of data at any point on the contour plots, corresponding to the current cursor location.

Data Tips Off

Since you will not always want **Data Tips** to appear while using INTERPRET, you may disable the Data Tips display by selecting **Data Tips Off**. This is always the default setting for Data Tips.

Note:

- Field Stress information is displayed by hovering the cursor over the Stress Block, while using either **Data Tips Max** or **Data Tips Min**

- In general, experiment with placing the cursor over various model and view elements, while in **Data Tips** mode, to discover what information is available for the various model and view elements.

Grayscale

The **Grayscale** option automatically converts the current view to Grayscale, suitable for black and white image requirements. You may find this useful when sending images to a black and white printer, or for capturing black and white image files.

Grayscale is available in the toolbar or the **View** menu. The **Grayscale** option works as a toggle, and all previous colour settings of the current view will be restored when **Grayscale** is toggled off.

Display Options

The **Display Options** dialog contains a wide variety of options for displaying information on the model, and customizing the display of the model and analysis results. Note:

- The **Display Options** dialog works a little differently than a regular dialog. It is known as a "roll-up" dialog, and can be left up on the screen while performing other tasks. The "arrow" button in the upper right corner can be used to "roll-up" or "roll-down" the dialog, without closing it. This allows the user to minimize / maximize the dialog, to keep it out of the way when not needed.
- Tip – the dialog can also be minimized / maximized by DOUBLE-CLICKING on the title bar of the dialog.

A "roll-up" dialog is useful when working with multiple views, since the user can apply different options to different views, without closing the dialog.

The following **Display Options** are available:

- Element Numbers
- Node Numbers
- Bolt Numbers
- Liner Numbers
- Joint Numbers
- Discretizations
- Boundaries
- Bolts
- Liners
- Restraints
- Tractions
- Forces
- Stress Trajectories
- Elements
- Shrink Elements
- Shrink Contours
- Bolt Elements
- Deform Mesh
- Deform Contours
- Deform Boundaries

- Deformation Vectors
- Scale Factor

You will find **Display Options** to be a very frequently used tool in PHASE2 INTERPRET, and the user is encouraged to experiment with all of the options that are available. Many different combinations of options can be applied to the model for many different data interpretation purposes.

*NOTE that **Display Options** operates on a per view basis, so that if multiple views are open, each view can have its own set of **Display Options** applied.*

Display Options is available in:

- the **View** menu
- the default right-click menu
- some of the most commonly used display options are available in the Display toolbar

The default **Display Options** are in effect whenever a file is opened, or a New Window is created. However, the settings in the **Display Options** dialog can be saved using Auto Options in the **View** menu, and recalled again when needed.

Element Numbers

The *Element Numbers* option in the **Display Options** dialog, will display the number of each finite element, as used in the PHASE2 data files (the .FEA and .R?? files).

When *Element Numbers* are displayed, they will overlap in dense regions of the mesh. This is normal, use the Zoom Window option to zoom in and make numbers readable.

You may also want to toggle on the display of *Elements*, when viewing *Element Numbers*, if you are performing a detailed inspection / troubleshooting of the mesh.

Node Numbers

The *Node Numbers* option in the **Display Options** dialog, will display the node numbers of the finite element mesh, as used in the PHASE2 data files (the .FEA and .R?? files).

When *Node Numbers* are displayed, they will overlap in dense regions of the mesh. This is normal, use the Zoom Window option to zoom in and make numbers readable.

You may also want to toggle on the display of *Elements*, when viewing *Node Numbers*, if you are performing a detailed inspection / troubleshooting of the mesh.

Note: you may notice that the node numbering begins at 4. Don't worry, this is correct, and is due to the automatic meshing process, which requires 4 nodes located "outside" of the external boundary. These 4 "invisible" nodes are numbered 0, 1, 2, 3.

Bolt Numbers

Bolt Numbers are used in conjunction with the Graph Bolt Data option, and allow the user to associate bolts with their data curves on a graph of bolt data. Note that the bolt numbers do NOT exist in the PHASE2 files, although they do correspond to the ordering of the bolt data in the PHASE2 files.

Tips

- The *Bolt Numbers* are automatically turned ON when bolt data is graphed, so it is not necessary for the user to do this each time bolt data is graphed.
- *Bolt Numbers* are NOT automatically turned OFF, even if all graphs of bolt data are closed. To clear the display of bolt numbers, the user will have to toggle *Bolt Numbers* off in the Display Options dialog.
- NOTE: the location of the bolt number on each bolt is important, as it identifies which end of the bolt corresponds to the end of each curve on a bolt graph. This may seem obvious if pattern bolting was used to install the bolts, however, it becomes more important when spot bolting is used, since the user can enter a spot bolt starting at either end of the bolt, which determines where the bolt number will appear on the bolt.
- If *Bolt Numbers* overlap when displayed, use Zoom Window to zoom in and make the numbers readable.

Liner Numbers

Liner Numbers are used in conjunction with the Graph Liner Data option, and allow the user to associate points on liners with the corresponding points on liner data curves. Note: the *Liner Numbers* are the same as the node numbers along the boundaries (you will see this if you toggle *Liner Numbers* and Node Numbers on at the same time).

Tips

- The *Liner Numbers* are automatically turned ON when liner data is graphed, so it is not necessary for the user to do this each time liner data is graphed.
- *Liner Numbers* are NOT automatically turned OFF, even if all graphs of liner data are closed. To clear the display of liner numbers, the user will have to toggle *Liner Numbers* off in the Display Options dialog.
- If *Liner Numbers* overlap when displayed, use Zoom Window to zoom in and make the numbers readable.

Joint Numbers

Joint Numbers are used in conjunction with the Graph Joint Data option, and allow the user to associate joints with their data curves on a graph of joint data. Note that the joint numbers do NOT exist in the PHASE2 files, although they do correspond to the ordering of the joints in the PHASE2 files.

Tips

- The *Joint Numbers* are automatically turned ON when joint data is graphed, so it is not necessary for the user to do this each time joint data is graphed.
- *Joint Numbers* are NOT automatically turned OFF, even if all graphs of joint data are closed. To clear the display of joint numbers, the user will have to toggle *Joint Numbers* off in the Display Options dialog.
- NOTE: the location of the joint number on each joint is important, as it identifies which end of the joint corresponds to the end of each curve on a joint graph.

Discretizations

The *Discretizations* option in the **Display Options** dialog, will display the finite element discretization along the model boundaries. Small coloured crosses will appear at each node location along the boundaries.

Boundaries

The *Boundaries* options in the **Display Options** dialog, toggles ON or OFF the display of boundaries (eg. Excavation, External, Material, Stage and Joint Boundaries). Default is ON, but you may want to toggle *Boundaries* OFF to view contour plots without boundaries, for example.

Bolts

The *Bolts* option in the **Display Options** dialog, will toggle on or off the display of bolts on the model. You may find this useful if the display of bolts interferes with the viewing of data (eg. contours, yielded elements, etc.) or other elements of your model (eg. liners, joints, etc.)

Liners

The *Liners* option in the **Display Options** dialog, will toggle on or off the display of liners on the model. (Liners are indicated by thick coloured line segments along boundaries where they have been applied).

Tip

If you are having trouble seeing *Liners* on the model:

- *Liners* may be easier to see if you turn the display of Boundaries off in the **Display Options** dialog.
- If bolts are installed on the same segments as liners, you may also want to toggle the display of Bolts off in the **Display Options** dialog.
- For maximum visibility, you can always turn off the display of contours in the Contour Options dialog, by setting the contour *Mode* to Off.

Restraints

The *Restraints* option in the **Display Options** dialog, will display all nodal restraints applied to the model.

- A node restrained in both the X and Y directions will display a triangular "pinned" symbol.
- A node restrained in the X direction only, will display a vertical "roller" symbol.
- A node restrained in the Y direction only, will display a horizontal "roller" symbol.
- Non-zero "displacement" values will be represented by an arrow and the displacement magnitudes in the X and Y direction.

In addition, if Infinite Elements are being used to model the external boundary conditions, the *Restraints* option will show this by displaying arrows along the nodes of the external boundary where the Infinite Elements are applied.

Tractions

The *Tractions* option in the **Display Options** dialog, will display all tractions you have applied to the model. Arrows represent the direction of the tractions and the numerical magnitude is also displayed.

The *Scale Factor* allows you to scale the size of the traction arrows on the screen. The scale factor is for display purposes only, and does not affect the magnitude of the traction loading.

Forces

The *Forces* option in the **Display Options** dialog, will display all nodal forces you have applied to the model (these are the forces you have added with the Add Nodal Forces option in the Loading menu of the PHASE2 MODELER). An arrow symbol and a numerical value represent the direction and magnitude.

Stress Trajectories

The *Stress Trajectories* option in the **Display Options** dialog, displays the *principal stress trajectories* in the form of a long line for σ1 and a short line for σ3. The trajectories are placed and centered at every finite element node.

If a square dot ■ appears instead of the stress trajectory symbol, this indicates that conditions are hydrostatic (σ1 = σ3) at that point.

If you are viewing differential data the stress trajectories will show the orientation of the differential principal stresses at each node.

Stress Trajectories is also available in the Display toolbar.

Elements

The *Elements* option in the **Display Options** dialog, toggles the display of the finite element mesh on or off.

Elements is also available in the Display toolbar.

Shrink Elements

The *Shrink Elements* option in the **Display Options** dialog, will "shrink" each finite element such that common sides between elements are drawn twice and offset from each other. This allows the user to check the integrity of the mesh if a problem is suspected. Or it can simply be used as an alternate way of displaying the mesh.

If it is not already selected, the Elements option will automatically be toggled on when *Shrink Elements* is selected, since the mesh must be displayed in order to view the shrunken elements.

Shrink Elements can be used in conjunction with Shrink Contours if desired.

Shrink Contours

The *Shrink Contours* option in the **Display Options** dialog, will "shrink" the contours within each finite element of the mesh. *Shrink Contours* can be used in conjunction with Shrink Elements, although the two options are independent of each other (ie. *Shrink Contours* can be applied regardless of whether or not the mesh is displayed).

Bolt Elements

For bonded bolts (*Fully Bonded*, *Plain Strand Cable* or *Swellex / Split Sets*), "bolt elements" are defined by the intersections of each bolt with the finite element mesh. The *Bolt Elements* option in the **Display Options** dialog, will display small yellow cross markers at each bolt/mesh intersection. (If you toggle the display of Elements on, and zoom in if necessary, you will see that the bolt elements correspond to intersections with the finite elements).

Note: for *End Anchored* bolts, the entire bolt is considered a single element, so the *Bolt Elements* option will only display the yellow cross markers at each end of an *End Anchored* bolt.

You may find it useful to display *Bolt Elements* when you are graphing bolt data, to aid in interpretation of the bolt graph data curves.

Deform Mesh

The *Deform Mesh* option in the **Display Options** dialog, allows the user to view the mesh with the displacement at all nodes magnified by the *Scale Factor*. The *Auto Scale Factor* button will provide a default scale factor, but the user can enter any number they wish for the *Scale Factor*.

The display of Elements will automatically be toggled on when *Deform Mesh* is selected.

Deform Contours

The *Deform Contours* option in the **Display Options** dialog, will deform the contour plotting on the model, by magnifying the displacement at all nodes by the *Scale Factor* in the **Display Options** dialog. *Deform Contours* can be used in conjunction with Deform Mesh, although the two options can be applied independently of each other.

The *Auto Scale Factor* button will provide a default scale factor, but the user can enter any number they wish for the *Scale Factor*.

Deform Boundaries

The *Deform Boundaries* option in the **Display Options** dialog, will draw the deformed shape of the excavation and external boundaries. The deformation can be magnified by any amount entered in the *Scale Factor*. The *Auto Scale Factor* button will provide a default scale factor, but the user can enter any number they wish for the *Scale Factor*.

Deform Boundaries is also available in the Display toolbar.

Deformation Vectors

The *Deformation Vectors* option in the **Display Options** dialog, will draw displacement vectors at each node of the mesh. This can be a useful supplement to the contour plots, as the displacement vectors show the relative magnitude and direction of displacements on the model.

The *Scale Factor* allows the user to magnify the displacement vectors by any amount. The *Auto Scale Factor* button will provide a default scale factor.

NOTE:

- If you are viewing contours of X Displacement then the *Deformation Vectors* option will only display the x-component of the total displacement vector at each node.
- If you are viewing contours of Y Displacement then the *Deformation Vectors* option will only display the y-component of the total displacement vector at each node.
- For all other PHASE2 contour plots (eg. Sigma1, Strength Factor, Total Displacement, etc.) the *Deformation Vectors* option will display the total displacement vector at each node.
- If you are viewing differential data , *Deformation Vectors* will display the differential displacement vectors (X, Y or Total, as described above).

Deformation Vectors is also available in the Display toolbar.

Scale Factor

The *Scale Factor* for deformations allows the user to magnify the deformations by any amount when applying the following options in the Display Options dialog:

Deform Mesh

Deform Contours

Deform Boundaries

Deformation Vectors .

NOTE: after you enter a *Scale Factor*, you must select the *Apply* button, if you want to see the results without closing the dialog.

Auto Scale Factor will calculate a default *Scale Factor*, however, if this is not suitable, the user can always enter a *Scale Factor*.

The *Scale Factor* for boundary conditions allows the user to scale the size of traction arrows on the display.

Contour Options

The **Contour Options** dialog contains a wide variety of options for customizing the display of contours on the model. Note:

- The **Contour Options** dialog works a little differently than a regular dialog. It is known as a "roll-up" dialog, and can be left up on the screen while performing other tasks. The "arrow" button in the upper right corner can be used to "roll-up" or "roll-down" the dialog, without closing it. This allows the user to minimize / maximize the dialog, to keep it out of the way when not needed.
- Tip – the dialog can also be minimized / maximized by DOUBLE-CLICKING on the title bar of the dialog.

A "roll-up" dialog is useful when working with multiple views, since the user can apply different options to different views, without closing the dialog.

The following **Contour Options** are available:

- Contour Range
- Format
- Mode

Contour Options is available in:

- the **View** menu
- the default right-click menu (this is the most convenient way to access **Contour Options**)

A great variety of contouring appearances can be achieved by the use of the **Contour Options**. The user is encouraged to experiment with all of the available options, to become familiar with all of the possibilities.

The default **Contour Options** are in effect whenever a file is opened, or a New Window is created.

Contour Range

For staged models, PHASE2 automatically calculates default ranges for each data type, for:

- each individual stage
- for all stages combined

Auto-Range (this stage)

Each stage will have it's own default contour range (for each data type)

Auto-Range (all stages)

All stages will use the same contour range, determined from the maximum range of data at all stages.

Custom Range

The user can enter custom values for the minimum and maximum range values, the number of intervals, or the actual interval value.

Contour Format

Several Auto-Format options are available:

- DOS-style – uses exactly the same 7 colours and number of intervals as the original Rocscience DOS-based programs (eg. PHASE2 v. 3.0)
- 16-colour display – uses 14 colours and intervals (black and white are not used, which is why only 14 instead of 16 are actually used)
- Hot to Cold – uses a blue to red range of contouring, simulating a "hot to cold" effect.
- Hot Spots – uses a white to red range of contouring, simulating a "hot spot" effect.
- Photo-E – uses alternating white and black intervals, simulating a "photoelastic" effect.
- Black to white – uses black to white contouring, and 7 intervals.
- Black lines – uses black lines, mode is automatically changed to Lines

You may find the "Black to white" and "Black lines" options useful for black and white screen captures.

Define Auto-Format

The Define Auto-Format option gives the user maximum flexibility to define their own customized Auto-Format contour option. To use this option:

1. Select the Define Auto-Format button in the Contour Options dialog
2. Use the options in the Define Auto-Format dialog, to create a new Auto-Format style.
3. When you are finished, select the Save button, and save the new Auto-Format (it will be saved in a file with an *.RAF filename extension).
4. Your newly defined Auto-Format will then appear in the Auto-Format list in the Contour Options dialog, and can be used like any of the default Auto-Format options.

Contour Mode

Contours can be displayed as Filled regions (the default), or as Lines, or they can be completely turned Off. Turning contours Off can be useful when you wish to view information without the contours (eg. deformation vectors, yielded elements, etc.)

The Materials mode turns off the contours and displays the material colours, if any. This allows the user to check on the material assignments without having to return to the modeler.

Auto Options

Auto Options allows the user to save the status of all viewing options in a separate file, which can be recalled at a later time. The following options are saved:

1. All Display Options (except the Scale Factors)
2. All Contour Options (except the Contour Range settings)
3. The Ruler , Stress Block and Ground Level status.

To save the current viewing options:

1. Select View -> Auto Options -> Save Current Options
2. Enter a file name and save the file. The file will be saved as a ".pof" file (PHASE2 Options File).
3. The file name will then appear in the list of Auto Options. These options can then be applied at any time by simply selecting View -> Auto Options -> (file name)

To quickly restore the default PHASE2 viewing options, select View -> Auto Options -> Default Options. Note that the default viewing options are always initially in effect:

- Whenever a file is opened in INTERPRET
- Whenever a New Window is created.

Auto Options is very useful because it allows the user to recall their favourite viewing options with a single mouse click. This saves the user the trouble of constantly using the Display Options and Contour Options dialogs. Finally, remember that view options operate on a per view basis, so that each view can have a different set of options applied.

Legend Options

The **Legend** option in the **View** menu allows the user to:

- show / hide the display of the legend
- customize the number of decimal places displayed in the legend, and also on contour labels and queries

NOTE:

- **Legend Options** operates on a per view basis, so each view (window) can have its own customized Legend Options.
- The *Number of Decimal Places* in **Legend Options**, can be customized for each Data Type (For example, Sigma 1, Strength Factor and Displacement, can each have an individually customized Number of Decimal Places). Furthermore, PHASE2 will automatically save the latest settings you have chosen for each Data Type, so that you do not have to reset the Number of Decimal Places each time you start the program.

Tips

- A convenient shortcut for accessing the **Legend Options** dialog, is to right-click ON the Legend, and select **Legend Options** from the popup menu.
- The legend can be moved anywhere on the view by hovering over it with the mouse. When you see the four-arrow "move" icon, press and HOLD the left mouse button, and move the legend anywhere in the view.
- The legend can also be lengthened or shortened. If you hover the mouse over the MIDPOINT of the top or bottom of the Legend, you will see a double-arrow icon. Press and HOLD the left mouse button, and drag the mouse up or down, to re-size the legend. As you resize the legend, the number of labeled contour intervals may increase or decrease, depending on the number of intervals defined in the Contour Options dialog. Experiment with this to see how it works.
- In addition to displaying the contour information, the Legend in PHASE2 also displays other information such as yielded element symbols and colours. These will appear at the bottom of the Legend, when they are applicable.

Ruler

The **Ruler** option in the **View** menu toggles the display of the ruler on or off.

The ruler appears at the left and bottom edge of each view.

Stress Block

The **Stress Block** option in the **View** menu toggles the display of the "stress block" on or off.

The stress block indicates the relative magnitude and direction of the IN-PLANE principal field stresses.

Ground Level

The **Ground Level** option in the **View** menu toggles the display of a dotted line marking the Ground Surface Elevation (as entered in the Field Stress dialog in the PHASE2 MODELER).

Ground Level is only applicable if you are using Gravity Field Stress. It is not applicable if you are using Constant Field Stress.

Snap

"Snapping" means that if the mouse cursor is moved close to a location that can be snapped, the cursor will jump to the exact location. (The x, y coordinate of the location will be shown on the status bar). In the PHASE2 INTERPRETER snapping is applicable when using the Add Material Query option. It can also be used with the Drawing Tools options.

There are many different ways in which Snap can be used in PHASE2:

Vertex Snap

If you hover the mouse over any vertex of the model boundaries, the cursor will change to a circle, indicating that you may snap exactly to the vertex location. Note that this also includes bolt vertices.

Grid Snap

If the Grid is displayed, you can snap exactly to grid points when the mouse is hovered over a grid point.

Boundary Snap

When the cursor is hovered over a model boundary, the cursor will change to an X. If you click the mouse when the X is displayed, the point will be snapped exactly on a boundary. Note that this also applies to bolts (ie. a point can be snapped exactly on a bolt when you see the X).

Ortho Snap

If the line you are currently drawing is nearly horizontal or nearly vertical, a small "right-angle" symbol will be displayed beside the cursor, and the line will snap exactly to the horizontal or vertical direction. This indicates that you can snap the line exactly along a horizontal or vertical direction.

Note that the Ortho Snap is useful in conjunction with Boundary Snap, since it allows you to simultaneously draw an exactly horizontal or vertical line, and also snap the point exactly on a boundary.

Extension Lines

When the cursor is hovered over a model boundary or a bolt, you will notice that dotted "extension lines" will automatically be drawn on the screen, extending from the boundary where the cursor is located. While these lines are on the screen, you may snap exactly to these extension lines, in exactly the same manner as the actual model boundaries. These extension lines can be useful as guides for adding queries or drawing tools to the screen. Note that these lines disappear when the current action is completed, they do not remain on the model.

Tips

There are numerous ways of toggling the SNAP option:

- By selecting the **Snap** option in the **View** menu
- By clicking on the SNAP option on the status bar
- While using the **Add Material Query** option, or any of the Drawing Tools options, a right-click will bring up a context menu with the SNAP option.
- By typing **s** or **n** in the prompt line followed by *Enter*, to toggle Snap or No Snap
- The F9 function key will also toggle the Snap option.

Grid

The **Grid** option allows the user to display a grid of points or lines over the model. To turn the grid on or off, select **Grid** from the **View** menu, and toggle the Show Grid checkbox. Grid settings can also be changed (*Grid Style*, *Spacing*, and *Colour*) in this dialog.

Uses of the Grid

- Since the grid is always the same size (in world coordinates) in all views, this will help give you a sense of scale if one view is "zoomed-in" and another view is "zoomed-out".
- If the Snap option is enabled, then the grid intersections can be "snapped" to. You may find this useful when using some of the INTERPRET options (eg. **Add Material Query**, or when using the Drawing Tools options in the **Tools** menu)

Tips

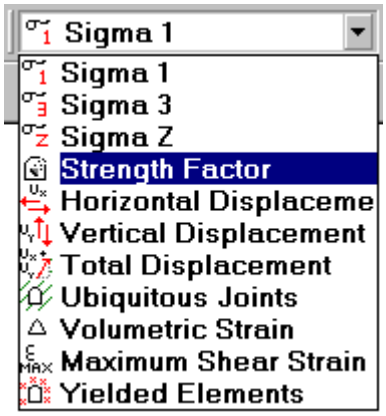
- A shortcut for turning the grid on or off is to left-click on the word GRID on the status bar (ie. the first box to the left of the x,y coordinate display at the lower right of the screen).
- The F7 function key can also be used to toggle the grid on and off.
- The Grid Settings dialog can be accessed by right-clicking on the word GRID on the status bar.
- The grid will eventually "disappear" if you keep zooming "out" on a model. This prevents the grid from becoming so dense that it hides the model.

Overview of Data

The primary means of data interpretation after a PHASE2 analysis, is the viewing of data contours directly on the model. The PHASE2 INTERPRET program allows the user to display contoured data from the finite element analysis, by selecting a data type from the drop-down list in the toolbar.



By default, when a file is opened (or when a new window is opened for an already open file), **Sigma 1** *contours will always be displayed on the model*. To view contours of other data, use the mouse to select the desired data from the drop-down list in the toolbar.



Basic Data

The following data types will always be available for contouring after a PHASE2 analysis:

- SIGMA 1
- SIGMA 3
- SIGMA Z
- STRENGTH FACTOR
- HORIZONTAL (X) DISPLACEMENT
- VERTICAL (Y) DISPLACEMENT
- TOTAL DISPLACEMENT
- STRENGTH FACTOR WITH UBIQUITOUS JOINTS
- VOLUMETRIC STRAIN
- MAXIMUM SHEAR STRAIN

In addition to the basic data listed above, other data types may appear in the drop-down list in the toolbar, as described below.

Yielded Elements

If any of the materials used in the analysis have been defined as having Plastic material properties, then **Yielded Elements** can be contoured on the model. See the Yielded Element Contours topic, for a description of how this contouring is achieved.

Effective Stress Analysis

If pore pressure has been included in the analysis (eg. Piezometric lines or a water pressure grid have been defined for the model), then effective stress data and pore pressure will also be available in the toolbar list:

- EFFECTIVE SIGMA1
- EFFECTIVE SIGMA3
- EFFECTIVE SIGMAZ
- PORE PRESSURE

User Data

Finally, with the **User Data** option, INTERPRET allows the user to contour almost any data used in the analysis, including user-defined functions of all input or output analysis variables. If **User Data** has been defined, then the names of these variables will appear at the end of the list of variables available from the drop-down list in the toolbar. See the User Data topic for details.

Note:

1. If the user wants to simultaneously view more than one data type for a single file (eg. Sigma 1 and Sigma 3), then they must select New Window from the Window menu. *This will generate a new window with a Sigma 1 contour plot*. The user can then plot any data they wish in the new window.
2. Any number of new windows can be opened for a given file, allowing the user to view different data in different windows, or to view different parts of the model in different windows, for example.
3. *Selecting a new data option from the toolbar, updates the contour plot in the active view only*.
4. If multiple windows have been opened, the user can select Tile Horizontally or Tile Vertically from the Window menu, to automatically tile the windows within the application window.
5. Many options are available in the Contour Options dialog, allowing the user to change the style, range, etc. of the contour plots. The Contour Options operate on a per view basis, so that each view can have its own set of contour options applied.

Sigma 1

The **Sigma 1** option will plot contours of the MAJOR IN-PLANE PRINCIPAL STRESS around the excavation(s).

Whenever a file is opened, or when a new window is opened for an already open file, the default plot will always be a **Sigma 1** contour plot.

Remember that the in-plane **Sigma 1** may not always be the major principal stress in 3-dimensions – if the value of Sigma Z is *greater than* **Sigma 1** at a given point, then the in-plane **Sigma 1** will actually be the 3-d intermediate principal stress.

Sigma 3

The **Sigma 3** option will plot contours of the MINOR IN-PLANE PRINCIPAL STRESS around the excavation(s).

Remember that the in-plane **Sigma 3** may not always be the minor principal stress in 3-dimensions – if the value of Sigma Z is *less than* **Sigma 3** at a given point, then the in-plane **Sigma 3** will actually be the 3-d intermediate principal stress.

Sigma Z

The **Sigma Z** option will plot contours of the OUT-OF-PLANE PRINCIPAL STRESS.

Remember that **Sigma Z** is not necessarily the intermediate principal stress – depending on the in-plane values of Sigma 1 and Sigma 3 at a given point, **Sigma Z** could be the 3-dimensional major, intermediate or minor principal stress.

If you are analyzing an AXISYMMETRIC problem, then **Sigma Z** will represent the circumferential stress around the excavation.

Strength Factor

The **Strength Factor** option will plot contours of strength factor around the excavation(s).

The strength factor is calculated by dividing the rock strength (based on your failure criteria) by the induced stress at every point in the mesh. All three principal stresses have an influence on the strength factor (Sigma 1 , Sigma 3 , and Sigma Z), so the strength factor in PHASE2 can be considered 3-dimensional.

In the case of ELASTIC materials, the strength factor can be less than unity, since overstressing is allowed. In the case of PLASTIC materials, the strength factor is always greater than or equal to unity.

NOTE that when TENSION is indicated in the strength factor contours, this means that Sigma 3 is less than the calculated negative stress (tension) cutoff for the failure criterion in use.

Displacement Contours

The three displacement options in the INTERPRET toolbar list, allow you to plot contours of displacement in:

- The Horizontal (X) direction
- The Vertical (Y) direction
- The TOTAL displacement ($\sqrt{X^2 + Y^2}$)

NOTE that the contouring of displacements is always based on the **absolute magnitude** of displacements, whether you are viewing X, Y or TOTAL displacement.

To view the *direction* of displacements, the user can toggle on Deformation Vectors in the Display Options dialog. Note that, when viewing the deformation vectors:

- For Horizontal Displacement contour plots, only the X-component of the total displacement vector at each node will be displayed.
- For Vertical Displacement contour plots, only the Y-component of the total displacement vector at each node will be displayed.
- For all other PHASE2 contour plots (eg. Total Displacement, Sigma1, Strength Factor, etc.) the total displacement vectors will be displayed.

Ubiquitous Joints

The **Ubiquitous Joints** option allows you to account for jointing in the surrounding rock by including ubiquitous joints in the strength factor calculations.

- When you select **Ubiquitous Joints** from the INTERPRET toolbar list, a strength factor will be contoured, which accounts for the Ubiquitous Joint properties currently defined in the **Define Ubiquitous Joints** dialog.
- The **Define Ubiquitous Joints** dialog, is available in the toolbar or the Data menu, and allows the user to define the properties of up to 3 sets of Ubiquitous Joints.

The following properties may be entered in the **Define Ubiquitous Joints** dialog.

Number of joint sets

You may select One, Two, or Three, to enter properties for up to three different joint sets.

Slip Criterion

You may select either Mohr-Coulomb or Barton-Bandis strength parameters for the ubiquitous joints.

Plot minimum of

The **Ubiquitous Joint** strength factor calculation works as follows:

- At each node of the mesh, the strength factor is calculated for EACH joint set using the Mohr-Coulomb or Barton-Bandis slip criterion properties you have entered.
- If you have defined more than one joint set (ie 2 or 3 joint sets), then the MINIMUM of these represents the ubiquitous joint strength factor at each node.
- The "Joints only" option will ignore the rock mass strength, and contour the strength factor for the ubiquitous joints ONLY.
- The "Joints and rock" option will contour the LESSER of the ubiquitous joint strength OR rock mass strength at each node.

To close the dialog and plot the contours, select the *Display* button.

Tips

- You can change the ubiquitous joint properties at any time, with the **Define Ubiquitous Joints** dialog. The current ubiquitous joint properties will be displayed in the dialog. Edit the properties you wish to change, and select *Display*, and a new **Ubiquitous Joints** contour plot will be created.

- When viewing ubiquitous joint strength factor plots, if the Stress Block is displayed, the ubiquitous joint slip planes will be indicated in the Stress Block. This is useful as a quick check of the angles you have entered for the Inclination of each joint set. If the angle(s) appear wrong, check the inclination(s) you have entered in the **Ubiquitous Joints** dialog.

NOTE that ubiquitous joint modeling in PHASE2 is a post-analysis feature, using the stresses calculated from the finite element analysis – the strength factor calculations have no effect on the stresses or displacements, which have already been calculated.

Strain Contours

Contours of strain can be viewed by selecting either **Volumetric Strain** or **Maximum Shear Strain** from the INTERPRET toolbar list.

Note that strain data is available as of PHASE2 version 5.0. If you are viewing results files calculated with PHASE2 version 4.0 or earlier, then the **Volumetric Strain** and **Maximum Shear Strain** options will not be available. If you wish to view strain contours in this case, then simply re-run the analysis using the latest version of PHASE2, and the strain data will then be available for viewing.

Yielded Element Contours

The **Yielded Elements** contouring option, allows the user to contour the degree of yielding in the rock mass around excavations. The **Yielded Elements** option will only be available in the toolbar list in INTERPRET, if at least one material has been defined with Plastic material properties. If all materials are Elastic, then **Yielded Elements** contouring will not be available.

The **Yielded Element** contours are determined as follows:

1. At each node of the finite element mesh:
   - the total number of elements connected to a node, is determined.
   - the number of yielded elements connected to a node, is determined.
2. The "percent yielding" at a node, is then simply determined by the number of yielded elements, divided by the total number of elements, attached to a node. For example, if the total number of elements attached to a node is 6, and the number of yielded elements attached to the node is 2, then the "percent yielding" at that node, is 2 / 6 = 33.33 percent.

   NOTE: for the purpose of determining the number of yielded elements at a node, an element may fail in shear only, tension only, or both shear and tension. No distinction is made between the mode of yielding, for the purpose of the yielded element contours.
3. The values of "percent yielding", obtained as described above, are then contoured like any other data type available in PHASE2 (eg. Sigma 1, Strength Factor etc).

Tips

There are various other ways of displaying yielding in PHASE2 INTERPRET.

- For example, the yielded elements may be displayed directly with the Show Yield option.
- The zone of yielding around an excavation, can be highlighted by displaying a Yield Line around a selected area.

The **Yielded Element** contours can be considered an alternative representation of yielding in the rock mass, and can be used in conjunction with these other options.

User Data

With the **User Data** option, the user is not limited to viewing principal stresses and displacements only, but can create contour plots of:

- Any stress tensor component
- Any displacement component
- Write a mathematical expression containing these components, and view the results of this function

Creating and Plotting User Data

To view contours of **User Data**:

1. You must first use the **Define User Data** option, available in the toolbar or the Data menu, to define the desired **User Data**. See the Defining User Data topic for details.
2. Once **User Data** has been defined, it will appear at the END of the drop-down data list in the INTERPRET toolbar. It can then be selected like any other data type (eg. Sigma 1, Strength Factor etc), and a contour plot of the selected **User Data** will be generated.

NOTE:

- When you first install PHASE2, a few **User Data** definitions have already been created for you. These will appear in the INTERPRET toolbar data list, and in the **Define User Data** dialog.
- **User Data** contour plots may take longer to generate than other PHASE2 data (eg. Sigma 1, Displacement, etc.), because extra calculation is required to generate the **User Data**. When you see the "hourglass" icon, just be patient while PHASE2 prepares your **User Data** for plotting.

For further information, see the following topics:

Defining User Data

Editing User Data

Deleting User Data

User Data Variables, Functions and Properties

User Data configuration file (PH2.CFG)

Defining User Data

To create a new **User Data** definition:

1. Select **Define User Data** from the toolbar or the **Data** menu.
2. Select the *Add* button in the **User Data** dialog.
3. You will see the **Define User Data Equation** dialog. This lists all of the variables, material properties, and functions which are available for defining user data equations. (Select the tabs to view all of the available variables, functions etc)
4. Enter a *Name* for the *Equation* you will be creating (this can be anything you wish).
5. Enter an Equation using the available variables, functions, properties.
6. Select *OK* on this dialog, and you will be returned to the **User Data** dialog. The *Name* of the *Equation* you just defined will appear in the list of *Existing User Data*.
7. To plot the user data you have just defined, select the *Display* button. To save the user data without plotting, select *Done*. To exit without saving any changes, select *Cancel*. To change or delete user data, select *Edit* or *Delete*.

Example

1. Select the *Add* button in the **User Data** dialog.
2. In the **Define User Data Equation** dialog, enter *Maximum Shear* in the *Name* box.
3. In the *Equation* box, type (s1 – s3) / 2.0
4. Select *OK*.
5. In the **User Data** dialog, *Maximum Shear* will now appear in the list of *Existing User Data*, and its *Equation* will appear in the *Equation* box below the list.
6. To plot *Maximum Shear*, make sure it is highlighted in the list of *Existing User Data*, and select the *Display* button. A contour plot of *Maximum Shear* will be generated.
7. *Maximum Shear* will also now appear in the drop-down data list in the INTERPRET toolbar, and can be selected for plotting at any time, like any other PHASE2 data (eg. Sigma 1, Strength Factor etc)

NOTE -- there are various checks to ensure that a valid expression is entered as a **User Data** equation, however, the checks may not cover all possible cases, so the user should be careful to type in a valid expression, to avoid unexpected results.

Editing User Data

To **Edit** existing user data definitions:

1. In the **User Data** dialog, highlight the Existing User Data you would like to modify. (Use the left mouse button or the up/down arrow keys to highlight a name in the *Existing User Data* list).
2. Select the *Edit* button.
3. The **Define User Data Equation** dialog will appear with the *Name* and *Equation* of the user data which you selected, already displayed.
4. Edit the *Name* and/or *Equation*, and select *OK*, and the changes will appear in the **User Data** dialog.
5. Select *Done* or *Display* in the **User Data** dialog, to save the changes. The changes will also appear in the drop-down data list in the INTERPRET toolbar.

Deleting User Data

To **Delete** existing user data definitions:

1. In the **User Data** dialog, highlight the Existing User Data you would like to delete. (Use the left mouse button or the up/down arrow keys to highlight a name in the *Existing User Data* list).
2. Select the *Delete* button.
3. The highlighted selection will immediately be deleted from the list. Repeat steps 1 and 2 for each user data definition you would like to delete.
4. To make the delete permanent, select the *Done* button. This will close the dialog and modify the PH2.CFG file, deleting the definitions from the file. If you select *Cancel*, no definitions will be deleted and the PH2.CFG file will not be modified. This is useful if you accidentally *Delete* a definition that you wished to keep.
5. All deleted user data definitions, will be removed from the drop-down data list in the INTERPRET toolbar, and will no longer appear in the list.

User Data Configuration File

The **User Data** definitions are stored in a file named PH2.CFG, which you should find in your PHASE2 installation directory. Whenever you Add , Edit or Delete user data definitions, and then select *Display* or *Done* in the **User Data** dialog, the PH2.CFG file gets updated with the latest user data definitions.

When you install the PHASE2 program, a few **User Data** definitions will already be created for you in the PH2.CFG file.

User Data Variables, Functions and Properties

Since all of the **User Data** variables, intrinsic functions, operators and material properties that can be used in PHASE2 are listed in the **Define User Data Function** dialog, they will not be re-listed here. However, here are some points to remember:

Variables

In the **Define User Data Equation** dialog, under the *Variables* tab, it is noted that "d" can be added in front of any variable (eg. "ds1") to denote the differential result, and also that "r" can be added in front of any variable to denote the reference result (eg. "rs1"). The following example is given to clarify this situation:

Example (using variable s1, major principal stress):

If the Stage Settings for the view were Stage = 4 and Reference Stage = 2, then:

- Variable **s1** = stage 4 results
- Variable **rs1** = stage 2 results
- Variable **ds1** = differential (stage 4 –stage 2) results (*see note below).

NOTE:

1. To use differential or reference stage variables, the *Reference Stage* in the Stage Settings option must be greater than zero (ie. you must already be viewing differential data)
2. *DIFFERENTIAL PRINCIPAL STRESSES: In general, ds1 is NOT equal to s1 – rs1 (and ds3 is NOT equal to s3 – rs3). This is because the differential principal stresses are obtained by subtracting the stress tensors, and calculating the differential principal stress tensor, from which ds1 and ds3 are obtained. (This is necessary since simple subtraction of the principal stress *magnitudes* at each node would not take into account differences in *orientation* of the principal stresses, at different stages).
3. For all other variables, the differential result IS equal to the visible stage – reference stage values. For example, the differential strength factor variable dsf = sf – rsf.

Select Composite Component

The **Select Composite Component** option in the **Data** menu, allows the user to select individual layers of a composite liner, for the purposes of viewing analysis results in individual layers.

By default, ALL layers of a composite liner are initially selected. This means that if liner graphs are generated (for example, an Axial Force graph), data for ALL liner layers will be plotted on the same graph. However, the user may wish to only view results for a single layer of a Composite Liner. This is the purpose of the **Select Composite Component** option, which allows the user to select any individual layer of a Composite Liner for the viewing of results. This includes:

- Graphs generated with the Graph Liner Data option.
- Display of liner yielding with the Show Yield option.

To use the **Select Composite Component** option:

1. Choose **Select Composite Component** from the **Data** menu.
2. Select a *Composite Liner Component* (ie. liner layer or joint) from the drop-down list.
3. Select *OK*.
4. Now when you use the **Graph Liner Data** option or the **Display Yielded Liners** option, the results which are displayed will apply to the selected component of the Composite Liner.
5. NOTE: when a Composite Liner component is selected with the **Select Composite Component** option, the liner colour displayed on the model, will be the colour of the selected component (as defined in the Define Liner Properties dialog or the Define Joint Properties dialog), and not the colour of the overall Composite Liner (defined in the Define Composite Liner dialog).

Viewing Joint Data

If a Composite Liner includes a joint (ie. slip allowed between the first layer and the rock mass), note that the joint data is graphed with the **Graph Liner Data** option, and NOT with the **Graph Joint Data** option, since the joint is considered to be a part of the Composite Liner.

NOTE:

- A Composite Liner in PHASE2, is a liner which may be composed of multiple layers of material. For information about defining Composite Liners in PHASE2, see the Define Composite Liner topic in the Help system for the PHASE2 MODEL program.

Stage Settings

The **Stage Settings** option in the **Data** menu allows you to set a Reference Stage > 0 if you wish to view differential data between any two stages of a multi-stage model. To view differential data:

1. Select **Stage Settings** from the **Data** menu.
2. In the **Stages** dialog, select a *Reference Stage*. You can also select a *Visible Stage*, but it is not really necessary to use this, since the stage tabs are always visible on each view. Note that the dialog automatically checks that the *Reference Stage* is less than the *Visible Stage*.
3. Select *OK* or press *Enter*. Notice that the tabs for changing stages (at the lower left of the view) now indicate the *Visible Stage* AND the *Reference Stage*. For example, if your model has 3 stages and the *Reference Stage* = 1, then only two stage tabs will be available: [Stage 2 – Stage 1] and [Stage 3 – Stage 1].
4. All data in the view will now be in differential format.
5. To set the *Reference Stage* = 0 again or to set it to a different stage, repeat steps 1 to 3.

Note that each view can have its own **Stage Settings**, allowing different Reference Stages in different views.

All data can be contoured in differential format, including stress, displacement, strength factor and user data. See the Differential Data topic for more information.

Minimum Animation Time

The **Stage Settings** dialog also allows you to set the *Minimum Animation Time* for the Animate Stages option.

Differential Data

For multi-stage models, data can be viewed in differential format by setting a non-zero Reference Stage in the Stage Settings option. Some important points to remember when viewing differential data are described below:

Sigma 1 and Sigma 3

The DIFFERENTIAL in-plane principal stresses (Sigma 1 and Sigma 3) are obtained by subtracting the stress tensors and calculating the differential principal stress tensor at each node. DIFFERENTIAL Sigma 1 and Sigma 3 are NOT obtained by simply subtracting the stress magnitudes, since this would not account for changes in principal stress orientation at different stages.

Displacement

DIFFERENTIAL Horizontal and Vertical Displacement are obtained by subtraction of Visible Stage – Reference Stage values. DIFFERENTIAL Total Displacement is obtained by subtracting the total displacement vectors.

Other Data

All other DIFFERENTIAL data (eg. Strength Factor, Sigma Z, Strain, User Data), is obtained by subtracting the Visible Stage – Reference Stage values.

Display Options

All applicable Display Options will indicate DIFFERENTIAL results, as follows:

- Stress Trajectories will indicate the orientation of the DIFFERENTIAL Sigma 1 and Sigma 3 principal stresses at each node.
- Deformation Vectors will indicate the differential displacement vectors.

Show Yield

Yielded elements are displayed as follows: Elements that yielded AFTER the Reference Stage are displayed in **red**. Elements that yielded IN or before the Reference Stage are displayed in **dark red**. This applies to the display of Yielded Elements, Bolts, Liners and Joints. See the Show Yield topic for details.

Animate Stages

The **Animate Stages** option, available in the toolbar or the **Data** menu, will automatically "select" the Stage Tabs for you in consecutive order. This will give an "animated movie" display of results as the staging progresses.

The timing of the animation can be controlled by setting the *Minimum Animation Time* in the Stage Settings dialog. This controls the *minimum* delay between the display of results at each stage. However, note that the actual delay may be controlled by the speed of your computer and size of the model (number of finite elements), and can be longer than the *Minimum Animation Time*.

To EXIT the **Animate Stages** mode:

- Press the *Escape* key!!! There may be some delay between pressing the *Escape* key and the stopping of the animation. This is normal, just wait while the program stops the animation.
- You can also re-select the **Animate Stages** option to stop the animation. Although the "hourglass" icon will be displayed for the cursor, the program will still register mouse clicks.
- When you EXIT **Animate Stages** mode, the program will always reset the view to the first Stage Tab, regardless of the stage at which the animation was stopped.

Show Values

The **Show Values** option allows the user to display bolt data , liner data , joint data or spring data directly on the model. The data can be displayed as numerical values and / or graphically, directly on the model elements.

To use **Show Values**:

1. Select **Show Values** from the toolbar or **Analysis** menu.
2. In the **Show Values** dialog, select the *Data* you would like to display, using the checkboxes and listboxes.
3. Select the desired display / drawing options. Note:
   - The *Display* checkboxes can be used to display data numerically (*Min & Max* or *All Values*), and / or graphically (*Bars* or *Lines*).
   - If the data is displayed graphically (*Bars* or *Lines* options), then the *Drawing Style* and *Size of Largest Value* options can be used to customize the graphical display.
4. Select *OK*, and **Show Values** will display data on ALL of the entities of your model, according to the selections you have made. For example, if you show values for bolts, data will be displayed on ALL bolts installed (at the stage being viewed).
5. Note: if you are not zoomed into an area of the model, the display of values will probably overlap. This is normal. To make values readable, you will have to zoom in to an area of interest. After you have zoomed, you may also find it useful to pan the view.

To hide or modify the display of values, you can re-select **Show Values**, and make the desired changes in the dialog. Alternatively, you can use the right-click shortcuts described below.

Right Click Shortcut

The **Show Values** options can also be conveniently accessed with the right mouse button, as follows:

- If you right-click the mouse directly on a model element (ie. a bolt, liner or joint), you will see a **Show Values** sub-menu in the popup menu.
- In this sub-menu, you can directly select the data to display on the model (eg. *Axial Force* for bolts). Note that the data will be displayed according to the current display settings in the **Show Values** dialog. To change these settings, you can select **Settings** from the **Show Values** sub-menu. This will display the **Show Values** dialog.
- To hide the **Show Values** display, select **Off** from the **Show Values** sub-menu.

Show Yield

The **Show Yield** option in the **Analysis** menu is used to display yielding in any elements of the model, including finite elements, support elements (bolts or liners), and joint elements. To use the **Show Yield** option:

1. Select **Show Yield** from the **Analysis** menu.
2. Use the checkboxes to display the desired results, and select *Done*.

NOTE:

- Each of the individual yield display options (yielded elements, bolts, liners and joints) is also available in the Display toolbar. This is a convenient shortcut for quickly toggling the yield display on or off, without having to use the **Show Yield** dialog.
- In the **Show Yield** dialog, you may also filter the display of yielding, to display only failure in TENSION, or failure in SHEAR / SLIPPING, using the *Filter* checkboxes. These options are not available in the toolbar, and are only available in the **Show Yield** dialog.
- When yielding is displayed, the meaning of the symbols / colours used to display yielding, will be shown at the bottom of the Legend, for each entity type.

The display of yielding for each entity type is described below.

Yielded Elements

The **Yielded Elements** option will plot symbols at the failed gauss points of all yielded finite elements.

- Failure in shear is indicated by an × symbol.
- Failure in tension is indicated by a ○ symbol.
- The × symbol will overlap the ○ symbol when an element fails in both shear and tension.

Yielding is only applicable to PLASTIC materials. If all materials in your model are ELASTIC, then no yielding can occur.

NOTE:

- When viewing differential data (ie. Reference Stage > 0), the yielded element symbols are displayed in **dark red** if yielded in the Reference Stage or earlier, and **red** if yielded AFTER the Reference Stage. See the Stage Settings / Differential Data topics for more information.

Yielded Bolts

The **Yielded Bolts** option will highlight all yielded bolt elements on the model.

- For bonded bolts (*Fully Bonded*, *Plain Strand Cable* or *Swellex / Split Sets*), individual segments of the bolt, corresponding to the intersections with finite elements, can yield, so you may notice bolts which indicate yielding only in some segments, most often near the excavation boundaries. (If you toggle the display of Elements on, and zoom in if necessary, you will see that the yielded segments correspond to intersections with the finite elements).
- For *Fully Bonded* or *Plain Strand Cable* bolts, yielded bolt elements will be highlighted in red.
- For *Swellex / Split Set* bolts, yielding may be displayed in two colours, corresponding to tensile or shear failure.
- If an *End Anchored* bolt yields, the entire bolt will be highlighted in red.

Yielded Liners

The **Yielded Liners** option will highlight all yielded Liner elements on the model.

Note that an individual liner element corresponds to one edge of a finite element, and not necessarily a whole boundary line segment, which in general will consist of several elements. A liner element in PHASE2 is also referred to as a "beam" or "beam element" since liners are modeled using beam elements on the excavation boundaries.

Yielded Joints

The **Yielded Joints** option will highlight all yielded joint elements. Yielding is determined according to the slip criterion properties you have defined for the joint(s) in the PHASE2 MODELER, and may indicate either tensile or shear failure.

Add Yield Line

The **Add Yield Line** option, available in the **Analysis** menu, allows the user to select an area of the model where yielded finite elements exist, and then automatically draw a line which encompasses all of the yielded finite elements within the selected area. To use **Add Yield Line**:

1. Select **Add Yield Line** from the **Analysis** menu.
2. Note: the display of Yielded Elements will automatically be toggled ON, if it was not already displayed. (The Yielded Elements option can be toggled on or off at any time, in the Show Yield dialog, or the toolbar.)
3. Select an area of the model, where yielding has occurred, by drawing a rectangular window with the mouse (click the left mouse button to enter two corners defining a window).
4. You will then see the **Add Yield Line** dialog. The *Visibility* option allows you to display the **Yield Line** on contour plots of all data types, or only on plots of the currently contoured data type. The colour of the **Yield Line** can be customized with the *Colour* option.
5. Select the *Add* button in the dialog, and the **Yield Line** will be displayed on the model, within the area selected by the user in step 3.

Note:

- A **Yield Line** is simply a line which encompasses all of the yielded finite elements, within the user selected region. It may be useful for helping to visualize the extent of yielded zones within the material. Any number of Yield Lines can be added to a view.
- The **Add Yield Line** option is best used by selecting only one excavation, or yielded area, at a time. If the selected area (step 3 above) is too large (for example, around multiple excavations), then a **Yield Line** will be drawn around all yielded elements in the region, and may not be representative of localized regions of yielding.
- An alternative representation of yielding is available by using the **Yielded Elements** contouring option. This is available by selecting **Yielded Elements** from the drop-down list of data types which may be contoured, in the INTERPRET toolbar. See the Yielded Element Contours topic for details.

Deleting a Yield Line

To delete a **Yield Line**:

- you can use the Delete Iso-Line option
- a convenient shortcut, is to right-click on the **Yield Line**, and select **Delete** from the popup menu

Add Iso-Line

An iso-line on a contour plot represents a constant value of a contoured variable. Whenever you are viewing a contour plot, the boundaries between contour intervals on the plot, represent iso-lines of the contoured variable.

The **Add Iso-Line** option allows the user to add user-defined iso-lines to a contour plot, representing any desired value of the contoured variable. To add an iso-line:

1. Select **Add Iso-Line** from the toolbar or the **Analysis** menu.
2. As you move the mouse cursor over a contour plot, the exact value of the contoured variable, corresponding to the current cursor location, will be displayed as a popup "data tip". This allows the user to easily determine the value of the contoured variable at different locations, without having to refer to the legend.
3. When a desired value of the variable appears in the popup "data tip", or when the cursor is at a desired location, click the left mouse button, and the **Add Iso-Line** dialog will appear. In this dialog:
   - You may edit the *Value* of the iso-line.
   - You may choose *Visibility* = *Always*, if you want the iso-line to be displayed on all contour plots, or you may choose *Visibility* = the currently contoured data type, so that the iso-line ONLY appears when you are viewing contours of that data type.
   - You may select a *Colour* for the iso-line.
4. When you are finished with your selections in the dialog, select the *Add* button to add the iso-line to the contour plot.
5. You may add any number of iso-lines to contour plots, by repeating steps 1 to 4.

Note:

- Iso-lines added with the **Add Iso-Line** option, are independent of the Contour Options, and are not affected by any changes which may be made to the **Contour Options**.
- Choosing *Visibility* = *Always* for an Iso-Line, can be useful for comparing contours of different data types on the same view. For example, you might add iso-lines of Strength Factor to a view, and then change the data type to Sigma 1, to examine the relationship between Sigma 1 and given values of Strength Factor.
- To delete an iso-line, use the Delete Iso-Line option.

Delete Iso-Line

The **Delete Iso-Line** option allows the user to delete iso-lines added with the Add Iso-Line option. To delete iso-lines:

1. Select **Delete Iso-Line** from the toolbar or the **Analysis** menu.
2. You can select individual iso-lines with the left mouse button, or multiple iso-lines with a selection window.
3. When finished selecting, press *Enter*, or right-click the mouse and select **Delete Selected**, and all selected iso-lines will be deleted.

Tips

- A shortcut to delete an individual iso-line, is to right-click on the iso-line, and select **Delete** from the popup menu.
- A shortcut to delete ALL iso-lines on a view, is to select **Delete Iso-Line**, and then right-click the mouse and select **Delete All**. A warning dialog will appear, select *OK* to delete ALL iso-lines in the view.

Query Overview

A query in PHASE2, is a user-defined line or polyline, defined within the rock mass, along which data is interpolated from the contour plots, at user-defined intervals. After a query is created, the data can be:

- graphed
- displayed directly on the model
- copied to the clipboard
- saved to a file

A query can be created in various ways:

1. With the Add Material Query option, the user can enter an arbitrary point, line or polyline, to obtain data at any location.
2. The Query Boundary option allows the user to select a boundary, and create a query exactly on the boundary, or at a user-specified offset from the boundary.
3. The Query Excavation option automatically creates query polylines along all excavated regions of the model.

NOTE:

- After creating a Query, as you select different data to view (eg. Sigma 1, Strength Factor, Displacement), OR select the Stage Tabs, the queried values automatically get updated to correspond to the contoured data. *The queried values always correspond to the contoured data and stage currently being viewed.*
- The values generated by a Query are true interpolated values, and are NOT simply the nearest nodal values.
- Queries are added on a per view basis, so that if multiple views are open, each different view can have its own queries.

For more information, see the following topics:

Editing Queries

Deleting Queries

Writing Query Files

Reading Query Files

Graphing Queries

Copy Data to Clipboard

Add Material Query

The **Add Material Query** option allows the user to define a query anywhere on the model, to obtain data anywhere within the rock mass. A query can consist of:

- a single point
- a single line segment
- an arbitrary polyline with any number of line segments

To create a query with the **Add Material Query** option:

1. Select **Add Material Query** from the toolbar or the **Query** menu.
2. You will be prompted to enter the points defining the query. Use the mouse to enter points graphically, or enter x,y coordinate pairs in the prompt line.
3. When you have finished entering points, press *Enter*, or right-click and select *Done*, and you will see the **Specify Query Locations** dialog. In this dialog, the three *Generate Values* options provide different ways of defining the intervals for generating the values along your query line. Note:
   - Keep in mind that the descriptions in the **Specify Query Locations** dialog refer to an arbitrary polyline with any number of line segments, and that the term "vertex" refers to vertices of a query polyline, which will not necessarily correspond to boundary vertices.
4. When you are finished with the **Specify Query Locations** dialog, select *OK*.
   - The query will be added to the model.
   - The interpolated values will be generated for your query, according to the *Generate Values* method chosen.
   - The queried values and / or the query ID number will be displayed on the query, if you selected these checkboxes in the **Specify Query Locations** dialog.
5. The data generated by the Query, can then be graphed, copied to the clipboard etc, as described in the following topics.

Editing Queries

Deleting Queries

Writing Query Files

Reading Query Files

Graphing Queries

Copy Data to Clipboard

Note: Queries can also be created with the Query Boundary and Query Excavations options.

Query Boundary

The **Query Boundary** option allows the user to automatically create a query, exactly on a selected boundary, or at a specified offset from the boundary. To use **Query Boundary**:

1. Select **Query Boundary** from the toolbar or the **Query** menu.
2. Use the mouse to select the desired boundary (ie. excavation, material, stage, joint or external boundary). Note that you may select only ONE boundary. (Also, see note below about the definition of a "boundary" for selection purposes).
3. After selecting a boundary, you will see a dialog. This dialog will allow you to either locate the query exactly on the selected boundary, or at a specified offset (distance) from the boundary. Note that the available offset options, will depend on the type of boundary, as well as its orientation, for example:
   - an excavation boundary will allow an offset "inside" or "outside" the boundary
   - a material or stage boundary will allow "left" or "right" offsets, or "above" or "below" offsets, depending on the orientation of the boundary.
4. After making your selections in this dialog, select *OK* and the Query will be added to the model, at the specified location (ie. either exactly on the boundary, or at the specified offset distance from the boundary).

Once queries are generated using **Query Boundary**, they are identical in all respects to queries created using **Add Material Query**, and are edited, deleted, graphed, etc in exactly the same manner.

Selecting a Boundary

It is important to note that the definition of a "boundary" which may be selected with the **Query Boundary** option, corresponds to a single boundary entity created during the modeling process. If a boundary actually consists of more than one modeling entity (for example, a material boundary which was added in two separate steps while creating the model), then only the clicked on boundary entity will be selected, even if two or more boundaries are continuous. If this situation occurs, and you wish to create a Query along a certain path of boundaries, you will have to use the Add Material Query option, which allows the user complete control over the creation of a Query.

Tips

- The **Query Boundary** option is also available as a right-click shortcut. If you right-click on any boundary, a popup context menu will appear, which makes available the **Query Boundary** option. You may find this a convenient shortcut for creating a Query on a boundary.

Query Excavations

The **Query Excavations** option will automatically create Queries around ALL EXCAVATED OPENINGS, with a single mouse click. To use **Query Excavations**:

1. Select **Query Excavations** from the **Query** menu.
2. Queries will automatically be generated along the boundaries of ALL EXCAVATED OPENINGS. The **Specify Query Locations** dialog will NOT appear.

NOTE:

- The **Query Excavations** option creates Queries around all regions which have been excavated, at the current stage which is being viewed. This does not necessarily correspond only to regions within "Excavation Boundaries" (as defined in the PHASE2). As of PHASE2 version 5.0, excavated regions can now be enclosed by any type of boundary, or combination of boundaries (for example, a material boundary, or combinations of stage and / or material and / or excavation boundaries, etc).
- By default, the *Generate Values* method will be "At each vertex" when using the **Query Excavations** option. If the "At each vertex" method is not suitable, then the queries can always be edited individually, to change the *Generate Values* method. See Editing Queries for more info.
- The **Query Excavations** option creates Queries exactly on boundaries. If an offset from the boundary is required, then the Query Boundary option can be used.
- If neither **Query Excavations** nor **Query Boundary** will create a desired Query, then it will be necessary to use the Add Material Query option, which allows a completely general, user-defined query to be created.

Once queries are generated using **Query Excavations**, they are identical in all respects to queries created using **Add Material Query**, and are edited, deleted, graphed, etc in exactly the same manner.

Editing Queries

To edit an existing query, right-click ON the query line, and select the **Edit** option from the popup menu. This will re-display the **Specify Query Locations** dialog for that query. All of the parameters you originally selected when you added the query will be shown.

Editing the query simply means that:

- You can select a different *Generate Values* option, or change the number of locations for generating the values.
- Show or hide the display of values, with the *Display Queried Values* checkbox.
- Show or hide the query ID number, with the *Display ID Number* checkbox.

The **Edit** option for queries is ONLY available as a right-click option, it is not available from the main menu.

Deleting Material Queries

To delete material queries:

1. Select **Delete Material Query** from the toolbar or the **Query** menu.
2. Select the material queries you would like to delete.
3. When you are finished selecting, press *Enter* or right-click and select *Delete Selected*, and the queries you selected will be deleted.

Tips

- A shortcut for deleting all material queries from a view is to select **Delete Material Query**, right-click anywhere in the view, and select **Delete All** from the popup menu. This will first display a dialog warning that you are about to delete all queries in the current view. Select *OK* to delete all of the queries, or select *Cancel* to abort the **Delete All** operation.
- A shortcut for deleting a single query, is to right-click ON a query line, and select **Delete Query** from the popup menu. This will delete the query you selected.

Writing Query Files

Queries can be saved to a file using the **Write Material Query** option in the **Query** menu, and read back in using the Read Material Query option. Queries are saved in a .PVF file, which stands for PHASE2 Points-Value file. A .PVF file contains:

1. The x-y coordinates of the vertices defining the query line(s).
2. The x-y coordinates of the generated query locations along the query line(s), and the data at these locations.

When you select **Write Material Query**, you will first be prompted to select the material query line(s) that you wish to write to a file. Any number of queries can be saved in the same file.

1. Select the queries you want to save to a file. You can select individual queries with the left mouse button, or multiple queries with a window, or all queries (in the view) by using **Select All** in the right-click menu.
2. When you are finished selecting queries, press *Enter*, or right-click and select **Write Selected** from the popup menu.
3. You will then see a **Save As** dialog. A query is saved as a .PVF file, which stands for PHASE2 Points-Value file. The data saved in the file will be determined by the data you are viewing (ie. Sigma 1, Strength Factor, Displacement, etc).

PHASE2 Points-Value File

The .PVF file is an ASCII file that can be viewed with a text editor. A .PVF file may not format correctly in a spreadsheet, as it was not designed for this purpose. If you wish to save queried data in a form that can be easily pasted into a spreadsheet, then use the **Copy Data** option, which is available as a right-click option only.

Reading Query Files

The **Read Query File** option allows the user to read in .PVF files saved with the Write Material Query option. A .PVF file contains:

1. The x-y coordinates of the vertices defining the query line(s).

2. The x-y coordinates of the generated query locations along the query line(s), and the data at these locations.

When you select the **Read Query File** option, you will see an **Open** file dialog. When you open a .PVF file, there are two possibilities:

1. If you are NOT graphing data (see below), then only the x-y location information will be read, and new data will be generated at these locations on your model. In this case, the query line will exist on your model, as if you had added it with the Add Material Query option.

2. If you select **Read Query File** while you are using the Graph Material Queries option, then the data in the .PVF file will also be read. This allows you to plot and compare data from different models on the same graph, within PHASE2. (Alternatively, if you right-click the mouse and select **Add From File** while using **Graph Material Queries**, this is equivalent to selecting **Read Query File**).

The **Read Query File** option will also read two other file types -- a PHASE2 Comma Separated Values File, and a PHASE2 Points File (these are listed in the *Files of Type* list box in the **Open** file dialog). These file types are generated by PHASE2 version 3.0 for DOS, for information about these file formats, see the PHASE2 v.3.0 documentation. PHASE2 for Windows will not write these file formats, they can only be read.

Graph Material Queries

After a Query has been created, using Add Material Query, Query Boundary or Query Excavations, the data generated by the Query can be graphed, using the **Graph Material Queries** option. To graph material Query data:

1. Select **Graph Material Queries** from the toolbar or the **Graph** menu. You will first be prompted to select the material queries to graph (any number of queries can be graphed on the same plot).
2. You can select individual queries with the left mouse button, or multiple queries with a window, or all queries (in the view) by using **Select All** in the right-click menu.
3. When you are finished selecting, press *Enter*, or right-click and select **Graph Selected** from the popup menu. You will see the **Graph Query Data** dialog.
4. Make your selections in the **Graph Query Data** dialog. Note:
   - The default *Data to Plot* option (*Data to plot* = *Query Data*), will plot data interpolated from the currently contoured data type (eg. Sigma 1, Strength Factor etc) which you are viewing. However, you may plot Normal or Shear stress along the Query line / polyline, by selecting the *Normal Stress* or *Shear Stress* options.
   - The *Stages to Plot* option, allows you to select any or all stages, for which to plot data, by selecting the desired checkboxes
   - The *Plot Configuration* option allows you to colour code graph lines according to stage number (if data for multiple stages is being graphed on the same plot).
5. When you have made your selections in the dialog:
   - The *Create Plot* button will generate the graph in PHASE2
   - The *Plot in Excel* button will automatically start up the Microsoft Excel program, export the data to Excel, and generate the graph in Excel.
   - The *Copy Data* button will copy the data to the Windows clipboard. From the clipboard, the data can be pasted into other applications for report writing, further analysis, etc.

   See the Plot in Excel and Copy Data to clipboard topics for more information.

Query Data Graphs

- The DISTANCE axis corresponds to the distance along the query line(s) ie. the distance is relative to the path of the query line. Zero distance corresponds to the first point of each query.
- The y-axis of a graph corresponds to the contour data being viewed (ie. Sigma 1, Strength Factor, etc), or to *Normal* or *Shear* stress along the query line, if these options were selected as the *Data to plot*.
- Note the ID numbers at the end of each curve. These correspond to the ID numbers on the query lines on the model, which are automatically generated, and which are automatically displayed whenever a material query line is graphed. The ID numbers also indicate which end of a query line (on the model) corresponds to the end of each curve (on the graph). The ID numbers are necessary for identifying multiple queries on

the same graph. To turn OFF the display of query ID numbers, you can simply right-click on a query, and uncheck the **ID Number** option (also available in the Edit option in the right-click menu).

Tips

- A shortcut for graphing a single query, is to right-click ON a query line, and select **Graph Data** from the popup menu. The procedure is then the same as steps 4 and 5 above.
- Queries can be saved to a file with the **Write Query File** option, and then read back in with the **Read Query File** option. To graph data from a query file, you can select Read Query File , or right-click the mouse and select **Add From File**, while you are using the **Graph Material Queries** option. This allows you to plot and compare material query data from different models, for example, within PHASE2. (Note that queries which are read in from a file do NOT get an ID number when graphed).

Graph Bolt Data

To graph bolt data:

1. Select **Graph Bolt Data** from the toolbar or the **Graph** menu. You will first be prompted to select the bolts to graph (any number of bolts can be graphed on the same plot).
2. You can select individual bolts with the left mouse button, or multiple bolts with a window, or all bolts by using **Select All** in the right-click menu.
3. When you are finished selecting bolts, press *Enter*, or right-click and select **Graph Selected** from the popup menu.
4. You will then see the **Graph Bolt Data** dialog, which allows you to select the data type to plot, as well as plotting data from multiple stages on the same plot. You can also choose whether to colour-code the lines according to the bolt type (if you are plotting different bolt types on the same graph), or according to the stage (if you are plotting data from multiple stages).
5. When you have made the desired selections in this dialog, select *Create Plot* and the graph will be generated in PHASE2. Alternatively, data may be plotted in Excel or copied to the clipboard, by selecting the *Plot in Excel* or *Copy Data* buttons. See the Plot in Excel and Copy Data topics for details.

Bolt Data Graphs

Note the following about the Bolt Data graphs in PHASE2:

- The DISTANCE axis corresponds to the distance along each bolt.
- The ID numbers on each curve correspond to the bolt ID numbers, which are automatically displayed whenever bolt data is graphed.
- The end of each curve (ie. the right-most point) corresponds to the location of the bolt ID number on each bolt on the model. This is important, and you should always check which end of a bolt displays the ID number, to make sure you are interpreting the graph correctly.
- After a graph has been generated in PHASE2, you may customize the graph appearance (axes titles and ranges etc), by right-clicking on the graph and selecting **Chart Properties**.

Tips

- A shortcut for graphing data for a single bolt, is to right-click ON a bolt, and select **Graph Bolt Data** from the popup menu. The procedure is then the same as steps 4 and 5 above.
- To turn OFF the display of the ID numbers after graphing, you can right-click on *any bolt*, and uncheck the **Bolt Numbers** option in the popup menu.
- You may find the Show Values option useful in conjunction with **Graph Bolt Data**, to display bolt data directly on the model.

Graph Liner Data

To graph liner data:

1. Select **Graph Liner Data** from the toolbar or the **Graph** menu. You will first be prompted to pick the liner(s) to graph.
2. Liners are selected indirectly, by selecting the boundary entity that they are lining (ie. excavation, stage, material, or external boundaries). You can select the desired boundaries individually with the left mouse button, or with a selection window, or by using **Select All** in the right-click menu to select all liners.
3. When you are finished selecting, press *Enter*, or right-click and select **Graph Selected** from the popup menu.
4. You will then see the **Graph Liner Data** dialog, which allows you to select the data type to plot, as well as plotting data from multiple stages on the same plot. You can also choose whether to colour-code the lines according to the liner type (if you are plotting different liner types on the same graph), or according to the stage (if you are plotting data from multiple stages).
5. When you have made the desired selections in this dialog, select *Create Plot* and the graph will be generated in PHASE2. Alternatively, data may be plotted in Excel or copied to the clipboard, by selecting the *Plot in Excel* or *Copy Data* buttons. See the Plot in Excel and Copy Data topics for details.

Liner Data Graphs

Note the following about the Liner Data graphs in PHASE2:

- The DISTANCE axis on liner data graphs corresponds to the distance as measured along the boundary entity(s). If liners are discontinuous along a given boundary, then there will be gaps in the liner data curves corresponding to these discontinuities.
- The ID numbering for liner graphs corresponds to the node numbering (of the finite element mesh) along the lined boundaries. This allows you to associate points on the curves with individual liner elements. The liner ID numbers are automatically displayed for all liners any time that liner data is graphed. To turn OFF the display of the ID numbers after graphing, you can uncheck the Liner Numbers option in the Display Options dialog. Alternatively, a convenient shortcut is to right-click on *any lined boundary*, and uncheck the **Liner Numbers** option in the popup menu.

Tips

- A shortcut for graphing liner data, is to right-click on a lined boundary, and select **Graph Liner Data** from the popup menu. The procedure is then the same as steps 4 and 5 above.
- You may find the Show Values option useful in conjunction with **Graph Liner Data**, to display liner data directly on the model.

Graph Joint Data

To graph joint data:

1. Select **Graph Joint Data** from the toolbar or the **Graph** menu. You will first be prompted to pick the joint(s) to graph.
2. You can select joints with the left mouse button, or multiple joints with a window, or all joints by using **Select All** in the right-click menu. Note that if a joint consists of multiple line segments, it is treated as a single joint for the purposes of selecting and graphing.
3. When you are finished selecting, press *Enter*, or right-click and select **Graph Selected** from the popup menu.
4. You will then see the **Graph Joint Data** dialog, which allows you to select the data type to plot, as well as plotting data from multiple stages on the same plot. You can also choose whether to colour-code the lines according to the joint type (if you are plotting different joint types on the same graph), or according to the stage (if you are plotting data from multiple stages).
5. When you have made the desired selections in this dialog, select *Create Plot* and the graph will be generated in PHASE2. Alternatively, data may be plotted in Excel or copied to the clipboard, by selecting the *Plot in Excel* or *Copy Data* buttons. See the Plot in Excel and Copy Data topics for more information.

Joint Data Graphs

- The DISTANCE axis corresponds to the distance along each joint. If a joint consists of multiple line segments, it is treated as a single joint for the purposes of graphing.
- Note the ID numbers at the end of each curve. These correspond to the joint ID numbers, which are automatically displayed whenever joint data is graphed. The ID numbers also indicate which end of a joint (on the model) corresponds to the end of each curve (on the graph). This is important, and you should always check where the joint ID numbers are on each joint, to make sure you are interpreting the graph correctly. To turn OFF the display of the ID numbers after graphing, you can uncheck the Joint Numbers option in the Display Options dialog. Alternatively, a convenient shortcut is to right-click on *any joint*, and uncheck the **Joint Numbers** option in the popup menu.

Tips

- A shortcut for graphing data for a single joint, is to right-click ON a joint, and select **Graph Joint Data** from the popup menu. The procedure is then the same as steps 4 and 5 above.
- You may find the Show Values option useful in conjunction with **Graph Joint Data**, to display joint data directly on the model.

Plot in Excel

Any of the data which can be graphed in PHASE², can also be exported to Microsoft Excel™ and graphed in Excel. Data which may be exported to Excel includes:

- Material Query Data
- Bolt Data
- Liner Data
- Joint Data

There are two ways in which data can be exported to Excel.

- While using any of the graph options (Graph Material Query, Graph Bolt Data, Graph Liner Data, Graph Joint Data), the **Plot in Excel** button can be selected in the **Graph Data** dialog.
- Alternatively, after a graph has been created in PHASE2, the user can right-click on the graph and select **Plot in Excel** from the popup menu.

In either case, the following will occur:

1. The Microsoft Excel™ program will automatically be launched (if you have Excel installed on your computer).
2. The raw data used to generate a graph, will be exported to an Excel spreadsheet.
3. A graph will be created in Excel.

The user can then customize the graph or process the data as desired, in Excel.

Chart Properties

Axis ranges and titles of graphs can be modified with the **Chart Properties** option. Graphs are created with:

- Graph Material Queries
- Graph Bolt Data
- Graph Liner Data
- Graph Joint Data

Chart Properties is also available by right-clicking on a graph.

Copy Data to Clipboard

Any of the data which can be graphed in PHASE², can also be copied to the Windows clipboard. From the clipboard, it can be pasted into other Windows applications (eg. Excel), for report writing, further analysis and processing, etc.

Data which may be copied to the clipboard includes:

- Material Query Data
- Bolt Data
- Liner Data
- Joint Data

There are various ways of copying data to the clipboard.

While selecting for Graph options

1. Select a graph option (Graph Material Queries , Graph Bolt Data , Graph Liner Data , Graph Joint Data).
2. Select entities as you would for graphing (see the links above for more information).
3. Right-click the mouse, and select **Copy Selected** from the pop-up menu.
4. Data for the selected entities will be copied to the clipboard, and can be pasted into a spreadsheet program such as Excel, for further processing by the user.

In Graph Data dialog

The above method will only copy data for the stage which is currently being viewed. If you wish to copy data for multiple stages, then this is possible with the *Copy Data* button in the Graph Data dialogs. For example:

1. Follow steps 1 and 2 above.
2. Right-click and select **Graph Selected**.
3. In the **Graph Data** dialog, you may now select multiple stages if desired, using the checkboxes provided. Then select *Copy Data*, and data for all selected stages will be copied to the clipboard.

Shortcut for single entity

A shortcut for copying data for a single entity (ie. a single material query, a single bolt, a single joint or a single lined boundary), is to right-click ON the entity, and select the appropriate **Copy** option from the popup menu. This will copy data for a single entity to the clipboard. You can use this shortcut at any time, you do not have to be in Graph mode.

Note – **Copy Data** is NOT available in the main menu; it is only available as a right-click option or in the Graph dialogs, as described above.

Tools Overview

The options in the **Tools** menu allow the user to add a variety of drawing objects to views, to customize the appearance of views for screen captures and reports etc. The following options are available:

- Contour Labels
- Axes
- Measurement Lines
- Lines
- Arrows
- Rectangles
- Circles
- Text Boxes
- Picture from file

All objects added to a view with the **Tools** menu options, are referred to as Drawing Tools in the PHASE^2 documentation.

After adding Drawing Tools to a view:

- the Drawing Tools can be deleted as described in the Delete Drawings topic
- the Drawing Tools can be edited as described in the Move, Re-size, Format topic
- the Drawing Tools can be saved to a file with the Save Tools File option

NOTE: All Drawing Tools are added on a per view basis, so if multiple views are open, each individual view can be customized as desired.

Contour Label

The **Label Contour** option allows the user to place contour labels on the contour plots. A contour label represents the numerical value of a contour line. To add Contour Labels:

1. Select **Label Contour** from the toolbar or the **Tools** menu.
2. Position the cursor over a contour line, and click the left mouse button. A Contour Label will be added at that position. NOTE: you may also enter x-y coordinates in the prompt line.
3. Repeat step 2 for all Contour Labels that you wish to add.
4. When finished adding labels, right-click and select *Cancel*, or press *Escape*.

After a Contour Label has been added, it can be deleted or formatted. See the Move, Re-size, Format topic for details.

Tips

- Contour labels can be added anywhere on a contour plot. It is up to the user to place them ON contour boundaries. If a contour label is not placed on a contour line, then the label will represent the value of the NEAREST contour line. If the user wishes to obtain exact interpolated data at points NOT on the contour lines, then they must use the Add Material Query option.
- As the data (ie. Sigma 1, Strength Factor, etc) or stage is changed, the values of the contour labels will be updated. However, contour labels added on one contour plot will in general, NOT be ON contour lines once the data or stage is changed. Since contour labels are added on a per view basis, each view can have its own set of contour labels, corresponding to the data and/or stage being viewed.

Measure

The **Measure** option allows the user to measure the distance and angle between any two points on a view. To use **Measure**:

1. Select **Measure** from the toolbar or the **Tools** menu.
2. Enter the first point graphically with the mouse, or enter x-y coordinates in the prompt line. NOTE: the Vertex Snap and Grid Snap options are available in the right-click menu, if you want to snap exactly to certain coordinates.
3. As you move the mouse, a line will connect the first point to the current cursor location, and the length and angle of the line will be displayed on the screen.
4. Enter the second point graphically with the mouse, or enter x-y coordinates in the prompt line, and the **Measure** line will be added to the view, with the length and angle displayed.

After a **Measure** line has been added, it can be deleted, moved, re-sized or formatted. See the Move, Re-size, Format topic for details.

Note: The ANGLE displayed by the **Measure** option is measured counter-clockwise from the positive x-direction. It is always a positive value between 0 and 360 degrees. Note that the ANGLE will change by 180 degrees, depending on which point is entered first.

Axes

The **Axes** option allows the user to add horizontal / vertical axes with x-y coordinates, anywhere on a view. To add Axes:

1. Select **Axes** from the toolbar or the **Tools** menu.
2. Enter the first point graphically with the mouse, or enter x-y coordinates in the prompt line. NOTE: the Vertex Snap and Grid Snap options are available in the right-click menu, if you want to snap exactly to certain coordinates.
3. As you move the mouse, scaled horizontal / vertical axes will be drawn between the point entered in step 2, and the current cursor location.
4. Enter the second corner graphically with the mouse, or enter x-y coordinates in the prompt line, and the **Axes** will be added to the view.
5. By default, **Axes** will initially display Global Coordinates corresponding to the coordinates of your model. **Axes** can also display Local Coordinates (lower left corner = 0 , 0), with the Format Tool dialog.

After an **Axes** object has been added, it can be deleted, moved, re-sized or formatted. See the Move, Re-size, Format topic for details.

Line

The **Line** option allows the user to add a line between any two points on a view. To add a **Line**:

1. Select **Line** from the toolbar or the **Tools** menu.
2. Enter the first point graphically with the mouse, or enter x-y coordinates in the prompt line. NOTE: the Vertex Snap and Grid Snap options are available in the right-click menu, if you want to snap exactly to certain coordinates.
3. As you move the mouse, a line will connect the first point to the current cursor location.
4. Enter the second point graphically with the mouse, or enter x-y coordinates in the prompt line, and the **Line** will be added to the view.

After a **Line** has been added, it can be deleted, moved, re-sized or formatted. See the Move, Re-size, Format topic for details.

Arrow

The **Arrow** option allows the user to add an arrow between any two points on a view. To add an **Arrow**:

1. Select **Arrow** from the toolbar or the **Tools** menu.
2. Enter the first point graphically with the mouse, or enter x-y coordinates in the prompt line. NOTE: the Vertex Snap and Grid Snap options are available in the right-click menu, if you want to snap exactly to certain coordinates.
3. As you move the mouse, an arrow head will appear at the point entered in step 2, and a line will connect the point to the current cursor location.
4. Enter the second point graphically with the mouse, or enter x-y coordinates in the prompt line, and the **Arrow** will be added to the view.

After an **Arrow** has been added, it can be deleted, moved, re-sized or formatted. See the Move, Re-size, Format topic for details.

Rectangle

The **Rectangle** option allows the user to add a rectangle or a square anywhere on a view. To add a Rectangle or Square:

1. Select **Rectangle** from the toolbar or the **Tools** menu.
2. Enter the first corner graphically with the mouse, or enter x-y coordinates in the prompt line. NOTE: the Vertex Snap and Grid Snap options are available in the right-click menu, if you want to snap exactly to certain coordinates.
3. As you move the mouse, a rectangle will be drawn between the point entered in step 2, and the current cursor location.
   - NOTE: to draw a SQUARE, hold the *Shift* key while you are moving the mouse.
4. Enter the second corner graphically with the mouse, or enter x-y coordinates in the prompt line, and the **Rectangle** or Square will be added to the view.

After a **Rectangle** has been added, it can be deleted, moved, re-sized or formatted. See the Move, Re-size, Format topic for details.

Circle

The **Circle** option allows the user to add a circle or ellipse anywhere on a view. To add a **Circle** or Ellipse:

1. Select **Circle** from the toolbar or the **Tools** menu.
2. Enter a point graphically with the mouse, or enter x-y coordinates in the prompt line.
3. As you move the mouse, an ellipse will be drawn between the point entered in step 2, and the current cursor location.
   - NOTE: to draw a CIRCLE, hold the *Shift* key while you are moving the mouse.
4. Enter a second point graphically with the mouse, or enter x-y coordinates in the prompt line, and the **Circle** or Ellipse will be added to the view.

After a **Circle** has been added, it can be deleted, moved, re-sized or formatted. See the Move, Re-size, Format topic for details.

Text Box

The **Text Box** option allows the user to add text anywhere on a view. Pre-formatted Auto-Text, corresponding to the information in the Info Viewer, can be added, or any user-defined text can be added, as explained below.

To add a Text Box:

1. Select **Text Box** from the toolbar or the **Tools** menu.
2. Pick the point where you want to add the Text Box. This point will locate the upper left corner of the Text Box. Enter the point graphically with the mouse, or enter x-y coordinates in the prompt line.
3. You will see the Add Text dialog.
4. TO ADD USER DEFINED TEXT: type any text in the "Text to add" area at the left of the dialog. Select *OK*, and the Text Box will be added to the view.
5. TO ADD AUTO-TEXT: use the "+" boxes and ☐ checkboxes in the "Auto text" area at the right of the dialog, to select the information you would like to add. When the desired checkboxes are checked, select the *Insert Auto-text* button, and the text will be added to the "Text to add" editing area at the left of the dialog. NOTE: once the Auto-text is added to the "Text to add" area, you can manually edit the Auto-text, if necessary. Select *OK*, and the Text Box will be added to the view.

After a **Text Box** has been added, it can be deleted, moved, edited or formatted. See the Move, Re-size, Format topic for details.

Picture from file

In the PHASE² INTERPRET program, images can be imported into the current view with the **Picture from File** option in the **Tools** menu. This can be used to add company logos, for example, or any other bitmap image, in order to customize views for output and presentations.

To import an image into the current view:

1. Select **Picture from file** from the toolbar or the **Tools** menu.
2. You will see an Open File dialog. The image must be in .BMP file format (bitmap). Select the .BMP file that you wish to import, and select Open.
3. The image will be added to the view, and will initially be centered in the view.

NOTE:

- Any number of images can be added to a view.
- After a **Picture** has been added, it can be re-sized, moved, edited, formatted or deleted, like any other drawing tool. See the Move, Re-size, Format topic for details.
- Like other drawing tools, **Pictures** can be permanently linked to a PHASE² file with the Save Tools File option.
- Image size can also be changed with the Scale Picture option in the **Tools** menu.

Scale Picture

The **Scale Picture** option in the **Tools** menu can be used to change the size of a picture added to a view with the Picture from file option.

However, note that the Picture size can also be changed by simply clicking on the picture, and dragging the control points with the left mouse button. Picture size can also be changed in the Format Tool dialog.

To change picture size with **Scale Picture**:

1. First, you must click on the image with the left mouse button, so that the image is "selected". When it is selected, you will see the control points highlighted on the image.
2. The **Scale Picture** option will now be enabled.
3. Select **Scale Picture** from the toolbar or the **Tools** menu.
4. You will be prompted to graphically pick the top left and lower right corners on the image, using the mouse.
5. You will then be prompted to enter the new top left and lower right corners for the image.
6. After the second point has been entered, you will see the Scale Image dialog. You can then modify the points you have entered in Step 5, if necessary.
7. When the desired coordinates are entered, select the *Scale* button in the Scale Image dialog, and the image will be scaled and translated to the desired coordinates.

Save Tools File

All Drawing Tools added to a view can be saved in a PHASE² Tools File, with the **Save Tools File** option. They can then be later recalled with the Open Tools File option, so that the user does not have to re-create drawings each time they open a file.

To save Drawing Tools:

1. If multiple views are open, first make sure that the view with the tools you wish to save, is the active view. If not, then click the mouse in the desired view. (Remember that Drawing Tools are added on a per view basis).
2. Select **Save Tools File** from the **Tools** menu.
3. You will see a Save As dialog. NOTE:
   - PHASE² Tools Files have a *.PHT filename extension.
   - By default, the name of the Tools File which will appear in the File name box, will be CURRENTFILE.PHT (where CURRENTFILE = the name of the PHASE² file you are viewing).
   - If you save a Tools File with the same name as the current PHASE² file, THEN THIS TOOLS FILE WILL AUTOMATICALLY BE OPENED WHEN THE PHASE² FILE IS OPENED IN INTERPRET. This means that the Drawing Tools you have saved will automatically appear when you open the corresponding PHASE² file in INTERPRET.
   - However, you DO NOT have to save the Tools File with the same name as the PHASE² file, you can enter any name you wish, as long as the extension is *.PHT. If you save a Tools File with a different name from the PHASE² file, then you will have to use the Open Tools File option to display the tools on a view.
4. Enter a name for the Tools File as described above, and select *Save* to save the file.

How Pictures are Saved in a Tools File

NOTE: when a Picture is saved in a Tools File (ie. images added to a view with the Picture from file option), the image is not actually saved in the Tools File. Instead the filename path where the image is currently stored, is saved in the Tools File. INTERPRET will use this filename path to load the file, when the Tools File is opened. See the Open Tools File topic for more information.

Open Tools File

Drawing Tools saved with the Save Tools File option can be displayed again at any time with the **Open Tools File** option.

To open a Tools File:

1. Select **Open Tools File** from the **Tools** menu.
2. You will see an Open file dialog. PHASE² Tools Files have a *.PHT filename extension.
3. Open the desired Tools File, and the Drawing Tools saved in the file will be displayed on the current view.

NOTE:

- If a Tools File has been saved with the SAME FILE NAME as the corresponding PHASE² (*.FEA) file, then you do not necessarily have to use the **Open Tools File** option. In this case, THE TOOLS FILE WILL AUTOMATICALLY BE OPENED WHEN THE PHASE² FILE IS OPENED IN INTERPRET, and the Drawing Tools will automatically appear on the view.
- Therefore, it is only necessary to use the **Open Tools File** option, if you have saved a Tools File with a DIFFERENT name from the corresponding PHASE² file.
- Once tools are displayed on a view with the **Open Tools File** option, they behave in the same manner as if they were added through the Tools menu options, and can be edited, deleted etc in the same manner.

How Pictures are Read From a Tools File

When a Picture is saved in a Tools File (ie. images added to a view with the Picture from file option), the image is not actually saved in the Tools File. Instead the filename path where the image is currently stored, is saved in the Tools File. In order for the image to appear when the Tools File is loaded, the image file must be located in either:

- the folder specified by the filename path saved in the Tools File, or
- the folder from which the Tools File was read (if this is different from the image filename path)

If INTERPRET cannot find the image file in either of these locations, a warning message will appear, and the image will not be displayed on the view. Similarly, if the image file has been deleted or re-named, it will not appear when the Tools File is loaded.

Delete Drawings

The **Delete Drawings** option can be used to delete any Drawing Tools added to a view, with the options in the Tools menu (ie. Contour Labels, Axes, Measures, Lines, Arrows, Rectangles, Circles, Text Boxes, Pictures).

To use **Delete Drawings**:

1. Select **Delete Drawings** from the toolbar or the **Tools** menu.
2. Select the Drawing Tools that you want to delete. You can select individual Drawing Tools by clicking on them with the mouse, or multiple Drawing Tools with a selection window.
3. When you are finished selecting, right-click the mouse and select *Delete Selected*, or press *Enter*, and the selected Drawing Tools will be deleted.

Delete All Drawing Tools

There are a number of shortcuts that can be used to quickly delete ALL Drawing Tools on a given view. For example:

- After Step 1 above, right-click the mouse and select *Delete All*. A warning dialog will first appear. If you select *OK* in this dialog, all Drawing Tools in the view will be deleted.
- After Step 1 above, type the asterisk character ***** in the prompt line, followed by *Enter*. This will select all Drawing Tools in the current view. Then right-click and select *Delete Selected*, or press *Enter*, and all Drawing Tools in the view will be deleted.
- You can also use the Delete All Drawings option in the **Tools** menu.

Shortcuts to Delete a single Drawing Tool

- A quick and easy shortcut to delete any individual Drawing Tool, is to right-click the mouse ON a Drawing Tool, and select *Delete* from the popup menu.
- You can also click on a Drawing Tool at any time with the left mouse button, and press the *Delete* key on the keyboard.

See the Move, Re-size, Format topic for more info.

Delete All Drawings

The **Delete All Drawings** option can be used as a shortcut to delete ALL Drawing Tools in a view. To Delete All Drawings:

1. Select **Delete All Drawings** from the **Tools** menu.
2. A warning dialog will appear. If you select *OK* in this dialog, all Drawing Tools in the view will be deleted.

NOTE: see the Delete Drawings topic for a list of other shortcuts that can be used to delete ALL Drawing Tools in a view.

Move, Re-size, Format Drawing Tools

In addition to the Delete options in the **Tools** menu, various editing operations can be performed on Drawing Tools, by clicking on them with the mouse. Tools can be deleted, moved, re-sized and formatted, as described below.

Right-Click

If you *right-click the mouse* on a Drawing Tool, you will see a popup menu which makes available various editing options.

- *Delete* – delete the clicked-on Drawing Tool
- *Format* – displays the **Format Tool** dialog for the clicked-on Drawing Tool (see below for details)
- *Edit Text* – (Text Box Drawing Tools only) – if you select *Edit Text* from the popup menu, you will see the Add Text dialog, with the text from the Text Box in the editing area, allowing you to edit the text as required.

Single Left-Click

If you *single-click the left mouse button* on a Drawing Tool, this will "select" the tool, and you will see the "control points" highlighted on the tool. While in this mode:

- *Re-sizing* – you can click and drag the control points, to re-size the tool (NOTE: not applicable for Contour Labels or Text Boxes, which cannot be re-sized)
- *Moving* – If you hover the mouse over any part of the Drawing Tool, but NOT on a control point, you will see the four-way arrow cursor, allowing you to click and drag the entire Drawing Tool to a new location (NOTE: not applicable for Contour Labels, which cannot be moved.)
- *Deleting* – you can delete the selected tool by pressing *Delete* on the keyboard (NOTE: not applicable for Contour Labels)

Double Left-Click

If you *double-click the left mouse button* on a Drawing Tool, you will see the **Format Tool** dialog. The **Format Tool** dialog allows the user to customize styles, colours etc. Only the options applicable to the clicked-on tool, will be enabled in the **Format Tool** dialog.

- *Position* – the position and size of each Drawing Tool, is defined by two points (except for Text Boxes and Contour Labels, which are defined by a single point only, defining the position of the upper left corner). These points are referred to as the Head and the Tail (the Head is the first point you entered when you added the Drawing Tool, and the Tail is the second point). The user may edit these values in the **Format Tool** dialog, in order to change the size and / or position of a Drawing Tool. However, in most cases, this will probably be done graphically, as described above for the "Single-Click" options. The *Lock Position* option can be used to lock the position of the object on the view, so that it cannot be moved or re-sized.

- *Hatch and Fill* – Hatch and Fill are only applicable to Rectangle or Circle Drawing Tools, and allow you to fill a Rectangle or Circle with a Fill colour or a Hatch pattern. NOTE: you cannot use both Hatch and Fill together, since they are always the same colour. If you use a Hatch pattern, then the Fill Colour determines the colour of the Hatch lines. Fill can also be used to change the background colour of Text Boxes or Contour Labels, but Hatch cannot be used for Text Boxes or Contour Labels.
- *Line Style* – Line Style is applicable to all Drawing Tools (except Picture from file). It allows you to change the Colour and / or Weight (thickness) of the Line used to draw the object. For Text Boxes or Contour Labels, the Colour will change the text or label colour, but the Weight is not applicable.
- *Arc* – only applicable for Circle Drawing Tools. Arc allows you to display an arc segment of a Circle or Ellipse, by entering starting and ending points of the arc segment in degrees.
- *Axis Style* – only applicable for Axes Drawing Tools. Allows you to specify any combination of Top, Left, Bottom or Right axes to be displayed.
- *Custom Axis Spacing* – allows you to customize the axis interval spacing for Axes Drawing Tools, and choose Global or Local coordinates. Local coordinates will ensure that the lower left corner of the Axes represents (0 , 0) regardless of where the Axes are placed on the view. Global coordinates will correspond to the view coordinates.
- *Arrow Style* – only applicable for Line or Arrow Drawing Tools. Allows you to specify arrow heads at either or both ends of a Line or Arrow Drawing Tool.
- *Font* – only applicable for Text Boxes or Contour Labels. Allows you to change the Font, Font Style and Font Size of the text in a Text Box or Contour Label.
- *Image Properties* –applicable to images added with the Picture from file option. *Transparent* allows you to select one colour on the image which will be transparent (invisible). *Alpha-blend* allows you to adjust the overall transparency of the image. *Image Behind Geometry* will ensure that the image is behind the model, if the image and model overlap. *Scaling* allows the user to maintain the original image size, if the view is zoomed in or out, or scale the image with the view as the view is zoomed.
- *Apply* – use the *Apply* button to apply the current dialog settings to the selected tool, to view the results without closing the **Format Tool** dialog. (NOTE: if you select *Cancel* after you use *Apply*, all formatting changes will be cancelled and the original formatting of the tool before applying the changes, will be restored)
- *OK* – select *OK* to apply the current settings to the selected tool, and close the dialog
- *Default* – select the *Default* button if you wish to make the current dialog settings the Default settings for ALL NEW TOOLS of the currently selected type (for example, if a Line object is selected, you can choose a Colour and Weight, and select Default, and all new Lines will use the new Colour and Weight settings). Other tool types will NOT be affected. A warning dialog will first appear, before the new Default settings are saved.

New Window

Any number of windows (views) can be opened for a file by selecting the **New Window** option from the **Window** menu. This allows the user to:

- View different data in different views (eg. Sigma 1, Strength Factor, etc.)
- Zoom to different regions of the model in different views.
- Enable different Display Options in different views.
- Select different Contour Options in different views.
- Add different Drawing Tools in different views (eg. contour labels , text).
- Add different material queries in different views.

INTERPRET will always open a **New Window** by displaying a **Sigma 1** contour plot. After a new window is opened, it is independent of all other windows open for the same file.

- Windows can be tiled using the **Tile Horizontally** or **Tile Vertically** options in the **Window** menu.
- A list of ALL windows for ALL files can always be found at the end of the **Window** menu (select **More Windows...** if you have more than 10 windows open). This is useful for switching from view to view when the window size is maximized. Windows are identified by their file name and the data they are displaying (eg. Example.fea:Sigma 1).

Right-Click Menus

Whenever you are using the mouse in PHASE2, the right mouse button will always bring up a context menu with option(s) appropriate to the action being performed, or the entity clicked on. The right mouse button can be used in many different ways in PHASE2:

1. When using any option in PHASE2, always check what is available in the right-mouse menu, you may find the options very useful and time saving. For example:
   - After selecting entities, the right mouse button always makes available *Done Selection*.
   - For other options, *Done* or *Cancel* is available.
   - The Snap mode can be toggled on or off.
2. Entities can be directly clicked on with the right mouse button, to display a popup menu with applicable options. For example:
   - Support elements (eg. bolts or liners) can be clicked on to display popup menus with graphing and display options.
   - Drawing Tools (eg arrows, text boxes etc) can be clicked on to display popup menus with editing and formatting options.
   - Boundaries can be clicked on to display popup menus with querying and display options.
   - Queries can be clicked on to display popup menus with graphing and display options.
3. If you are not using any menu option, or clicking directly on a model or screen entity, the default right-click menu makes available:
   - Zoom Window
   - Zoom Mouse
   - Zoom All
   - Pan
   - Display Options
   - Contour Options
   - Export Image File

The user is encouraged to experiment with the many different options and shortcuts which are available by using the right mouse button in PHASE2.

Selecting

For many options in PHASE2 you are required to select entities on the screen, for example:

- Queries for graphing, deleting
- Support (bolts and liners) for graphing
- Drawing Tools for formatting and editing

Here are some useful tips to remember when selecting:

1. Individual entities can be selected with the left mouse button.
2. Multiple entities can be selected with a window. To draw a selection window, press and HOLD the left mouse button, and as you move the mouse, you will see a selection window opening up. When you release the left mouse button, all entities WITHIN or CROSSING the window will be selected.
3. When entities are selected, they will change colour to indicate that they are selected.
4. IMPORTANT!! Selection works as a toggle -- entities can be "de-selected" at any time during the selection process, by simply re-selecting them with the left mouse button. You will notice that they change back to their original colour when de-selected. De-selection can only be done on individual entities, with the left mouse button, and NOT with a selection window. De-selection is useful any time that you select an entity by mistake, or if you wish to exclude some entities after multiple selection with a window.
5. **Select All** (or **Delete All**) options are available in the right-click menus. Select All can also be done by typing the asterisk (*) character in the prompt line.
6. You can use the above selection procedures in any combination, as required. For example, you can select a few entities with the left mouse button, then select other entities with a window, and then de-select some of the entities selected within the window. Experiment and see how this works.
7. When selection is complete, press *Enter* to carry out the option, or right-click and select an option from the popup menu (eg. **Delete Selected** or **Graph Selected**).

Shortcut Keys

The following keyboard shortcuts are available:

Zoom All	F2 function key
Zoom In	F5 function key OR Home key
Zoom Out	F4 function key OR End ke
Zoom Excavation	F8 function key
Pan Left	left arrow
Pan right	right arrow
Pan up	up arrow
Pan down	down arrow
Grid toggle	F7 function key
Snap toggle	F9 function key
Stage tabs	Page Up / Page Down keys OR Ctrl-Page Up / Ctrl-Page Down

Index

A

Add Iso-Line, 68
Add Material Query, 71
Add Yield Line, 67
Analysis Results, 6
Animate Stages, 63
Arrows, 92
Auto Options, 41
AVI Movie Creation, 13
Axis, 90

B

Bolt Elements, 33
Bolt Numbers, 28
Bolts, 30
Boundaries, 30

C

Chart Properties, 84
Circle, 94
Contour Format, 39
Contour Labels, 88
Contour Mode, 40
Contour Options, 37
Contour Range, 38
Copy Data to Clipboard, 85
Copy Picture, 11

D

Data Overview, 47
Data Tips, 23
Defining User Data, 56
Deform Boundaries, 34
Deform Contours, 34
Deform Mesh, 33
Deformation Vectors, 35
Delete All Drawings, 101
Delete Drawings, 100
Delete Iso-Line, 69
Deleting, 57, 75
  Queries, 75
  User Data, 57
Differential Data, 62
Discretizations, 30
Displacement Contours, 51
Display Options, 25, 26

E

Editing, 57, 74
  Queries, 74
  User Data, 57
Element Numbers, 27
Elements, 32
Excel
  plotting in, 83
Exporting Images, 10

F

File
  Opening, 7
Forces, 31

G

Graph Bolt Data, 80
Graph Joint Data, 82
Graph Liner Data, 81
Graphing Queries, 78
Grayscale, 24
Grid, 46
Ground Level, 43

I

Info View, 9

J

Joint Numbers, 29

L

Label Contour, 88
Legend Options, 42
Liner Numbers, 29
Liners, 30
Lines, 91
Log File, 8

M

Measure, 89
Modeler, 8
Multiple Files, 7

N

New Window, 104
Node Numbers, 27

O

Open Tools File, 99
Opening, 7, 8
  File, 7
  Log File, 8
  Modeler, 8
  Multiple Files, 7

P

Pan, 20
Phase2 Interpreter, 5
Picture Format, 12
Picture from file, 96
Plot in Excel, 83

Q

Query, 74, 75, 78, 79
  Deleting, 75
  Editing, 74
  Graphing, 79
Query Boundary, 72
Query Excavations, 73
Query Files, 76, 77
  Reading, 77
  Writing, 76
Query Overview, 70
Quick Zoom, 21

R

Rectangle, 93
Restraints, 31
Right-Click Menus, 105
Ruler, 43

S

Save Tools File, 98
Scale Factor, 36
Scale Picture, 97
Select Composite Component, 60
Shortcut Keys, 107
Show Values, 64
Show Yield, 65
Shrink Contours, 33
Shrink Elements, 32
Sigma 1, 49
Sigma 3, 49
Sigma Z, 49
Snap, 44, 45
Stage Settings, 61
Strain Contours, 53
Strength Factor, 50
Stress Block, 43
Stress Trajectories, 32

T

Text Box, 95
Tools Overview, 87
Tractions, 31

U

Ubiquitous Joints, 52
User Data, 55, 56, 57
  Defining, 56
  Deleting, 57
  Editing, 57
User Data Configuration File, 58
User Data Variables, 59

Y

Yielded Element Contours, 54

Z

Zoom All, 14
Zoom Excavation, 19
Zoom In, 15
Zoom Mouse, 18
Zoom Out, 16
Zoom Window, 17