

Phase²

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

Phase² Model Program Reference Manual

© 1998 - 2001 Rocscience Inc.

Table of Contents

Introducing Phase2	5
Phase2 Modeler	6
File Menu	
Starting a New Project	7
Opening a File	8
Opening Multiple Files	8
Saving a File	8
Project Settings	9
Plane Strain Analysis	13
Axisymmetric Analysis	14
Info Viewer	16
Compute	17
Interpret	18
Import DXF	19
Import Properties	20
Export DXF	22
Exporting Images	23
Edit Menu	
Undo / Redo	24
Copy	25
View Menu	
Limits	26
Zoom All	27
Zoom In	28
Zoom Out	29
Zoom Window	30
Zoom Mouse	31
Zoom Excavation	32
Pan	33
Display Options	34
Element Numbers	35
Shrink Elements	35
Node Numbers	35
Restraints	36
Forces	36
Tractions	36
Grid	37
Snap	38
Ruler	39
Stress Block	39
Ground Level	39
Boundaries Menu	
Boundary Types	40
Entering Coordinates	42
Correcting Mistakes	43
Drawing Arc Segments and Circles	44
Automatic Boundary Intersection	45
Add Excavation	46

Add External	47
Add Material	50
Add Stage	52
Add Joint	54
Add Piezometric Line	57
Water Pressure Grid	58
DXF Import/Export	60
Delete Boundaries	60
Move Boundaries	61
Add Vertices	62
Move Vertices	63
Delete Vertices	64
Scale Excavation	65
Rotate Excavation	66
Copy Excavation	67
Selection Filter	68
Delete Piezometric Line	69

Mesh Menu

Mesh Overview	70
Mesh Setup	71
Mesh Type	72
Graded Mesh	73
Uniform Mesh	76
Radial Mesh	77
Element Type	79
Advanced Discretization	80
Discretize	81
Custom Discretize	82
Mesh	83
Mesh Refinement	84
Mapped Meshing	85
Mesh Quality	87
Reset Mesh	89

Loading Menu

Field Stress	90
Constant Field Stress	92
Gravity Field Stress	94
Custom Field Stress	95
Load Splitting	96
Seismic Loading	98
Tractions Overview	100
Normal / Shear Traction	101
Horizontal / Vertical Traction	102
Angle Traction	102
Traction Examples	103
Modifying Tractions	104
Deleting Tractions	104
Add Nodal Force	105
Delete Nodal Force	106
Add Springs	107
Delete Springs	108

Restraints Menu

Restraints Overview	109
Restraint Options	110
Infinite Elements	111
Applying Restraints	112
Restrain X	113
Restrain Y	113
Restrain XY	114
Free	114
Add Displacement	115
Delete Displacement	116

Support Menu

Bolts Overview	117
Add Pattern Bolts	118
Add Spot Bolt	120
Delete Bolts	121
Stretch Bolts	122
Bolt DXF Import	123
Liner Overview	124
Add Liner	125
Delete Liner	126

Properties Menu

Properties Overview	127
Define Materials	128
Material Name	129
Material Colour	129
Initial Element Loading	130
Unit Weight	131
Elastic Properties	132
Strength Parameters	133
Staged Material Properties	134
Define Groundwater Properties	136
Pore Pressure Calculation	139
Define Bolts	140
End Anchored Bolts	141
Fully Bonded Bolts	141
Plain Strand Cable	142
Swellex / Split Sets	145
Define Liners	148
Define Joints	150
Define Composite Liner	152
Assign Properties Overview	154
Assign Materials	155
Assign Bolts	157
Assign Liners	159
Assign Joints	159
Assign Composite Liners	160
Reset Assign	160

Tips

Selecting	161
Right-click Menus.....	163
Shortcut Keys.....	164

Index

Index	165
-------------	-----

Introducing Phase2

PHASE2 is a 2-dimensional plastic finite element program for calculating stresses and displacements around underground openings, and can be used to solve a wide range of mining and civil engineering problems, involving:

- PLANE STRAIN or AXISYMMETRY
- ELASTIC or PLASTIC materials
- STAGED excavations (up to 50 stages)
- MULTIPLE MATERIALS
- SUPPORT (bolts / shotcrete)
- CONSTANT or GRAVITY field stress
- JOINTED rock
- GROUNDWATER (include pore pressure in analysis)

The **PHASE2** program consists of 3 program modules: MODEL, COMPUTE and INTERPRET.

You are now using the **MODEL** module.

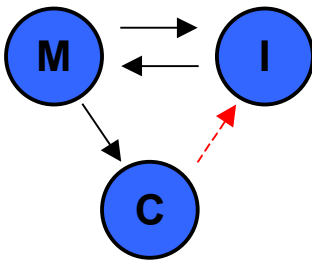
Phase2 Modeler

The **PHASE2** program consists of 3 program modules:

- MODEL
- COMPUTE
- INTERPRET

You are now using the MODEL module. MODEL is the pre-processing module used for entering and editing the model boundaries, support, in-situ stresses, boundary conditions, material properties, and creating the finite element mesh.

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.

Starting a New Project

To start creating a new project with PHASE2, select the **New** option from the toolbar or the **File** menu. A new blank project window will be opened, in which you can start creating your model.

There are two things you will generally need to do before you start entering your model boundaries:

1. Select **Project Settings** from the toolbar or the **File** menu. Various analysis and model parameters are set in the **Project Settings** dialog. In particular, if your model is staged you should set the number of stages before you proceed further, as this affects the availability and operation of certain modeling options.
2. Select **Limits** from the **View** menu, to set the initial limits of your drawing region.

See the Quick Start Tutorial for a lesson on how to create a simple model with PHASE2.

Tips

- When the PHASE2 modeler is first started, you will always be presented with a new blank project window named "Project1". You can immediately start creating a model in this window. If you are not starting a new project, then simply close this window, and **Open** the file you want to work with.

Opening a File

To open an existing PHASE2 file:

1. Select **Open** from the toolbar or the **File** menu.
2. You will see a Windows **Open** file dialog. PHASE2 files have a **.fea** filename extension (**.fea** stands for **F**inite **E**lement **A**alysis). Select the **.fea** file you want to work with, and it will be opened in PHASE2.

Opening Multiple Files

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

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

Saving a File

A PHASE2 input file (.FEA filename extension) can be saved at any point during the modeling process.

However, note that you will not be able to run COMPUTE on an .FEA file, unless the finite element **mesh** exists.

If you only wish to save the **boundaries** of a model, then you can use the **Export DXF** option for this purpose.

Project Settings

The **Project Settings** option, available in the toolbar or the **File** menu, allows the user to set various important modeling and analysis parameters. The **Project Settings** should always be chosen at the start of the modeling, since some of the settings affect the availability and operation of certain Model options.

- In particular, the *Number of Stages*, *Analysis Type*, and *Groundwater Method*, should always be chosen at the start of modeling.
- Other settings, such as *Tolerance* and *Solver Type*, can be changed at any point during the modeling, up until the time you run **Compute**, since these are strictly analysis parameters.

Project Name

The user can enter a *Project Name*, if desired. The *Project Name* appears in the **Info Viewer** listing.

Number of Stages

PHASE2 allows multi-stage finite element analysis of excavations in up to **50** separate stages.

The default *Number of Stages* is one (ie. a single stage model). Enter any number between 2 and 50 to allow multi-stage modeling and analysis. You should set the *Number of Stages* before you start to create your model, since several of the Model options operate differently depending on whether your model is single stage (*Number of Stages* = 1) or multi-stage (*Number of Stages* = 2 to 50).

Analysis Type

Two types of models can be created and analyzed in PHASE2:

- A *Plane Strain* model assumes that the excavations are of infinite length in the out-of-plane direction, and therefore the strain in the out-of-plane direction is zero. Most PHASE2 models will use a *Plane Strain* analysis.
- An *Axisymmetric* analysis allows the user to analyze a 3-dimensional model which is rotationally symmetric about an axis (for example, the end of a circular tunnel). Although the input is 2-dimensional, the analysis results apply to the 3-dimensional problem.

For more information, see the **Plane Strain** and **Axisymmetric Analysis** topics.

Maximum Number of Iterations

The *Maximum Number of Iterations* controls the maximum number of iterations allowed in each *Load Step*. The default value is 500.

Tolerance

The (local) *Tolerance* value controls the allowable unbalanced nodal force at any single node. This value is the fraction of the Euclidian norm of the load vector for a given *Load Step*. The suggested value is .01 to .001. The default is .001.

Note: for an ELASTIC analysis (ie. all materials are elastic), the *Tolerance* is not used since the solution is "exact". The *Tolerance* is relevant for PLASTIC materials, and you may wish to try different values – larger values (eg. .01) will speed up the solution time for problems which have trouble converging, and smaller values (eg. .0001) may give a more accurate solution, however, this will vary depending on the problem.

Load Steps

By default, the *Number of Load Steps* used by COMPUTE at each stage is automatically determined by PHASE2 (*Number of Load Steps* = *Auto*). The user can force COMPUTE to use a pre-determined *Number of Load Steps*, by entering a number from 1 to 30 in the *Number of Load Steps* option of **Project Settings**.

Note the following:

- When a user-defined *Number of Load Steps* is entered, this number will be used AT EACH STAGE of calculation.
- When *Number of Load Steps* = *Auto*, the number of Load Steps used at each stage will not necessarily be the same.

In general it is recommended that the user leave *Number of Load Steps* = *Auto*. Entering a user-defined *Number of Load Steps* is recommended for advanced users only.

Solver Type

This option determines how COMPUTE solves the matrix representing the system of equations defined by your model. Three methods are available:

- *Gaussian Elimination*
- *Conjugate Gradient Iteration*
- *Pre-Conditioned Conjugate Gradient Iteration*

The default method is *Gaussian Elimination*, but note that:

-
- For solving large problems, you may have to use one of the *Conjugate Gradient* solution techniques, since disk swapping (swapping between disk and memory during the analysis) is enabled with the *Conjugate Gradient* methods. Disk swapping is NOT available with *Gaussian Elimination*.
 - If all your materials are ELASTIC, the solution time will be faster with the *Conjugate Gradient* techniques.

Groundwater

As of PHASE2 version 5.0, the effect of groundwater can be included in the finite element analysis, and effective stress results can be obtained. Two different methods of defining *Groundwater* conditions are available:

- *Piezometric Lines*
- *Water Pressure Grids*

Piezometric Lines

If the *Groundwater Method* = *Piezometric Lines*, then the user may define one or more *Piezometric Lines* in order to define the groundwater pore pressure conditions for each soil type. Note:

- Piezometric Lines are created with the **Add Piezometric Line** option.
- After Piezometric Lines are created, they must then be assigned to the desired materials, using the **Define Groundwater Properties** option.

See the [Add Piezometric Line](#) and [Define Groundwater Properties](#) topics for details.

Water Pressure Grids

There are three *Groundwater Method* options for defining a *Water Pressure Grid* in PHASE2:

- *Grid (Total Head)*
- *Grid (Pressure Head)*
- *Grid (Pore Pressure)*

If the *Groundwater Method* is set to one of these three options, a *Water Pressure Grid* can be created or read from a file, using the **Water Pressure Grid** option in the **Boundaries** menu. See the [Water Pressure Grid](#) topic for details.

If one of the three *Water Pressure Grid* options is selected, then the user may also select the method of interpolation, used to obtain the water pressure at any point in the soil, from the grid data. Two interpolation methods are available:

Thin-Plate Spline

The *Thin-Plate Spline* solution uses an infinite thin elastic plate under tension, to obtain a solution which allows interpolation of data at any point, based on the data at the grid points.

Reference: Franke, Richard. Thin plate splines with tension, Computer Aided Geometric Design 2 (1985) 87 – 95, North-Holland.

Chugh's Method

This is an interpolation method based on finding the nearest water pressure grid point, in each of the four quadrants with origin centered at the point where the interpolation is required. A plane is then fit through each combination of three quadrant grid points, and an interpolation is performed for each plane. This results in four interpolations, which are then averaged to obtain the final interpolated value at the desired point.

Reference: Chugh, A.K. Pore Water Pressure in Natural Slopes, International Journal for Numerical and Analytical Methods in Geomechanics, Vol. 5, 449 – 454 (1981), John Wiley & Sons Ltd.

Pore Fluid Unit Weight

- The calculated pore pressure at any given point, is directly proportional to the *Pore Fluid Unit Weight*, if the *Groundwater Method = Piezometric Lines, Grid (Total Head) or Grid (Pressure Head)*.
- The *Pore Fluid Unit Weight* does NOT enter into the pore pressure calculations, if the *Groundwater Method = Grid (Pore Pressure)*, since in this case, pore pressures are determined directly from the grid data.

Plane Strain Analysis

In the **Project Settings** dialog, two different *Analysis Types* are available, *Plane Strain* or *Axisymmetric* analysis.

Plane Strain assumes that the excavation(s) are of infinite length normal to the plane section of the analysis. In most cases you will be performing a *Plane Strain* analysis. In a *Plane Strain* analysis PHASE2 calculates:

- the major and minor in-plane principal stresses (Sigma 1 and Sigma 3),
- the out-of-plane principal stress (Sigma Z)
- in-plane displacements and strains

By definition, the out-of-plane displacement is ZERO in a *Plane Strain* analysis.

In practice, as the out-of-plane excavation dimension becomes less than five times the largest cross-sectional dimension, the stress changes calculated assuming *Plane Strain* conditions begin to show some exaggeration because the stress flow around the "ends" of the excavation is not taken into account. This exaggeration becomes more pronounced as the out-of-plane dimension approaches the same magnitude as the in-plane dimensions.

If you have an excavation which is rotationally symmetric about an axis, then you can use the *Axisymmetric* modeling option, instead of *Plane Strain*.

Axisymmetric Analysis

In the **Project Settings** dialog, two different types of analysis can be selected, *Plane Strain* or *Axisymmetric* analysis.

The *Axisymmetric* option allows you to analyze a 3-dimensional excavation which is rotationally symmetric about an axis. The input is 2-dimensional, but because of the rotational symmetry, you are in fact analyzing a symmetric 3-dimensional problem.

A typical use of the *Axisymmetric* modeling option, is to analyze the stress state around the end of a circular tunnel.

See the PHASE2 User's Guide for a tutorial on AXISYMMETRIC modeling.

Some simple *Axisymmetric* models are shown below. Only an EXTERNAL boundary is required, the shape of the EXTERNAL boundary implicitly defines the excavation.

EXAMPLES of AXISYMMETRIC models -- If the left edge of each mesh is coincident with the $X = 0$ axis, then the model on the left represents a SPHERE and the model on the right represents a CYLINDER, in three dimensions.

The mathematical formulation of an *Axisymmetric* finite element is actually similar to *Plane Strain* (and plane stress) problems. By symmetry, the two components of displacement in any plane section of the excavation through its axis of symmetry define completely the state of strain, and therefore, the state of stress. Instead of analyzing a unit out-of-plane depth, the analysis is performed on a unit radian.

Restrictions on Axisymmetric Modeling

There are several restrictions on the use of *Axisymmetric* modeling in PHASE2:

1. The **Field Stress** must be axisymmetric ie. aligned in the axial and radial directions. Out-of-plane (or circumferential) field stress exists, but is equal to the radial stress, and cannot be independently varied.
2. Cannot be used with **GROUNDWATER** (ie. **Piezometric lines** or **Water Pressure Grid**)
3. Cannot be used with **BOLTS** (however **LINERS** are permitted).
4. Cannot be used with **JOINTS**.
5. All materials must have **ISOTROPIC elastic properties** (cannot use transversely isotropic or orthotropic elastic properties).
6. The true orientation of your excavation is arbitrary (ie. it could be horizontal, vertical or at any inclination). However, for the purposes of the PHASE2 *Axisymmetric* analysis,

you will have to map your coordinates so that the model is symmetric about the $X = 0$ axis (ie. a vertical axis located at $X = 0$), since all finite elements are rotated about this axis.

7. To form a closed excavation, one edge of your mesh must be coincident with the $X = 0$ (vertical) axis. If this is not the case, the excavation will be "open-ended".
8. All other PHASE2 modeling options can be used with an *Axisymmetric* model, however, always keep in mind the nature of an *Axisymmetric* model (for example, when defining loads, boundary conditions, etc.)

Info Viewer

The **Info Viewer** option provides a convenient summary of model 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.

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.

Compute

The PHASE2 finite element analysis engine (FEAWIN.EXE) can be launched from the PHASE2 modeler by selecting the **Compute** button on the toolbar, or from the **File** menu. The **Compute** option is not enabled unless the finite element mesh exists.

- If you have started a new project and you select **Compute** before saving the file, you will first see the Save As dialog. After you give the project a name and save the file, the analysis will be run.
- If you are working on an existing file, the file will automatically be saved and the analysis will be run when you select **Compute**.
- If for some reason you need to abort the analysis after selecting **Compute**, select the **Abort** button on the **Compute** dialog, and the analysis will be terminated.
- The PHASE2 analysis engine (FEAWIN.EXE) can be run as a standalone program. The **Compute** button in Model is simply a convenient shortcut.

Output Files

After a PHASE2 analysis of your input .FEA file, the following output files are created:

- .R??** The .R?? files are the main PHASE2 output files, containing all of the nodal stress and displacement data. The ?? characters represent the stage numbers, which can range from 01 to 50. For example, a single stage model will only have an .R01 file. A two stage model will have .R01 and .R02 files, etc.
- .X??** The .X?? files contain bolt data, if bolts are being used for support. The ?? characters represent the stage (01 to 50)
- .U??** The .U?? files contain strain data. The ?? characters represent the stage (01 to 50)
- .LOG** A .LOG file is always created, which summarizes a few important analysis parameters (number of iterations, run time, etc.) for each stage. The .LOG file can be opened in the PHASE2 Interpreter, or it can be viewed with any ASCII text editor.

All PHASE2 input and output files are ASCII files which can be viewed with an ASCII text editor.

Interpret

The PHASE2 Interpreter can be launched from the PHASE2 Modeler by selecting the **Interpret** button on the toolbar, or from the **File** menu. The **Interpret** button is enabled as soon as the finite element mesh is generated, however, you must of course run **Compute** on a file before you can look at the results in Interpret.

When Interpret is started from Model, the active file in Model will automatically be opened in Interpret. Furthermore, the user can return back to Model using the Model button in Interpret. This allows the user to switch back and forth between Model and Interpret, so that they can edit a model, re-compute and view new results.

Import DXF

Boundaries and **bolts** can be imported into PHASE2 from a DXF file (AutoCAD Drawing Exchange File), using the **Import DXF** option in the **Import** sub-menu of the **File** menu. This allows users to digitize drawings in AutoCAD, for example, and import them into PHASE2.

Importing boundaries or support in DXF format is equivalent to adding boundaries or support using the options within PHASE2 (**Add Excavation** , **Add External** , **Add Material** , **Add Stage** , **Add Joint** , **Pattern Bolt / Spot Bolt**). That is, all the same rules apply, and boundaries/bolts will be automatically checked for validity, intersected if necessary, etc.

To import boundaries / bolts from a DXF file:

1. Select **Import DXF** from the **Import** sub-menu of the **File** menu.
2. In the DXF Options dialog you will see, select the entities you would like to import, using the checkboxes provided.
3. Select the *Import* button, and you will see an Open File dialog. Select the required DXF file and select *OK*, and all entities you selected in the DXF Options dialog will be imported into the current PHASE2 document.

Tips

- Bolts cannot be imported unless the mesh exists.
- If boundaries already exist in your current PHASE2 document when you use **Import DXF**, you may see warning message(s) if the imported boundaries are not compatible with the existing boundaries. For example, if you try to import an excavation which is outside of the existing external boundary. So always be aware of what is in your DXF file when you try to import entities.
- If you are using AutoCAD to create DXF files for import into PHASE2, use the following conventions for layering and entity type:

PHASE2 Object	LAYER	AutoCAD Entity Type
Excavation Boundaries	EXCAVATION S	Closed LINE / PLINE
External Boundary	EXTERNAL	Closed LINE / PLINE
Material Boundaries	MATERIALS	LINE / PLINE
Stage Boundaries	STAGES	LINE / PLINE
Joint Boundaries	JOINTS	LINE / PLINE
Piezometric Lines	PIEZO	LINE / PLINE
Bolts	BOLTS	LINE

Import Properties

The **Import Properties** option allows the user to import **properties** (ie. material, bolt, liner, joint) from one PHASE2 file into another PHASE2 file. This allows the user to simply read properties from another file, without having to re-define properties each time they are needed.

To import properties:

1. Select **Import Properties** from the **Import** sub-menu of the **Files** menu.
2. You will see the Properties Import dialog. This dialog allows the user to read only a certain subset of properties (ie. you may want SUPPORT properties, but not MATERIAL properties to be loaded into your current model). Select the desired data you wish to read, using the checkboxes, and select the *Import* button.
3. You will then see an Open File dialog, allowing you to select the PHASE2 file from which to import properties. Open a file, and only the data selected in the Properties Import dialog will be read into your current document.

NOTE: you may also import properties from PHASE2 version 4 property files, see below for details.

4. After importing properties, you should verify that the correct properties have been read. Select a **Define** option (eg. **Define Materials**), and check that the properties in the dialog are the ones you wanted.

PHASE2 version 4 Property Files

In PHASE2 version 4, properties could be stored in special "property files" which had a .PRP filename extension. PHASE2 version 5 no longer supports the creation of .PRP files, since properties are now read directly from other PHASE2 (*.FEA) model files. However, .PRP files which were created with PHASE2 version 4 can still be read by PHASE2 version 5. In step 3 above, if you select *Files of Type* in the Open File dialog, you will be able to select .PRP files for import.

Default Properties File – DEFAULTPROPERTIES.FEA

You can create your own custom DEFAULT property file, which will always be loaded at program startup, by naming a PHASE2 file **phase2defaultproperties.fea**.

If you create such a file, then the properties in this file will always be the DEFAULT properties which are loaded at program startup. This allows you to always have your favorite DEFAULT properties automatically loaded, without the need to use the **Import Properties** option.

To re-instate the original PHASE2 default properties, you will have to delete (or rename) the **phase2defaultproperties.fea** file. If PHASE2 does not find this file at program startup, it

will load the original program defaults (ie. the default properties when you installed the program).

Note that the **phase2defaultproperties.fea** file does NOT exist when you install PHASE2, it must be created by the user.

Export DXF

PHASE2 **boundaries** and **bolts** can be exported to a DXF file (AutoCAD Drawing Exchange File), using the **Export DXF** option in the **Export** sub-menu of the **File** menu.

To export boundaries/bolts to a DXF file:

1. Select **Export DXF** from the **Export** sub-menu of the **File** menu.
2. In the DXF Options dialog you will see, select the entities you would like to export, using the checkboxes provided.
3. Select the *Export* button, and you will see a Save As dialog. Type in a filename, or select an existing filename. Select *OK* and all entities you selected in the DXF Options dialog will be saved to a DXF file with the name you entered.

Tips

- The **Export DXF** option can be used for exporting your model geometry into AutoCAD.
- It can also be useful for saving boundaries / bolts for future import back into PHASE2 using the **Import DXF** option. For example, you could save different bolt patterns in different DXF files. This could allow you to save many different bolt patterns for the same model, without having to save complete PHASE2 files. This could save disk space for very large files, and/or save the trouble of re-creating bolt patterns each time they are needed.

Exporting Images

Various options are available for exporting image files in PHASE2.

Export Image File

The **Export Image File** option in the **Export** sub-menu of 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)

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 program window is not maximized on your desktop).
-

Undo / Redo

Undo

The **Undo** option in the **Edit** menu allows you to undo changes to your model, for example:

- adding / deleting a boundary
- adding / deleting vertices
- adding / deleting loads
- changing material properties
- etc

Any number of changes can be reversed with **Undo**, starting with the most recent change and progressing backwards.

Redo

The **Redo** option in the **Edit** menu allows you to restore changes that you have *undone* with the **Undo** option. **Redo** will restore each use of the **Undo** option, starting with the most recent **Undo** and progressing backwards. **Redo** is only enabled if at least one **Undo** action has been performed.

Undo and **Redo** are also available as toolbar buttons.

Copy

The **Copy** option in the **Edit** menu 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.

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.

Limits

The first step in any computer-aided design process is setting the drawing limits of the region so that the limits encompass the model geometry. The **Limits** option allows the user to do this.

1. Select **Limits** from the **View** menu.
2. Enter the X and Y coordinates of the lower left (*Minimum X Y*) and upper right (*Maximum X Y*) corners of the drawing region you would like to start working with.
3. Press *OK* and the limits of the view will be updated to encompass the limits you have just entered.
4. The actual limits will in general not be exactly the ones you entered, unless the box you have defined is exactly the same "shape" as the current view. In any case, the limits will always encompass the coordinates you entered (ie. it will never be smaller than the limits you entered).

The **Limits** option is intended for use when initially creating the model. After a model has been created, the **Zoom** and **Pan** options can be used to adjust the viewing region with respect to the model, it is not necessary to use the **Limits** option.

Once your limits are set, you are ready to start creating your model!

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 on different views, or on different models, for example.
- 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, 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.
- A **Pan** can also be defined by entering x-y coordinates for a base point and a second point in the prompt line. Using the mouse is recommended, but if you want to Pan by an exact amount, then you may find the keyboard entry useful (for example, to pan 5 meters to the left, you could enter a base point of 0 , 0 and a second point of -5 , 0.)
- The arrow keys (up / down / left / right) can also be used to pan

Pan is available in:

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

Display Options

The **Display Options** dialog can be used to display:

Element Numbers

Shrink Elements

Node Numbers

Restraints

Forces

Tractions

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.

To use the dialog:

1. Make your selections using the checkboxes.
2. The *Apply* button will apply your selections to the active view, without closing the dialog.
3. The *Done* button will apply your selections to the active view AND close the dialog.
4. 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.
5. The **X** button in the upper right corner will close the dialog and cancel any selections. The **X** button will NOT cancel selections if they have already been applied using the *Apply* button.

Tip

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

Element Numbers

Element Numbers 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.

The display of Element Numbers can be useful if you are performing a detailed inspection or troubleshooting of the mesh.

Shrink Elements

The Shrink Elements option 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.

Node Numbers

This option 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.

The display of Node Numbers can be useful if you are performing a detailed inspection or 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.

Restraints

The Restraints option 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.

Forces

The Forces option will display all nodal forces you have applied to the model with the **Add Nodal Forces** option in the **Forces** sub-menu of the **Loading** menu. An arrow symbol and a numerical value represent the direction and magnitude.

Note: the Forces option also displays springs added with the **Add Springs** option.

If not already toggled on, Forces and Springs are automatically toggled on anytime the user adds or deletes forces or springs with the Add Nodal Forces, Delete Nodal Forces, Add Springs, or Delete Springs options.

Tractions

The Tractions option 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.

The display of Tractions is automatically toggled on, anytime the user adds or deletes tractions.

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 Grid Status. 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 in GRID SNAP mode, then the grid intersections can be "snapped" to. You may find this useful when adding boundaries, support, etc.

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). If the grid is on, the word GRID will be displayed, if the grid is off, this box will be empty.
- 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.

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).

There are three snapping modes available, which can be selected at any time:

- NO SNAPPING -- disables the snapping feature
- GRID SNAP -- snaps to the closest point on the **grid**.
- VERTEX SNAP – snaps to the closest boundary vertex. NOTE that this also includes BOLT vertices.

Tips

There are numerous ways of changing the SNAP mode:

- By selecting the **Snap** option in the **View** menu (this will bring up the **Snap** dialog)
- By left-clicking on the SNAP option on the status bar (this will toggle through the three options)
- By right-clicking on the SNAP option on the status bar (this will bring up the **Snap** dialog)
- While adding boundaries, etc, a **right-click** will bring up a context menu with the SNAP options.
- By typing **v** or **g** or **n** in the prompt line followed by **Enter**, to toggle vertex snap, grid snap, or no snap
- The F9 function key will also toggle through the three snap modes.

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, in the upper right corner of the view. 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 horizontal dotted line marking the *Ground Surface Elevation*, as entered in the **Field Stress** dialog. It is only applicable if you are using **Gravity Field Stress**.

If you are using **Constant Field Stress**, the **Ground Level** option is not applicable.

Boundary Types

The first step in creating a PHASE2 model (after setting your **Limits** and any necessary **Project Settings**) is to draw the boundaries defining the model. Boundaries are added using the **Add** options in the **Boundaries** menu or the toolbar (eg. **Add Excavation** , **Add External** , **Add Material** , **Add Stage** etc). Boundaries can also be added using the **Import DXF** option.

The different boundary types defined in PHASE2 are:

EXCAVATION

Closed polylines representing excavations. In the case of staged excavations, an excavation boundary represents the final stage (or maximum extent) of an excavation, intermediate boundaries within the excavation, being represented by stage or material boundaries (see below).

EXTERNAL

Closed polyline, defining the extents of the finite element mesh, and encompassing all other boundaries. Only one external boundary can be defined for a model.

MATERIAL

Open (or closed) polylines, used to define boundaries between different material types.

STAGE

Open (or closed) polylines, used to define boundaries between different stages of an excavation.

JOINT

Open polyline, representing a joint or other discontinuity in the rock mass. Unlike material boundaries, a joint boundary is **assigned** properties (stiffness and strength).

PIEZOMETRIC LINE

A polyline representing a water table or Piezometric surface, used to calculate pore pressures for an effective stress analysis.

NOTE:

- An EXTERNAL boundary **must always be defined for every model**.
- At least one EXCAVATION will usually be present in a model, except in certain cases such as 1) surface excavation models may not always utilize an EXCAVATION boundary 2) a "joints only" model 3) an AXISYMMETRIC model will not normally

require EXCAVATION boundaries, since the EXTERNAL boundary implicitly defines the shape of the excavation.

- MATERIAL, STAGE, JOINT and PIEZOMETRIC boundaries are optional, and are only defined if needed.
-

Entering Coordinates

To enter model geometry, you may use the keyboard, the mouse, or a combination of the two, to enter coordinates. *All boundaries are modeled by a series of straight-line segments defined by x-y coordinates. The coordinate system uses the convention: x is horizontal and increasing to the right and y is vertical and increasing upwards.*

Keyboard Entry

Use the keyboard to type coordinates in the prompt line at the bottom right of the PHASE2 application window. Simply type in x-y coordinate pairs, and press *Enter* for each pair. For example:

Enter vertex [a=arc, esc=quit]: 5 , 5 <Enter>

Enter vertex [a=arc, u=undo, esc=quit]: 5 , 10 <Enter>

Note that the use of a comma to separate the x-y coordinates is optional, and you may simply leave a space between the x and y values in the prompt line.

Graphical Entry

Use the mouse to move the cursor, and enter a point by pressing the left mouse button. Note:

- The cursor position is always displayed in the right-most box of the Status Bar.
- The Length and Angle between the current cursor location, and the last vertex entered, is displayed at the bottom of the screen, as the cursor is moved.
- The **Grid** and / or **Snap** options can help with graphical entry of boundaries.

Tips

- To complete the entry of an EXCAVATION boundary or a user-defined EXTERNAL boundary, right-click and select *Close Boundary*, or type **c** and *Enter* in the prompt line, after the last vertex has been entered.
- To complete the entry of MATERIAL, STAGE, JOINT or PIEZOMETRIC boundaries, right-click and select *Done*, or press *Enter with nothing in the prompt line*, after the last vertex has been entered.
- To abort the entry of any boundary press *Escape* or right-click and select *Cancel*.
- While entering boundaries, various options are available in the right-click menu or by typing single-letter commands in the prompt line. See the following topics for details:

Correcting Mistakes

Drawing Arc Segments and Circles

Toggling the **Snap** mode

Correcting Mistakes

If at any time you make a mistake while entering boundaries, you may "undo" the vertices of the boundary you are entering, by:

- Typing **u** followed by *Enter* in the prompt line, OR
- Selecting *Undo* from the right-click menu.

Either action will remove the last vertex from the current polyline you are entering. *Undo* can be repeated any number of times to remove successive vertices from a boundary you are drawing – experiment with this option to see how it works.

Tips

- If you have made a mistake entering an **arc**, the *Undo* command will remove the entire arc (ie. all vertices on the arc segment).
- Each time you undo a vertex, a small cross remains at the location. These are temporary, so that the user can see the location of the removed vertex. They will disappear as soon as the boundary has been completed.
- The **u** command is only operative while you are entering a boundary. After a boundary has been added, you can use the **Undo** or Edit options in the **Edit** menu, to remove or edit boundaries.

Drawing Arc Segments and Circles

While you are entering a boundary in PHASE2, you can easily create an arc segment or a circle. The following instructions assume that you have already selected one of the Add Boundary options (eg. **Add Excavation**, **Add External**, **Add Material** etc), and are in the mode for entering vertices of a boundary.

Arc Segment

To create an arc segment:

1. Enter the first point of the arc, graphically with the mouse, or through the keyboard.
2. Select *Arc* from the right-click menu, or type **a** followed by *Enter* in the prompt line.
3. In the prompt line, enter a number of segments for the arc, or press *Enter* to accept the default value of 20.
4. You will be prompted to enter the second arc point. Enter any intermediate point on the arc (for example, the midpoint between the beginning and end points of the arc).
5. As you move the mouse, you will see an arc drawn on the screen, connecting the first and second points with the current cursor location.
6. You will be prompted to enter the third arc point. After entering the third point, the arc will be drawn connecting the three points you have entered.
7. Continue entering vertices as usual, to complete the boundary.

Note:

- The arc is actually made up of straight-line segments. The number of segments will equal the number you have entered in Step 3 above.
 - If you press **c** and *Enter* before selecting the third point, the arc will close on the start point of the current polygon.
-

Circle

To create a circle:

1. Enter any point on the circle, graphically with the mouse, or through the keyboard.
2. Select *Arc* from the right-click menu, or type **a** followed by *Enter* in the prompt line.
3. In the prompt line, enter the number of segments for the circular boundary.
4. Enter a point on the circle diametrically opposite the first point entered in Step 1.
5. Type **c** and *Enter* (or right-click and select *Close Boundary*), and you will have drawn a circle with a diameter between the two points.

The *Arc* option is intended mainly for use with EXCAVATION boundaries, *but may be used with any boundary type* .

Automatic Boundary Intersection

When adding or editing boundaries, PHASE2 has built-in logic which automatically intersects boundaries with each other. The routine is very general – whenever boundaries are drawn (or edited) such that they intersect existing boundaries:

- new vertices will be inserted at all intersection points
- all invalid boundary segments will be automatically removed (for example, segments outside of the external boundary)

EXAMPLE1

1. Create a simple EXCAVATION and EXTERNAL boundary model.
2. Add a single segment MATERIAL boundary with both endpoints outside of the EXTERNAL boundary, and which crosses the EXCAVATION.
3. After the boundary is added, you should notice the following:
 - Vertices are automatically created at all intersection points of the MATERIAL boundary with the EXCAVATION and EXTERNAL boundaries.
 - the portions of the MATERIAL boundary outside of the EXTERNAL boundary are automatically removed.

EXAMPLE2

1. Using the same model created above, **move** the EXCAVATION a short distance from its original location, such that it still intersects the MATERIAL boundary.
2. Notice that new vertices are again created where the MATERIAL and EXCAVATION boundaries intersect.

Tips

- This automatic intersection capability is intended as a convenience, and may prove useful in many situations. The user is encouraged to EXPERIMENT with this capability of PHASE2. However, there may also be times when it is more appropriate, or simpler, to manually add new vertices to boundaries, rather than having them automatically generated.
- If you are in Vertex Snap mode while entering boundaries, if you hover the mouse over a boundary, and click the left mouse button, the new vertex will automatically be inserted at the nearest point on the boundary. See the **Snap** topic for details.

Add Excavation

To add an EXCAVATION boundary to your model:

1. Select **Add Excavation** from the toolbar or the **Boundaries** menu.
2. Enter the vertices of the excavation as described in [Entering Coordinates](#).
3. *An excavation boundary must always be closed.* To close the excavation boundary, after all vertices are entered, right-click and select *Close Boundary*, or you can type **c** and *Enter* in the prompt line. This automatically and correctly closes the excavation, and saves the user the trouble of having to re-enter the first vertex to complete the boundary.

Tips

- While you are entering a boundary, you can undo mistakes, draw arc segments or circles, and activate the vertex snap or grid snap options. See the following topics for information:

[Correcting Mistakes](#)

[Drawing Arc Segments and Circles](#)

Toggling the [Snap](#) mode

- If the [vertex snap](#) option is activated, you can also close an excavation boundary by snapping to the first vertex after all other vertices are entered.

Excavation Boundaries and Staging

In a multi-stage model, an EXCAVATION boundary represents the maximum extent of an excavated region. Intermediate boundaries within excavations, representing different stages of excavation, must be represented by [STAGE](#) boundaries or [MATERIAL](#) boundaries.

Surface Excavations

If you are using EXCAVATION boundaries to model a surface excavation, you will have to make sure that the EXCAVATION and EXTERNAL boundaries are coincident with each other, where appropriate.

For a tutorial on modeling surface excavations, see the PHASE2 User's Guide.

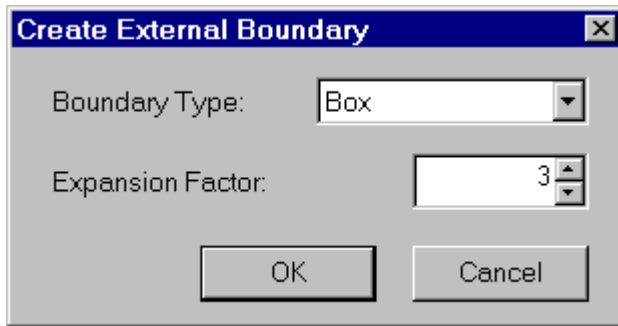
Add External

The EXTERNAL boundary, as defined in PHASE2, encompasses all other model boundaries, and defines the extents of the finite element mesh. Note:

- An EXTERNAL boundary must be defined for every model; otherwise the finite element mesh cannot be generated.
- Only **one** EXTERNAL boundary can be defined for a given model.

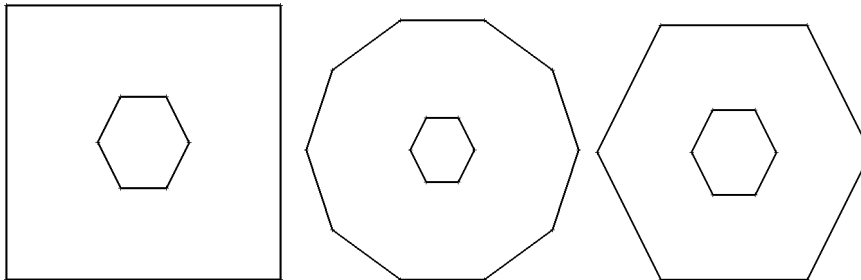
To add the EXTERNAL boundary to your model:

1. Select **Add External** from the toolbar or the **Boundaries** menu.
2. You will see the Create External Boundary dialog, shown below. This dialog allows you to choose from 3 different automatically generated EXTERNAL boundary types, or a user-defined, manually entered external boundary.



Automatically Generated External Boundary

The three automatically generated external boundary types are *Box*, *Circle* and *Hull*. If you select one of these options as the *Boundary Type*, the external boundary will be generated as soon as you select *OK* in the Create External Boundary dialog. The *Box*, *Circle* and *Hull* options are illustrated below, for an *Expansion Factor* of 1.



BOX

CIRCLE

HULL

Box

The *Box* external boundary will in general be a rectangle. It will be a square if the horizontal and vertical extents of your excavation(s) are equal.

Circle

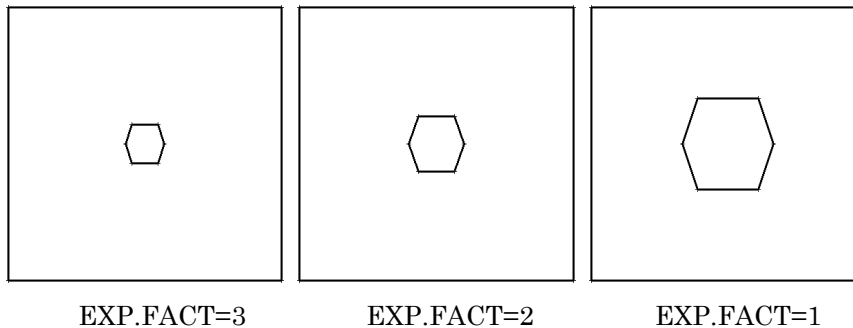
The *Circle* external boundary option actually generates a regular 10-sided polygon approximating a "circle" around the excavation(s).

Hull

The *Hull* option produces an external boundary which is a magnified version of the excavation(s). It will work for multiple excavations, although it is more appropriate for a single, non-convex excavation.

Expansion Factor

The *Expansion Factor* in the Create External Boundary dialog determines how far the automatically generated external boundary will be projected relative to your excavation dimensions. The default *Expansion Factor* is 3, but can be changed to any number between 0.2 and 1000. If more than one excavation is present, the *Expansion Factor* takes into account the combined extent of all excavations.



Effect of Expansion Factor (Boundary Type = BOX)

User-Defined External Boundary

To enter an arbitrary, user-defined EXTERNAL boundary:

1. Select *User Defined* as the *Boundary Type* in the Create External Boundary dialog.
2. Select *OK* in the dialog, and you can begin entering a user-defined external boundary. A user-defined external boundary is entered in exactly the same way as an EXCAVATION boundary – see the [Add Excavation](#) topic for details.

The EXTERNAL boundary must always be closed – as with excavations, you can close the boundary after entering all vertices, by selecting *Close Boundary* from the right-click menu, or typing **c** and *Enter* in the prompt line. You must enter a user-defined EXTERNAL boundary if:

- Your model does not include EXCAVATION boundaries (eg. a single stage surface excavation, a "joints only" analysis, or an **AXISYMMETRIC** problem), OR
- You want to enter the external boundary before the excavations.

This is because the automatic generation options for the EXTERNAL boundary (*Box*, *Circle* and *Hull*, see above) are only applicable if EXCAVATION boundaries exist.

NOTE:

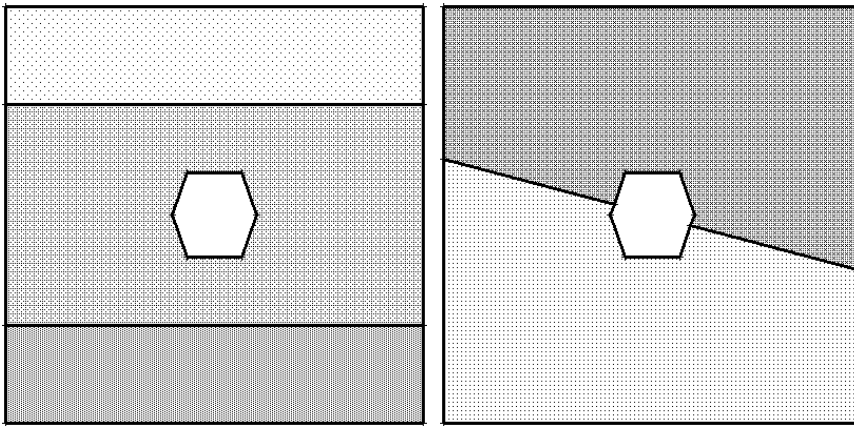
- If EXCAVATION boundaries have not been defined, and you select the **Add External** option, you will NOT see the Create External Boundary dialog. Instead you will be able to immediately begin entering the vertices of the user-defined EXTERNAL boundary.
-

Add Material

A MATERIAL boundary is used in PHASE2 to define regions with different material properties. If your entire model consists of a single rock type with uniform material properties throughout, then MATERIAL boundaries are not necessary. If you wish to define regions having different material properties (eg. strength and elastic properties), then use MATERIAL boundaries to define the regions.

To add a MATERIAL boundary to your model:

1. Select **Add Material** from the toolbar or the **Boundaries** menu.
2. Enter the vertices of the material boundary as described in [Entering Coordinates](#).
3. When all vertices are entered, right-click and select *Done*, or press *Enter* with nothing in the prompt line, and the MATERIAL boundary will be added to the model.



Examples of MATERIAL boundaries: On the left, two MATERIAL boundaries define three regions. On the right, a single material boundary defines two regions.

Typically, a MATERIAL boundary will be a single line segment dividing the model into sub-regions by virtue of intersection with other boundaries, as illustrated above. However, the definition of a MATERIAL boundary in PHASE2 is completely general – it can have any number of segments, can intersect any other boundaries (including other MATERIAL boundaries), and can even be a closed boundary.

NOTE:

- The EXTERNAL boundary must exist before MATERIAL boundaries can be added to a model.
- MATERIAL boundaries should, in general, define closed regions by virtue of intersection with other boundaries (EXCAVATION, EXTERNAL, JOINT, STAGE or other MATERIAL boundaries). If a MATERIAL boundary is terminated without

intersecting another boundary, then it will not serve the purpose of defining a separate material region (you will discover this when you try to [Assign](#) properties). In the interest of generality PHASE2 allows "open-ended" MATERIAL boundaries to be defined. One possible use of such boundaries is for conforming the finite element mesh along any desired line.

Material Boundaries vs. Stage Boundaries

As of PHASE2 version 5.0, MATERIAL boundaries and STAGE boundaries can be used interchangeably. That is, anywhere a MATERIAL boundary can be used, a STAGE boundary can also be used, and vice versa. The user may decide which boundary type is more appropriate in a given situation (ie. use MATERIAL boundaries to represent boundaries between different materials, and use STAGE boundaries to represent the boundaries between different stages of excavation).

- Note: in previous versions of PHASE2, MATERIAL and STAGE boundaries were not exactly equivalent, since STAGE boundaries could only be defined within EXCAVATION boundaries. This restriction has now been removed. See the [Add Stage](#) topic for more information.

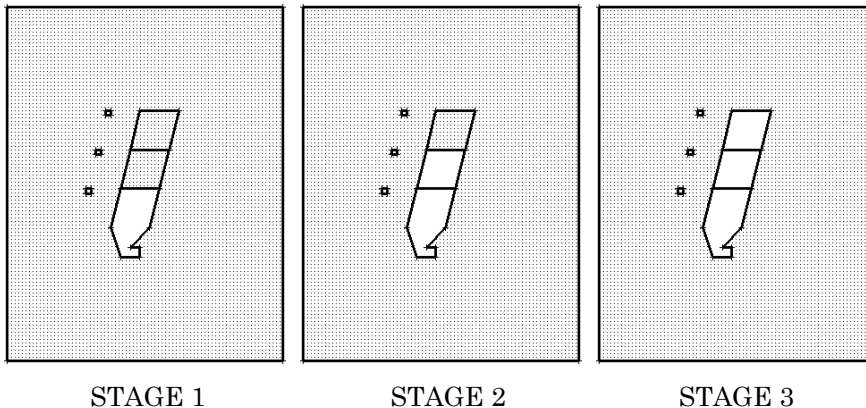
Add Stage

PHASE2 allows staged analysis of excavations in up to 50 separate stages. The *Number of Stages* is set in the [Project Settings](#) dialog.

- If you are working with staged models, then you can use STAGE boundaries to define intermediate excavation stages within EXCAVATION boundaries.
- **IMPORTANT NOTE:** as of PHASE2 version 5.0, STAGE boundaries may now be defined outside of EXCAVATION boundaries, and are no longer restricted to be within EXCAVATION boundaries. See note below.

To add a STAGE boundary to a model:

1. Select **Add Stage** from the toolbar or the **Boundaries** menu.
2. Enter the vertices of the stage boundary as described in [Entering Coordinates](#).
3. When all vertices are entered, right-click and select *Done*, or press *Enter* with nothing in the prompt line, and the STAGE boundary will be added to the model.



Example of a three stage model, using two STAGE boundaries within the main EXCAVATION boundary.

NOTE:

- As of PHASE2 version 5.0, STAGE boundaries may now be defined outside of EXCAVATION boundaries, and are no longer restricted to be within EXCAVATION boundaries. This is because (as of PHASE2 version 5.0), the user may now excavate any closed region of the model, regardless of what boundary types are used to define the region. Previously, only regions within EXCAVATION boundaries could be excavated. This allows the user maximum flexibility in defining model boundaries, using STAGE and / or MATERIAL and / or EXCAVATION boundaries.
-

Stage Boundaries vs. Material Boundaries

Because STAGE boundaries may now be defined anywhere within the EXTERNAL boundary, MATERIAL boundaries and STAGE boundaries can be used interchangeably in PHASE2. That is, anywhere a MATERIAL boundary can be used, a STAGE boundary can also be used, and vice versa. The user may decide which boundary type is more appropriate in a given situation (ie. use MATERIAL boundaries to represent boundaries between different materials, and use STAGE boundaries to represent the boundaries between different stages of excavation).

Add Joint

The **Add Joint** option allows the user to define either:

- natural joints existing in the rockmass prior to excavation, or
- artificial (man-made) joints, which only exist after a region has been excavated, and then backfilled with another material.

To add a JOINT to your model:

1. Select **Add Joint** from the toolbar or the **Boundaries** menu.
2. You will see the Create Joint dialog. This allows you to specify the joint as either *Natural* or *Artificial*. Make your selection in this dialog, and select *OK*. (See below for details about *Natural* and *Artificial* Joints, and also the *Pin Ends of Joint Together* option in the Create Joint dialog).
3. Enter the vertices of the joint boundary as described in [Entering Coordinates](#).
4. When all vertices are entered, right-click and select *Done*, or press *Enter* with nothing in the prompt line.

In terms of its behavior as a boundary in a PHASE2 model, a JOINT is identical in most respects to a MATERIAL boundary – see the [Add Material](#) section for details. The main difference between a JOINT and a MATERIAL boundary is that a JOINT is **assigned** strength and stiffness properties, whereas a MATERIAL boundary cannot be assigned properties.

Although typically a JOINT will consist of a single line segment, PHASE2 allows a very general definition of a JOINT boundary. It can consist of any number of segments, and can intersect any other type of boundary, including other JOINTS.

There is one restriction on the use of JOINT boundaries in PHASE2 – *JOINTS cannot be used in conjunction with **AXISYMMETRIC** modeling*, as the joint elements used in PHASE2 are only compatible with a PLANE STRAIN model. If you are using the *Axisymmetric* modeling option, the **Add Joint** option will be disabled. A discussion of joint elements can be found in the book "Numerical Methods in Rock Mechanics" by G.N. Pande, G. Beer & J.R. Williams, John Wiley and Sons Ltd., 1990.

Natural Joints

A *Natural* joint, as defined in PHASE2, is a joint which exists in the rockmass prior to excavation. It will remain present in the model, at all stages, unless the material on one (or both) sides of the joint, has been excavated. Note that if the material has been excavated on one (or both) sides of a *Natural* joint, that portion of the joint will no longer exist in the model, even if the excavated region is later backfilled at a subsequent stage. After excavating, the joint boundary will still be displayed on the model, however, it will be

displayed in a lighter colour, indicating that it is no longer in effect (for all stages after excavation).

Artificial Joints

An *Artificial* joint, as defined in PHASE2, is a man-made joint, which only exists AFTER the region on one (or both) sides of the joint, has been excavated in one stage, and then backfilled with another material, in a subsequent stage. When you initially add an *Artificial* joint (and then mesh the model), you will notice that the joint is displayed in a lighter colour. This indicates that the joint does not yet actually exist in the model. The joint will be displayed in its normal colour, indicating that it is in effect, only after you have done the following (using the [Assign Properties](#) dialog):

- excavated the material on one (or both) sides of the joint, in one stage
- backfilled against the joint, with another material, in a subsequent stage

This is the method of defining an artificial joint in PHASE2, and is intended to model any type of artificial joint which is designed or expected to slide, during the staged construction of an excavation.

Pin Ends of Joint Together

In the Create Joint dialog, you will see the *Pin Ends of Joint Together* option. The significance of this option is as follows:

After the finite element mesh has been generated, each segment of a joint boundary corresponds to the edges of two finite elements, and represents a "joint element". Unlike other boundaries in PHASE2, each node along a joint boundary, *is in fact two different and independent nodes*. You may observe this as follows:

- Add a joint boundary to a model, and generate the mesh. In the [Display Options](#) dialog, turn on the *Node Numbers* option and the *Shrink Elements* option. Now zoom in closely, to any node along the joint boundary. You will see that the node numbering, on either side of the joint, is different.

This dual node numbering of joint boundaries, is required in order to allow opposite sides of the joint to move with respect to each other, if yielding (slip) occurs in the joint.

At the ends of a joint boundary, this dual node scheme can lead to unexpected results, particularly if a joint boundary is terminated within the finite element mesh. This is because, at the end of a joint boundary, the joint boundary will still have two nodes, but in order for displacement continuity to be maintained within the finite element mesh, a single node is required at this location. This situation can lead to unexpected "gaps", or "overlapping" of material, at the ends of a joint, which is not kinematically feasible.

In order to address this situation, the *Pin Ends of Joint Together* option, allows the user to ensure that if a joint boundary is terminated within the finite element mesh, **ONLY ONE NODE** will exist at this location. However, it is also important to realize that, if the ends of a joint are pinned together, stress concentrations may result at these locations, since relative movement can no longer occur at the joint ends, but may occur elsewhere along the joint.

The user may decide when it is appropriate to use the *Pin Ends of Joint Together* option. A detailed examination of the displacements and stresses at the ends of a joint, may be necessary in order to determine how the end of the joint is behaving.

NOTE:

- When the *Pin Ends of Joint Together* option is in effect, this will be indicated on the model, by small circles drawn around the first and last vertices of the joint boundary.
-

Add Piezometric Line

Piezometric Lines are used for calculating pore pressure when the *Groundwater Method* in [Project Settings](#) is set to *Piezometric Lines*. A Piezometric Line in PHASE2 can represent a water table, or an actual Piezometric surface obtained from piezometer measurements, for example.

To add a Piezometric Line to your model:

1. Select **Add Piezometric Line** from the toolbar or the **Boundaries** menu.
2. Enter the vertices of the Piezometric Line as described in [Entering Coordinates](#).
3. When all vertices are entered, right-click and select *Done*, or press *Enter* with nothing in the prompt line.
4. The Piezometric Line will appear on the model as a blue line, with an automatically assigned ID number (see below for details).

Piezometric Line ID Number

Piezometric Lines are automatically given an ID number, which will appear at the mid-point of the first and last segments of the Piezometric Line. This ID number is used to identify Piezometric Lines with the corresponding material region(s) of the model, in the [Define Groundwater Properties](#) dialog. The Piezometric Line ID numbers range from 1 to 20, and are assigned in order, according to how many Piezometric Lines are already defined. A maximum of 20 Piezometric Lines can be defined for a model.

Piezometric Lines and Staging

For staged models, different Piezometric Lines can be applied at different stages of the model, in order to model changing groundwater conditions at different stages. This is done through the [Define Groundwater Properties](#) dialog, see that topic for details.

Pore Pressure Calculation

Pore pressure calculation for a Piezo Line is described in the [Pore Pressure Calculation for a Piezometric Line](#) topic.

Water Pressure Grid

Groundwater conditions are most accurately modeled by a grid of water pressure data, from which the pore pressure at any point in the ground can be determined. The grid could be based on field measurements, or could be the output from a groundwater analysis program, for example.

The **Water Pressure Grid** option is only enabled if the *Groundwater Method* in **Project Settings** is set to one of the three Water Pressure Grid options available in PHASE2:

- *Grid (Total Head)*
- *Grid (Pressure Head)*
- *Grid (Pore Pressure)*

To define a Water Pressure Grid:

1. Set the desired Water Pressure Grid type in the **Project Settings** dialog.
2. Select **Water Pressure Grid** from the **Boundaries** menu.
3. Grids can be imported (see below for details), or entered manually as described in steps 4 to 5.
4. Each point is defined by x and y coordinates, and a value, corresponding to the Water Pressure Grid type (*Total Head*, *Pressure Head*, or *Pore Pressure*) selected in **Project Settings**.
5. Enter the data in the grid. When finished select *OK*, and the location of each grid point will be marked by a blue triangle on the model.

NOTE: The Water Pressure Grid values can be displayed at each grid point with the **Display Options** dialog. By default, they are NOT displayed.

Importing Water Pressure Grid Files

The *Import* button in the Water Pressure Grid dialog allows the user to import an existing Water Pressure Grid file, into the current PHASE2 model. Two file formats can be imported:

- .PWP files – PHASE2 uses a .PWP (**p**ore **w**ater **p**ressure) file name extension to identify Water Pressure Grid data files. A .PWP file is a simple text file, where each row of the file has the format (x, y, value). The type of the value must correspond to the *Groundwater Method* in **Project Settings** (*Total Head*, *Pressure Head* or *Pore Pressure*)
- .DXF files – Water Pressure Grids can also be imported into PHASE2 from a .DXF file (this is useful if for example, a flow net has been digitized from a drawing using AutoCAD).

Exporting Water Pressure Grid Files

The *Export* button in the Water Pressure Grid dialog allows the user to export the current data in the Water Pressure Grid dialog, to a .PWP file (see above for the definition of a .PWP file). This is useful if you wish to later import the same Water Pressure Grid into another PHASE2 model.

DXF Import/Export

Boundaries can be imported into a PHASE2 model from a DXF file using the **Import DXF** option in the **File** menu. See the [Import DXF](#) topic for details.

Boundaries can also be exported to a DXF file using the [Export DXF](#) option.

Delete Boundaries

To delete boundaries:

1. Select **Delete Boundaries** from the toolbar or the **Edit** sub-menu of the **Boundaries** menu.
2. Use the mouse to [select](#) the [boundaries](#) you would like to delete.
3. When you are finished selecting, press *Enter* or right-click and select *Done Selection*, and the boundaries you selected will be deleted.

[Move Boundaries](#)

[Add Vertices](#)

[Delete Vertices](#)

[Move Vertices](#)

[Scale Excavation](#)

[Rotate Excavation](#)

[Copy Excavation](#)

[Selection Filter](#)

Move Boundaries

To move boundaries:

1. Select **Move Boundaries** from the toolbar or the **Edit** sub-menu of the **Boundaries** menu.
2. Use the mouse to **select** the **boundaries** you would like to move.
3. When you are finished selecting, press *Enter* or right-click and select *Done Selection*.
4. You will then be prompted to enter two points defining the move. You can enter the points graphically with the mouse, or with the keyboard in the prompt line.
5. After the second point is entered, all of the boundaries selected in Step 2 will be translated from their original positions according to the vector defined by the two points.

Tips

- The two points defining the move can be anywhere in the x-y plane, only their relative positions are important.
- If you enter 0 , 0 as the first point defining the move, then you can enter the actual x and y displacement as the second point. For example, if you want to move a boundary 5 meters to the left, enter 0 , 0 as the base point, and -5 , 0 as the second point.

[Delete Boundaries](#)

[Add Vertices](#)

[Delete Vertices](#)

[Move Vertices](#)

[Scale Excavation](#)

[Rotate Excavation](#)

[Copy Excavation](#)

[Selection Filter](#)

Add Vertices

New vertices can be added to existing boundaries using the **Add Vertices** option:

1. Select **Add Vertices** from the toolbar or the **Edit** sub-menu of the **Boundaries** menu.
2. As you move the mouse, a red cross snaps to the nearest point on the nearest boundary, following the cursor. You can use this as a guide to graphically add a new vertex, by clicking the left mouse button when the green cross is at the desired location. NOTE: the coordinates displayed in the Status Bar at the lower right of the screen, are the coordinates of the green cross on a boundary, and NOT the cursor coordinates.
3. You can also enter the coordinates of a new vertex in the prompt line, however, note the following – the coordinates you enter in the prompt line must be either exactly ON a boundary, or very close to a boundary, in order for a vertex to be added. If the point entered is too far from a boundary, you will see a warning dialog, and be prompted to enter new coordinates. If the point is only slightly off, a vertex will be added at the nearest point on the boundary.
4. You can add any number of new vertices by repeating Steps 2 or 3. When you are finished, press *Escape* or right-click and select *Done*.

Delete Boundaries

Move Boundaries

Delete Vertices

Move Vertices

Scale Excavation

Rotate Excavation

Copy Excavation

Selection Filter

Move Vertices

To move vertices to a new location:

1. Select **Move Vertices** from the toolbar or the **Edit** sub-menu of the **Boundaries** menu.
2. You can select individual vertices by clicking on them with the left mouse button or multiple vertices by drawing a window around them (click and hold the left mouse button, and drag to create a window).
3. When you are finished selecting, press *Enter* or right-click and select *Done Selection*.
4. You will then be prompted to enter two points defining the move. You can enter the points graphically with the mouse, or with the keyboard in the prompt line.
5. After the second point is entered, all of the vertices selected in Step 2 will be translated from their original positions according to the vector defined by the two points.

Tips

- The two points defining the move can be anywhere in the x-y plane, only their relative positions are important.
- If you enter 0 , 0 as the first point defining the move, then you can enter the actual x and y displacement as the second point. For example, if you want to move a vertex 5 meters to the left, enter 0 , 0 as the base point, and -5 , 0 as the second point.
- You may also find it useful to use the **snap** options (vertex or grid snap), to snap exactly to existing vertices, or grid points, when selecting the base point and / or the second point, defining the move.

[Delete Boundaries](#)

[Move Boundaries](#)

[Add Vertices](#)

[Delete Vertices](#)

[Scale Excavation](#)

[Rotate Excavation](#)

[Copy Excavation](#)

[Selection Filter](#)

Delete Vertices

To delete boundary vertices:

1. Select **Delete Vertices** from the toolbar or the **Edit** sub-menu of the **Boundaries** menu.
2. You can select individual vertices by clicking on them with the left mouse button, or you can select multiple vertices by drawing a window around them (click and hold the left mouse button, and drag to create a window).
3. When you are finished selecting, press *Enter* or right-click and select *Done Selection*.
4. All selected vertices will be deleted and all boundaries will be redrawn using the remaining vertices.

[Delete Boundaries](#)

[Move Boundaries](#)

[Add Vertices](#)

[Move Vertices](#)

[Scale Excavation](#)

[Rotate Excavation](#)

[Copy Excavation](#)

[Selection Filter](#)

Scale Excavation

The **Scale** option can be used to scale the size of an EXCAVATION. To use **Scale**:

1. Select **Scale** from the **Edit** sub-menu of the **Boundaries** menu.
2. Use the mouse to **select** the excavation(s) you would like to scale.
3. When you are finished selecting, press *Enter* or right-click and select *Done Selection*.
4. You will then be prompted to enter a base point for the scaling. Use the mouse or enter the x-y coordinates of the base point in the prompt line.
5. You will then be prompted to enter the scaling in the x and y directions. Enter scale factors for both the x and y directions and press *Enter*, and all selected excavation(s) will be scaled about the base point. (Example: [1.5 , 1] would increase the x-axis dimensions by 50% and leave the y-axis dimensions the same).

Tips

- To scale an excavation about its center, enter the centroid of the excavation as the base point.
- To scale an excavation about any vertex, enter the vertex as the base point (this will leave the position of the vertex at the base point unchanged, and scale the location of all other vertices according to the (x,y) scale factors and the distance from the base point).
- An ellipse can be created from a **circle** by using the **Scale** option.

[Delete Boundaries](#)

[Move Boundaries](#)

[Add Vertices](#)

[Delete Vertices](#)

[Move Vertices](#)

[Rotate Excavation](#)

[Copy Excavation](#)

[Selection Filter](#)

Rotate Excavation

The **Rotate** option can be used to rotate an EXCAVATION. To use **Rotate**:

1. Select **Rotate** from the **Edit** sub-menu of the **Boundaries** menu.
2. Use the mouse to **select** the excavation(s) you would like to rotate.
3. When you are finished selecting, press *Enter* or right-click and select *Done Selection*.
4. You will then be prompted to enter a base point for the rotation. Use the mouse or enter the x-y coordinates of the base point in the prompt line.
5. You will then be prompted to enter a rotation in degrees. The positive convention is counter-clockwise. Enter the rotation in the prompt line and press *Enter*, and all selected excavation(s) will be rotated about the base point.

Tips

- To rotate an excavation about its center, enter the centroid of the excavation as the base point.
- To rotate an excavation about any vertex, enter the vertex as the base point (this will leave the position of the vertex at the base point unchanged, and rotate all other vertices about the base point).

[Delete Boundaries](#)

[Move Boundaries](#)

[Add Vertices](#)

[Delete Vertices](#)

[Move Vertices](#)

[Scale Excavation](#)

[Copy Excavation](#)

[Selection Filter](#)

Copy Excavation

The **Copy** option can be used to make copies of EXCAVATIONS. To use **Copy**:

1. Select **Copy** from the **Edit** sub-menu of the **Boundaries** menu.
2. Use the mouse to **select** the excavation(s) you would like to copy.
3. When you are finished selecting, press *Enter* or right-click and select *Done Selection*.
4. You will then be prompted to enter a base point. Use the mouse or enter the x-y coordinates of the base point in the prompt line.
5. After the base point is entered, if you move the mouse, you will see an outline of your selected excavation(s) following the mouse. Enter a second point with the mouse or the keyboard, and the excavation(s) will be copied to the new location.

Tips

- To define a relative distance for the copy, enter 0 , 0 as the base point in the prompt line, and the relative distance as the second point. That is, if you wanted a copy of an excavation 20 meters above and 30 meters to the right of the original excavation, enter 0 , 0 as the base point, and 20 , 30 as the second point.
- If you do not know the relative distance, then enter a vertex of the original excavation as the base point, and the new coordinates of the equivalent vertex on the copied excavation, as the second point.

Delete Boundaries

Move Boundaries

Add Vertices

Delete Vertices

Move Vertices

Scale Excavation

Rotate Excavation

Selection Filter

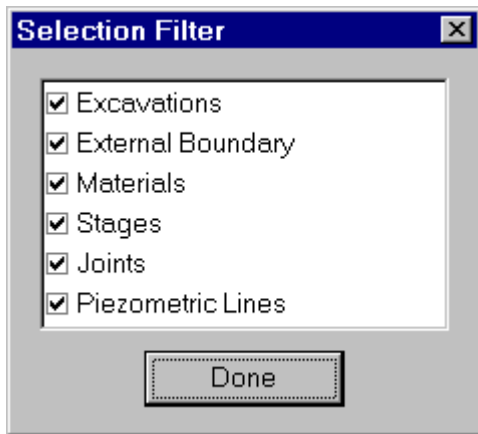
Selection Filter

The **Selection Filter** allows the user to choose which **boundary types** can be selected for editing.

The **Selection Filter** is available in:

- the **Edit** sub-menu of the **Boundaries** menu
- the right-click menu for all applicable options (eg. **Delete Boundary**, **Move Boundary** etc)

By default, all boundaries are initially selectable in the **Selection Filter** dialog. The main purpose of the **Selection Filter** is to allow the user to **select** boundaries with a window, but filter out the selecting of some boundary types which may be within the window.



Example

Let's say a model had numerous STAGE boundaries that the user wanted to delete, without deleting any EXCAVATIONS. They could turn "off" all boundary types in the **Selection Filter** except for STAGE boundaries, and then select all excavations with a window while using the **Delete Boundaries** option, and only the STAGE boundaries would be selected and deleted.

NOTE:

- The **Selection Filter** is automatically reset after each use, so that all boundaries are selectable, once the current operation has been completed.
- The **Selection Filter** can be used at any time during the selection process. For example, if you accidentally select boundaries that you did not intend to select, you can use the **Selection Filter** to un-select any given boundary type, even after boundaries have been selected.
- In addition to the boundary editing options, the **Selection Filter** is also applicable to other PHASE2 options. For example, while **Custom Discretizing**, Adding / Deleting

[Traction](#)s , Adding / Deleting [Lin](#)ers , etc, the **Selection Filter** is available in the right-click menu.

See the [Selecting](#) topic for more tips and information on selecting entities in PHASE2.

Delete Piezometric Line

The purpose of the **Delete Piezometric Line** option, is to allow the user to delete a Piezometric Line, if necessary, after the finite element mesh has been generated. This is because all regular boundary editing options in PHASE2 are disabled after the mesh is generated, since the mesh must be reset in order to add, edit or delete boundaries other than Piezometric Lines.

Unlike other boundary types in PHASE2, Piezometric Lines may be added or deleted, after the mesh has been generated, since a Piezometric Line boundary is independent of the finite element mesh, and is not involved in generating the mesh.

Note:

- The **Delete Piezometric Line** option works in the same manner as the **Delete Boundary** option. See the [Delete Boundary](#) topic for details.
- The **Delete Boundary** option can be used to delete a Piezometric Line, like any other boundary, before the mesh has been generated.
- If you wish to add, delete or move vertices of Piezometric Lines, then the mesh must be reset, since these editing options are disabled when a mesh exists.
- The **Delete Piezometric Line** option is available in the toolbar or the **Edit** sub-menu of the **Boundaries** menu.

Mesh Overview

After defining all boundaries, the next step is to create the finite element mesh. PHASE2 incorporates a state-of-the-art 2 dimensional automatic finite element mesh generator, which can generate meshes based on either triangular or quadrilateral finite elements.

To give the user maximum flexibility in defining the mesh, the mesh generation procedure is performed in two steps:

1. FIRST DISCRETIZE!!!

You must first discretize the boundaries by selecting the **Discretize** option. This process subdivides the boundary line segments into discretizations which will form the framework of the finite element mesh.

2. THEN MESH!!!

After discretizing, select the **Mesh** option, to generate the finite element mesh, which will be based on the discretization, and the mesh and element types in the Mesh **Setup** option.

NOTE:

- The main parameters controlling the discretization and meshing, are found in the **Mesh Setup** dialog.
 - Before the mesh has been generated, the discretization can be customized by the user with either the **Advanced Discretization** option in the **Mesh Setup** dialog, and / or the **Custom Discretize** option in the toolbar or the **Mesh** menu.
 - After the mesh has been generated, various options are available for refining or customizing the mesh, in the **Mesh Refinement** sub-menu of the **Mesh** menu.
-

Mesh Setup

The **Mesh Setup** option, available in the **Mesh** menu or the toolbar, controls the main parameters which define the discretization and meshing of the model. This includes:

Mesh Type

Element Type

Gradation Factor

Number of Excavation Nodes

Advanced Discretization

NOTE:

- The *Discretize* and *Mesh* buttons in the **Mesh Setup** dialog, allow the user to see the results of applying the current Mesh Setup parameters, without closing the dialog.
- It is important to note that these buttons are exactly equivalent to the use of the **Discretize** and **Mesh** options in the mesh menu or the toolbar.
- The current parameters in the **Mesh Setup** dialog, will be applied whenever *Discretize* or *Mesh* are selected from the toolbar, the menu, or the **Mesh Setup** dialog.
- Customizing of the discretization and mesh, beyond what is determined by the **Mesh Setup** dialog, can be done with the **Custom Discretize** option, and the options in the **Mesh Refinement** sub-menu in the mesh menu.

Mesh Type

The user can choose from 3 different *Mesh Types* in the **Mesh Setup** dialog:

Graded

Uniform

Radial

A *Graded* mesh will be used in almost all models, but *Uniform* or *Radial* meshing may occasionally be useful in special cases.

Note that each different *Mesh Type* requires different associated setup parameters, as described in the *Graded*, *Uniform* and *Radial Mesh Type* topics.

Graded Mesh

In the **Mesh Setup** dialog, the default *Mesh Type* is *Graded*, which produces a good graded mesh for most models, using what is known as a quadtree nodal insertion technique. In the majority of cases you will use a *Graded* mesh type.

When the *Mesh Type* = *Graded*, you will be required to specify the **Element Type** , *Gradation Factor* and *Number of Excavation Nodes* in the **Mesh Setup** dialog.

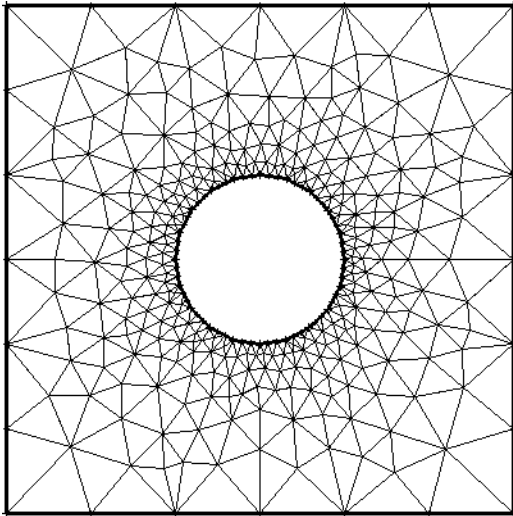
In the example below:

Mesh Type = *Graded*

Number of Excavation Nodes = 60

Gradation Factor = 0.1

Element Type = *Triangles*



Number of Excavation Nodes

The *Number of Excavation Nodes* directly determines the discretization of the EXCAVATION boundaries. If more than one EXCAVATION is defined, remember that this number is the **total** for **all** excavations.

PHASE2 subdivides the EXCAVATION boundaries in such a way that the number of excavation nodes is *approximately* equal to the *Number of Excavation Nodes* in **Mesh Setup**. You may notice that the actual number of excavation nodes after discretizing may differ slightly from the *Number of Excavation Nodes* in the **Mesh Setup** option. This is normal, and will vary depending on the excavation(s).

The *Number of Excavation Nodes* in conjunction with the *Gradation Factor*, determines the discretization of *all other boundaries* in the model, as described below.

Gradation Factor

The *Gradation Factor*, in conjunction with the *Number of Excavation Nodes*, determines the discretization of all other boundaries in the model (ie. EXTERNAL, MATERIAL, STAGE and JOINT).

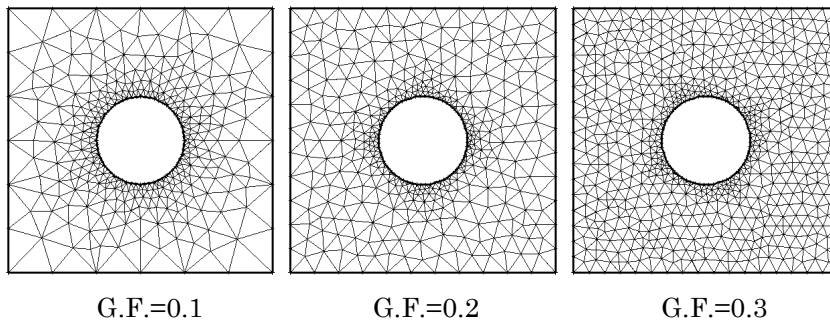
The *Gradation Factor* is the ratio of:

- the average length of discretizations on EXCAVATION boundaries, to
- the length of discretizations on the EXTERNAL boundary, at the maximum distance from the EXCAVATION boundaries

The discretization of all other boundaries in the model is graded, according to the distance of the boundary from EXCAVATION boundaries.

For a simple model where the EXTERNAL boundary is equally distant from the EXCAVATION boundary, a *Gradation Factor* of 0.1 implies that the average length of the EXTERNAL boundary discretizations will be (approximately) 10 times the average length of the EXCAVATION discretizations. This is illustrated below.

Due to the nature of the discretization process, the *Gradation Factor* should be considered an approximate number.



Effect of Gradation Factor on the mesh. From left to right, Gradation Factor = 0.1, 0.2, 0.3. (The *Expansion Factor* of the External Boundary is equal to 1 in this case).

As a rule of thumb, the farther away the EXTERNAL boundary is from the excavations (ie. the greater the *Expansion Factor*), the *smaller* the *Gradation Factor* should be, so that you do not end up with an unnecessarily large number of elements.

Number of External Boundary Nodes

Not all PHASE2 models will necessarily use EXCAVATION boundaries. In the case of a model with no EXCAVATION boundaries, for example:

- surface excavation models
- an AXISYMMETRIC model
- a "joints only" model
- any other model that does not require an EXCAVATION boundary

the **Mesh Setup** dialog will allow you to enter the *Number of External Boundary Nodes*, rather than the *Number of Excavation Nodes*. In this case, the *Gradation Factor* is not applicable and will be disabled, since the *Gradation Factor* is only relevant if EXCAVATION boundaries exist.

Uniform Mesh

In the **Mesh Setup** dialog, if the *Mesh Type = Uniform*, this will produce a mesh of approximately equally sized finite elements.

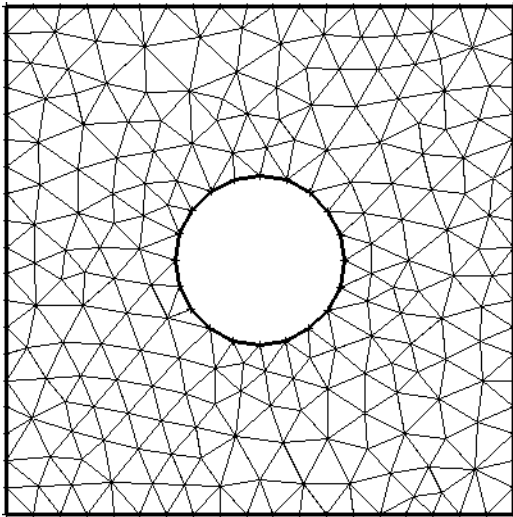
When the *Mesh Type = Uniform*, you will be required to specify the **Element Type** and the *Approximate Number of Elements*, in the **Mesh Setup** dialog.

In the example below:

Mesh Type = Uniform

Approximate Number of Elements = 300

Element Type = *Triangles*



Approximate Number of Elements

For a *Uniform* mesh, an *Approximate Number of Elements* is entered, rather than a *Gradation Factor* and *Number of Nodes*, because:

- The *Gradation Factor* is by definition, equal to 1 for a *Uniform* mesh.
- It is more intuitive to specify a total number of elements, rather than the number of boundary nodes, for a *Uniform* Mesh.

Remember that the *Approximate Number of Elements* is an *approximate* number. The actual number of elements generated is displayed in the status bar after meshing, and will in general be different from the *Approximate Number of Elements* entered in the **Mesh Setup** dialog.

Radial Mesh

A third *Mesh Type* is available in the **Mesh Setup** dialog, *Radial* meshing. Developed mainly for testing and verification purposes, it is intended for use with circular (or elliptical) excavations – other shapes may produce unpredictable results (you can experiment and see what happens). Notice that for a *Radial* mesh you must enter an *Expansion Factor* rather than a *Gradation Factor* (see below). Also, there are several modeling restrictions associated with a *Radial* mesh, as described below.

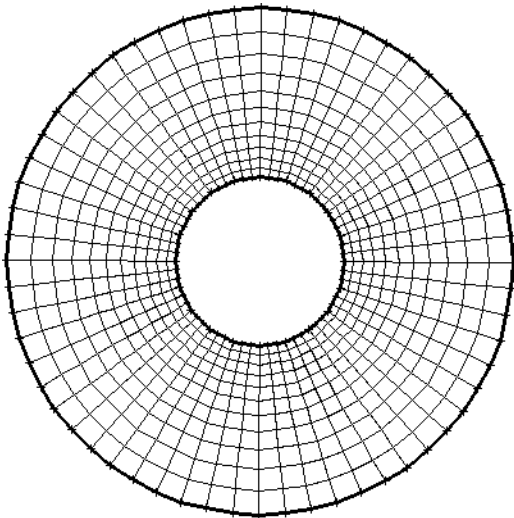
In the example below:

Mesh Type = Radial

Number of Excavation Nodes = 60

Expansion Factor = 1.0

Element Type = Quadrilaterals



Expansion Factor

Because of the way in which a *Radial* mesh is generated, the **Mesh Setup** dialog will ask you for an *Expansion Factor* rather than the *Gradation Factor*, when you toggle the *Mesh Type* to *Radial*.

Note that the *Expansion Factor* for a *Radial* mesh in the **Mesh Setup** dialog is the same as the *Expansion Factor* which is entered for an automatically generated **EXTERNAL boundary**. Since the EXTERNAL boundary generation is integrated with the *Radial* mesh generation, this allows the user to change the *Expansion Factor* for a *Radial* mesh without having to select the **Add External** option to access the Create External Boundary dialog.

Also note that the *Gradation Factor* for a *Radial* mesh is automatic, and cannot be directly altered by the user, except by changing the *Number of Excavation Nodes*.

Creating a Radial Mesh

Radial meshing was developed mainly for testing and verification purposes, since it gives a reproducible*, symmetrical radial mesh for a single symmetrical excavation. You may want to use it in cases where you have a symmetrical (preferably circular or elliptical) excavation.

(* For a given discretization, *Graded* or *Uniform* meshing will actually produce a different mesh each time you re-mesh the model. A *Radial* mesh will always produce exactly the same mesh, for a given discretization.)

Since it is a specialized mesh generation option, there are several restrictions on the use of *Radial* meshing:

1. Only a single excavation can be present.
2. MATERIAL, STAGE and JOINT boundaries are not permitted.
3. In general, it is intended for use with a single, convex, symmetric excavation.

To create a *Radial* mesh, use the following steps:

1. Create your excavation.
2. Instead of adding the EXTERNAL boundary, go to the **Mesh Setup** option and toggle the *Mesh Type* to *Radial*. Note that for a *Radial* mesh, an *Expansion Factor* is entered rather than a *Gradation Factor*. The *Expansion Factor* has the same effect as the *Expansion Factor* in the [Create External Boundary](#) dialog.
3. Select **Discretize** to discretize the excavation.
4. Select **Mesh** and the *Radial* mesh will be generated. As you will see, the EXTERNAL boundary will appear at the end of the *Radial* mesh generation.
5. Although you cannot directly control the **Gradation Factor** for a *Radial* mesh, you can control the fineness and/or extent of the mesh by changing the *Number of Excavation Nodes* and/or the *Expansion Factor* in **Mesh Setup**.

Element Type

The user can choose between 4 different finite *Element Types* in the **Mesh Setup** dialog:

- 3 NODDED TRIANGLE
- 6 NODDED TRIANGLE
- 4 NODDED QUADRILATERAL
- 8 NODDED QUADRILATERAL

NOTE:

- If you are using elements with midside nodes (ie. 6 noded triangles or 8 noded quadrilaterals), the midside nodes will be displayed on the mesh as small circular dots at the midpoint of each finite element edge. The display of the midside nodes can be turned on or off at any time, in the **Display Options** dialog.
- The user should be aware that using elements with midside nodes will greatly increase the size of the matrices used to solve the problem, and will therefore increase solution time and memory requirements.

Advanced Discretization

The *Advanced Discretization* option in the **Mesh Setup** dialog, allows the user maximum flexibility in customizing the discretization of the model boundaries.

In most cases, the regular discretization procedure, of specifying the *Number of Excavation Nodes* for all excavations, and the *Gradation Factor*, will be sufficient to obtain suitable discretization of boundaries. However, with the *Advanced Discretization* options, the user has complete control of the discretization of any part of the model.

Advanced Discretization is only applicable if the *Mesh Type* = *Graded* in the **Mesh Setup** dialog. To use *Advanced Discretization*:

1. Select the *Advanced Discretization* checkbox, and the **Mesh Setup** dialog will expand to display the *Advanced Discretization* options.
2. The *Number of Nodes* for each individual excavation boundary of your model, can be customized by entering the desired number of nodes for any excavation. Note that by default, when you initially select *Advanced Discretization*, the total number of nodes for all excavations, will be approximately equal to the *Default Number of Nodes for All Excavations*, in the main part of the **Mesh Setup** dialog.
3. *User Defined Regions* can be created, by using the *Add* button, to select any rectangular region of the model. All boundaries within the *User Defined* region, can then be assigned a *Number of Nodes*, just like an excavation boundary.
4. To delete *User Defined Regions*, use the *Delete* button.
5. To reset all *Advanced Discretization* parameters to the defaults, select the *Reset* button.
6. Note that the *Element Length* displays the average *Element Length*, for each *Region Type*. However, the user may also specify an *Element Length*. To do this, enter an *Element Length* for the desired *Region Type*. Then click the mouse anywhere in the grid, and the *Number of Nodes* will be updated, to the required number necessary to achieve the approximate *Element Length* entered.

NOTE:

- Since the *Advanced Discretization* option is within the **Mesh Setup** dialog, all *Advanced Discretization* settings will be preserved, even if the Discretization has been reset with the **Reset Mesh** option in the **Mesh** menu or the toolbar.
- The **Custom Discretize** option in the **Mesh** menu or toolbar, can also be used to specify custom discretizations along model boundaries. However, all custom discretization is lost if the Discretization has been reset with the **Reset Mesh** option in the **Mesh** menu or the toolbar.

Discretize

The **Discretize** option in the toolbar or the **Mesh** menu, will discretize all model boundaries, according to the settings in the **Mesh Setup** dialog.

The discretization of the model boundaries, forms the framework for the finite element mesh, and is indicated by small red crosses subdividing the boundary line segments. Each cross indicates the position of a finite element node on the boundary. The Status Bar will indicate the number of discretizations created for each boundary type. You must always select **Discretize** before you can generate the mesh.

If you are not satisfied with the discretization, you can:

1. Change the settings in the **Mesh Setup** dialog (eg. *Number of Nodes*, *Gradation Factor*, or use the *Advanced Discretization* settings), and then re-select **Discretize**.
2. Use the **Custom Discretize** option.

After discretizing, the finite element mesh is generated by selecting the **Mesh** option.

NOTE:

- The **Discretize** option is also available as a button in the **Mesh Setup** dialog.
 - The **Reset Mesh** option can be used at any time to reset the mesh and / or discretization.
-

Custom Discretize

After selecting **Discretize**, you may need to tailor the discretization on one or more boundary line segments. This can be done with the **Custom Discretize** option. To Custom Discretize:

1. Select **Custom Discretize** from the toolbar or the **Mesh** menu.
2. Use the mouse to **select** the boundary line segments that you would like to Custom Discretize.
3. When you are finished selecting, press *Enter* or right-click and select *Done Selection*.
4. You will see a dialog. Enter the number of discretizations that you would like to apply to EACH selected segment.
5. Select *OK*, and all selected line segments will be re-discretized with the number you entered in the dialog.
6. Repeat steps 1-5 if different segments require different discretizations, since only one number at a time can be entered as a custom discretization.

Custom Discretize always applies a uniform discretization to the selected line segments, using the user-supplied number of discretizations per line segment. This will replace the "graded" discretization which exists by default, on boundaries outside of excavation boundaries. If you wish to simulate a "graded" discretization along a boundary using **Custom Discretization**, you can **insert vertices** along a boundary to divide it up into smaller segments, and then apply a Custom Discretization to each segment.

NOTE:

- You must always use the **Discretize** option first, to produce an initial discretization, before you can **Custom Discretize**.
- Before using **Custom Discretize**, you may want to first try changing the **Mesh Setup** parameters (eg. *Number of Excavation Nodes* and / or *Gradation Factor*), and re-selecting **Discretize**. This may be sufficient to create a discretization to your liking.
- Discretization can also be customized using the *Advanced Discretization* option in the **Mesh Setup** dialog. See the **Advanced Discretization** topic for details.
- Anytime the discretization is reset (eg. with the **Reset Mesh** option), all **Custom Discretization** *will be lost*. If you accidentally reset a Custom Discretization, remember that you can always use the **Undo** option to Undo any actions in PHASE2.

Mesh

After the boundaries are **Discretized**, select the **Mesh** option from the toolbar or the **Mesh** menu, and the finite element mesh will be generated, with no further intervention by the user.

The mesh will be generated according to the **Mesh Type** and **Element Type** in the **Mesh Setup** dialog. The total number of elements and number of nodes in the mesh will be displayed in the Status Bar.

NOTE:

- The **Mesh** option is also available as a button in the **Mesh Setup** dialog. This allows you to experiment with different mesh and discretization parameters, and view the results, while using the **Mesh Setup** dialog.
- After the mesh has been generated, options are available in the Mesh Refinement sub-menu of the **Mesh** menu, which allow the user to refine or customize any area of the mesh. See the **Mesh Refinement** topic for details.
- The **Reset Mesh** option can be used at any time to reset the mesh and / or discretization.

NOTE TO PHASE2 version 4.0 USERS:

- As of PHASE2 version 5.0, regions within excavation boundaries, are now always meshed, even for single stage models. Previously, for a single stage model, the region within an excavation boundary was NOT meshed. This change was made to improve the consistency and flow of the PHASE2 modeling procedure. Users will now have to excavate regions within excavation boundaries, even for single stage models, with the *Excavate* button in the **Assign** dialog.

Mesh Refinement

The options in the **Mesh Refinement** sub-menu of the **Mesh** menu, provide the user with some very powerful options for customizing the finite element mesh, after the mesh has been generated. These include:

- Increasing the density of the mesh in any area with the **Increase Density** option
- Creating a regular (mapped) mesh in any closed region bounded by 3 segments (**Mapped Triangular Region**) or 4 segments (**Mapped Rectangular Region**). See the [Mapped Meshing](#) topic for a discussion of these options.
- Troubleshooting mesh problems with the **Show Mesh Quality** and **Define Mesh Quality** options (see the [Mesh Quality](#) topic for a discussion of these options)

Remember that these options require that the finite element mesh has already been generated, with the [Mesh](#) option.

Increase Density

The **Increase Density** option allows the user to select any area of the mesh within a rectangular window, to increase the density of the mesh within the window. To use this option:

1. Select **Increase Density** from the **Mesh Refinement** sub-menu of the **Mesh** menu.
2. Use the mouse (or the keyboard) to enter two points defining opposite corners of a rectangular region, in which you would like to increase the mesh density.
3. After the second corner of the window has been entered, the density of the mesh within the selected region will be automatically increased, according to the following rules:
 - A new node is automatically inserted at the center of each finite element, within the selected region.
 - A new mesh is then generated, which uses ALL OF THE ORIGINAL nodes which existed, as well as the new nodes inserted at the element centers within the selected region.

If the results of using **Increase Density** are unsatisfactory, remember that you can always use the [Undo](#) option to Undo any actions in PHASE2, or you can use **Increase Density** any number of times, until the mesh is sufficiently refined.

NOTE:

- The discretization of the model boundaries is NOT affected by the **Increase Density** option. Only the mesh density within the boundaries is affected. If you need to increase or modify the density of nodes on the boundaries, you must do this BEFORE the mesh is generated, using either the options in [Mesh Setup](#), or with the [Custom Discretize](#) option.
-

Mapped Meshing

There are situations when it is necessary or desirable, to create a geometrically regular finite element mesh. This is referred to as "mapped" meshing. In PHASE2, a mapped mesh may be created within triangular or rectangular regions, using the **Mapped Triangular Region** or **Mapped Rectangular Region** options in the **Mesh Refinement** sub-menu of the **Mesh** menu.

Mapped Triangular Region

The **Mapped Triangular Region** option allows the user to create a regular (mapped) mesh, within any region bounded by THREE line segments. NOTE:

- The three line segments must form a closed region
- Each of the three line segments MUST have the same number of nodes (discretizations). In most cases, you will have to use the **Custom Discretize** option, in order to ensure that each line segment has the same number of nodes.

If the above two conditions are met, then you may use the **Mapped Triangular Region** option as follows:

1. Select **Mapped Triangular Region** from the **Mesh Refinement** sub-menu of the **Mesh** menu.
2. Use the mouse to select three boundary line segments which form a closed triangular region.
3. Right-click the mouse and select *Done Selection* (or press *Enter*), and a regular mapped mesh will automatically be generated, within the triangular region. The mapped mesh will be generated according to the discretization of the three line segments, and the *Element Type* in **Mesh Setup** (*Triangles* or *Quadrilaterals*).

Mapped Rectangular Region

The **Mapped Rectangular Region** option allows the user to create a regular (mapped) mesh, within any region bounded by FOUR line segments. NOTE:

- The four line segments must form a closed region, and the region must be concave (convex regions not allowed).
- Opposite line segments of the rectangular region, MUST have the same number of nodes (discretizations). In most cases, you will have to use the **Custom Discretize** option, in order to achieve this. (Unlike the **Mapped Triangular Region** option, it is not necessary for ALL line segments to have the same discretization).

If the above conditions are met, then you may use the **Mapped Rectangular Region** option as follows:

-
1. Select **Mapped Rectangular Region** from the **Mesh Refinement** sub-menu of the **Mesh** menu.
 2. Use the mouse to select FOUR boundary line segments which form a closed rectangular region.
 3. When you are finished selecting line segments, right-click the mouse and select *Done Selection* (or press *Enter*), and a regular mapped mesh will automatically be generated, within the rectangular region. The mapped mesh will be generated according to the discretization of the four line segments, and the *Element Type* in **Mesh Setup** (*Triangles* or *Quadrilaterals*).

NOTE:

- Either *Triangular* or *Quadrilateral* finite elements may be used with either mapped meshing option. However, if *Quadrilateral* elements are used with a Mapped Triangular Region, then some of the elements will necessarily be triangles.
 - If the mesh is reset with the **Reset Mesh** option, all mapped meshing will be lost. However, remember that you can always use the **Undo** option to Undo any actions in PHASE2.
-

Mesh Quality

In a finite element mesh, it is generally desirable to avoid elements of high aspect ratio (ie. long "thin" elements). The presence of such elements can have adverse effects on the analysis results.

- In general, such elements can influence analysis results, and lead to misleading and inaccurate results, which are dependent on the mesh.
- In extreme cases, such elements may even be responsible for non-convergence of the finite element solution, and the analysis will be aborted.

In order to help the user determine the "quality" of a finite element mesh, PHASE2 can automatically locate and highlight elements in a mesh, which are deemed to be of "poor" quality, according to user-definable criteria.

This is done with the **Show Mesh Quality** and **Define Mesh Quality** options, in the **Mesh Refinement** sub-menu of the **Mesh** menu.

Show Mesh Quality

In order to locate and highlight poor quality elements in a mesh, select the **Show Mesh Quality** option in the **Mesh Refinement** sub-menu of the **Mesh** menu. If poor quality elements exist:

- All poor quality elements will be filled with a red hatch pattern
- Since these elements will often be difficult to see, without zooming in to view, the general area(s) where poor quality elements have been found, will be additionally highlighted, by blue rectangular boxes drawn around the regions of poor quality elements. The display of these boxes can be turned off if desired, by de-selecting the *Highlight* checkbox in the **Define Mesh Quality** dialog.

The elements which are highlighted, are those which conform to the criteria of a "poor quality" element, as defined in the **Define Mesh Quality** dialog. In most cases, you will need to zoom in to get a good view of the highlighted poor quality elements, and determine what may be the cause of the problem.

Define Mesh Quality

The **Define Mesh Quality** dialog allows the user to specify the criteria which define a "poor quality" element. There are three criteria which may be specified:

- ratio of (maximum side length) / (minimum side length)
- Minimum interior angle
- Maximum interior angle

The defaults for the above criteria, are 10 , 20 and 120, respectively. However, the user may change any of these criteria. NOTE that an element will be determined as "poor quality", if ANY of these three criteria are met.

Highlight

By default, the *Highlight* checkbox is toggled ON in the **Define Mesh Quality** dialog. This option draws an additional blue rectangle around regions of poor quality elements, since in many cases the poor quality elements may be difficult to see, even with the red hatch fill, unless you are zoomed in to that region of the mesh. If you do not wish to see these blue rectangles displayed, then you can turn the *Highlight* option OFF by clearing the checkbox.

Number of divisions for statistics

Statistics for poor quality elements can be found listed in the **Info Viewer**. These statistics allow the user to examine the quantity and distribution of poor quality elements, for each of the three quality criteria. The *Number of divisions for statistics* in the **Define Mesh Quality** dialog, allows the user to define the number of divisions used for calculating the distributions tabulated in the **Info Viewer**. For a demonstration of this, it is suggested that the user view the Mesh Quality statistics listed in the **Info Viewer**.

Reasons for poor quality mesh elements

There are various reasons why poor quality mesh elements may be generated. Typically the problems are due to:

1. Poorly graded discretization of model boundaries (for example, a finely discretized boundary immediately adjacent to, or close to, a coarsely discretized boundary, will often result in a poor quality mesh).
2. Vertices which are very close to each other. This can happen when a boundary has been intersected with another boundary, without using vertex snap, such that two vertices end up very near to each other, when the user may have intended both vertices to be at the same location. Due to the automatic boundary intersecting capability of PHASE2, this may occur without the user being aware of it.
3. The geometry of your model boundaries may also be responsible, for example, the relative distances between boundaries, or the way in which they intersect (eg. nearly parallel boundaries which intersect at a very small angle).

Fixing a poor quality mesh

1. If a poor discretization is responsible for the poor mesh, then you will need to reset the mesh, and modify the discretization of the appropriate boundaries. This may involve use of the **Custom Discretize** option, or the **Advanced Discretization** option in the **Mesh Setup** dialog, or simply changing the main **Mesh Setup** parameters (eg. *Number of Excavation Nodes*, *Gradation Factor*)
2. If the problem is vertices which are very close to each other, then you may need to delete unnecessary vertices, or move vertices, so that they are further apart, or exactly coincident. If vertices were intentionally located very near to each other, and this is

causing a problem, then you may want to consider modifying your geometry slightly, so that poor quality elements are not generated.

3. If the actual model geometry is responsible, then various solutions might be appropriate, involving **Custom Discretize**, or modifying the boundaries slightly, so that the problem does not occur.

Reset Mesh

To reset the finite element mesh and / or the discretization, select **Reset Mesh** from the toolbar or the **Mesh** menu.

Resetting the mesh is separated into two steps:

1. Resetting the mesh.
2. Resetting the discretization.

This is because there will be times when you only want to reset the mesh, but not the discretization. For example:

- you may want to preserve a **Custom Discretization** for further customizing (remember that all Custom Discretization is lost when the discretization is reset), or
- you may only want to change the **Element Type** from triangles to quads, or vice versa, but retain the same discretization.

Tips

- If the model has been discretized but not meshed, then **Reset Mesh** will reset the discretization only.
- **Reset Mesh** does not reset any of the **Mesh Setup** options, these can only be changed through the **Mesh Setup** dialog.
- If you are using the *Radial* mesh type, then **Reset Mesh** will also delete the EXTERNAL boundary when the mesh is reset, since this is integrated with the *Radial* mesh generation.
- If your model includes support (ie. **BOLTS** or **LINERS**), then **Reset Mesh** will delete all support as well as the mesh. This is because the support must be defined *after* the mesh is generated in PHASE2.
- If you delete the mesh or support accidentally, remember that you can always use the **Undo** option to Undo any actions in PHASE2. Also, note that BOLT geometry can be **saved to a DXF file** , and then read back in again – you may find this useful for saving complex bolting patterns.

Field Stress

The **Field Stress** option, in the toolbar or the **Loading** menu, allows the user to define the in-situ stress conditions prior to excavation. Two options are available for defining field stress in PHASE2, *Constant* or *Gravity* field stress.

Constant Field Stress

The *Constant* field stress option is used to define a constant stress field prior to excavation (ie. one which does not vary with position). See the [Constant Field Stress](#) topic for details.

Gravity Field Stress

The *Gravity* field stress option is used to define a gravity stress field which varies linearly with depth from a user-specified *Ground Surface Elevation*. *Gravity* field stress is typically used for surface or near surface excavations. See the [Gravity Field Stress](#) topic for details.

Custom Field Stress

For either *Constant* or *Gravity* field stress, the field stress for individual materials may be customized, if necessary, by selecting the *Advanced* button in the **Field Stress** dialog, and specifying the field stress for any material(s) in the model. See the [Custom Field Stress](#) topic for details.

Stress Block

The relative magnitude and direction of the *In Plane* principal field stresses, are indicated by the **Stress Block**, displayed in the upper right corner of the view. If the **Stress Block** looks wrong then make sure you have correctly entered field stress parameters (ie. *Sigma 1*, *Sigma 3* and *Angle*, for a *Constant* field stress, or the *Horizontal / Vertical In Plane Stress Ratio*, for *Gravity* field stress). The display of the **Stress Block** can be toggled on or off with the [Stress Block](#) option in the **View** menu.

Load Splitting

If your model is staged (ie. the *Number of Stages* in the [Project Settings](#) dialog is greater than 1), then by default, 100% of the field stress induced load is applied to the model in the **FIRST** stage. However, for staged models, the user has the option of "splitting" the field stress induced load, between any desired stages of the model (rather than applying the entire load in the first stage). This is done with the **Load Split** option in the **Loading** menu, see the [Load Split](#) topic for details.

Also note:

- In PHASE2 it is assumed that the stress field is aligned with the analysis section – ie. one of the three in-situ principal stresses must be aligned with the longitudinal axis of the excavation. A fully 3-dimensional, arbitrarily oriented stress field cannot be entered in PHASE2.

-
- Since the yield functions used in the COMPUTE engine are three-dimensional (ie. they depend on all components of the stress tensor) it is important that the user enters a stress (or stress ratio) in the out-of-plane direction.
 - The convention for stress is POSITIVE COMPRESSION, so in most cases you will be entering positive values for stresses.

Constant Field Stress

Use the *Constant* field stress option to define a constant stress field prior to excavation (ie. one which does not vary with position). In the **Field Stress** dialog, the following parameters are required to define a *Constant* field stress:

Sigma 1

Sigma 1 is the MAJOR IN-PLANE principal field stress.

Sigma 3

Sigma 3 is the MINOR IN-PLANE principal field stress.

Sigma Z

Sigma Z is the OUT-OF-PLANE field stress.

Angle

The *Angle* is measured between the positive x-axis and the direction of *Sigma 1*. The *Angle* is measured in degrees, and the *counter-clockwise direction is defined as positive*.

NOTE: Remember that the **Stress Block** indicates the relative magnitude and direction of the *In-Plane* principal field stresses. If the **Stress Block** looks wrong then try entering the field stress data again, keeping in mind the definition of *Angle*. The display of the **Stress Block** can be toggled on or off in the **View** menu.

Custom Field Stress

Customized field stress may be defined for individual materials in a model, by selecting the *Advanced* button, and defining custom field stress as required. See the **Custom Field Stress** topic for details.

Axisymmetric Constant Field Stress

If you are modeling an **AXISYMMETRIC** problem (this is set in the **Project Settings** dialog), then the **Field Stress** dialog will ask you for:

- *Horizontal Stress*
- *Vertical Stress*

(rather than Sigma 1, Sigma 3, Sigma Z and Angle).

It is very important to note that *Vertical* and *Horizontal* refer to the setup of your problem in PHASE2, and not necessarily the true orientation of your excavation. The *Vertical* stress is

the stress in the **axial** direction (ie. the axis of rotational symmetry), and the *Horizontal* stress is the field stress **perpendicular** to the axis of symmetry. The terms *Vertical* and *Horizontal* are used, because in PHASE2, an axisymmetric model must always be defined such that the axial direction is oriented with the y-axis of the coordinate system.

Out-of-plane (or circumferential) field stress exists, but is automatically set equal to the *Horizontal* (or radial) stress, and cannot be independently varied by the user.

See the PHASE2 User's Guide for a tutorial on AXISYMMETRIC modeling.

Gravity Field Stress

The *Gravity* field stress option is used to define a gravity stress field which varies linearly with depth from a user-specified *Ground Surface Elevation*. *Gravity* field stress is typically used for surface or near surface excavations. In the **Field Stress** dialog, the following parameters are required to define a *Gravity* stress field:

Ground Surface Elevation

The *Ground Surface Elevation* is the vertical (y) coordinate of the ground surface with respect to your coordinate system. It is NOT the depth below ground surface of your excavations!

NOTE: When you are using *Gravity* field stress, a horizontal dotted line marking the *Ground Surface Elevation* will automatically be displayed on the model. The display of this line can be toggled on or off in the **View** menu.

Unit Weight of Overburden

The *Unit Weight Of Overburden* (note units: MN / m³) is used, in conjunction with the *Ground Surface Elevation*, to calculate the **vertical** component of the gravitational field stress, at any point in the model.

Stress Ratios

The *Horizontal / Vertical* stress ratios (*In-Plane* and *Out-Of-Plane*), are used to calculate the **horizontal** components of the gravitational field stress, based on the vertical component at any point in the model.

NOTE: Remember that the **Stress Block** indicates the relative magnitude and direction of the *In-Plane* principal field stresses. For a *Gravity* field stress, the **Stress Block** is always oriented with the vertical and horizontal axes. If the **Stress Block** looks wrong then check the value of the *In-Plane Horizontal / Vertical Stress Ratio*.

Locked In Stress

You may also include horizontal *Locked In* stress components in the *In Plane* and / or *Out-Of-Plane* directions. A *Locked In* stress component means that the horizontal field stress will be equal to the *Locked In* stress at the *Ground Surface Elevation*, and this fixed component of stress will be added to the horizontal field stress, below the *Ground Surface Elevation*, to determine the total horizontal stress at any point.

Custom Field Stress

Customized field stress may be defined for individual materials in a model, by selecting the *Advanced* button, and defining custom field stress as required. See the **Custom Field Stress** topic for details.

Custom Field Stress

There are situations when it is required that the user be able to specify the field stress for individual materials in a model, rather than only a single field stress which applies to all materials.

For example, individual layers of material in bedded geological structures, will sometimes have a field stress which is different from the field stress in surrounding layers. When this situation occurs, the user can define a customized field stress for any material(s) in their model, as follows:

1. In the **Field Stress Properties** dialog, select the *Advanced* button.
2. The dialog will expand, allowing the user to define a customized field stress, for any material(s) in their model. Note that the materials in the **Field Stress Properties** dialog, correspond to the materials you have defined in the **Define Material Properties** dialog.
3. To define customized field stress for a material, select the *Apply Custom Field Stress* checkbox, for the desired material, and enter the field stress parameters which apply to the material.
4. Repeat Step 3, for each material which requires a custom field stress.

NOTE: the default field stress properties, in the main (upper) part of the **Field Stress Properties** dialog, will still apply to all materials which have NOT had custom field stress properties specified.

Load Splitting

By default in PHASE2, the entire load on the model due to Field Stress, is applied in the first stage of the model.

The **Load Split** option in PHASE2 allows the user to "split" the field stress induced load, between any stages of the model, rather than applying the entire field stress load in the first stage. A typical use of Load Splitting, is to simulate the delayed installation of support, as discussed at the end of this topic.

To implement Load Splitting in PHASE2:

1. First note that the **Load Split** option is only applicable, and is only enabled if your model is STAGED (ie. the *Number of Stages* in the **Project Settings** option is greater than 1).
2. Select **Load Split** from the **Loading** menu.
3. In the Load Split dialog, select the *Enable Load Split* checkbox.
4. Enter the Load Split information at each stage (*Split Factor* and *Stress Behaviour*) as described below.

Split Factor

A *Split Factor* may be defined at each stage of the model. The *Split Factor* is used as follows:

- The *Split Factor* is simply multiplied by the values of the **Field Stress**, to obtain the actual Field Stress which will be applied at that stage.
- In most cases, a *Split Factor* will be a number between 0 and 1. However, in the interests of generality, numbers greater than 1 may be entered (this allows you to increase the Field Stress entered in the Field Stress dialog), or even negative numbers may be entered (allowing you to decrease field stress).
- For example, if you were using a Constant Field Stress with $\text{Sigma } 1 = 20$, and $\text{Sigma } 3 = \text{Sigma } Z = 10$, and the *Split Factor* = 0.5 at a given stage, then the actual Field Stress which will be applied at that stage, will be $\text{Sigma } 1 = 10$, and $\text{Sigma } 3 = \text{Sigma } Z = 5$.
- The Field Stress will be applied to the model according to the *Stress Behaviour* (ie. *Initialize* or *Add*, see below for details).

Stress Behavior

- When the *Stress Behaviour* = *Initialize*, the Field Stress at that stage will be INITIALIZED to the value determined by the *Split Factor* at that stage.
- When the *Stress Behaviour* = *Add*, the Field Stress determined by the *Split Factor*, will be ADDED to the existing stress state of the model, at the indicated stage.

NOTE:

-
- By default, for all materials, the *Stress Behaviour* = *Initialize* at Stage 1, and *Stress Behaviour* = *Add* in all subsequent stages. The user cannot change this. However, with Custom Load Splitting (see below), the user may define the *Stress Behaviour* (*Initialize* or *Add*), for any material in the model, at any stage.
-

Custom Load Split

To allow the user maximum flexibility in defining a Load Split, the Custom Load split option is available, which allows the user to define *Split Factor* and *Stress Behaviour* on a per material basis. To define a Custom Load Split:

1. Select the *Advanced* button in the **Load Split** dialog.
2. The dialog will expand, allowing the user to define a customized Load Split, for any material(s) in their model. Note that the materials in the **Load Split** dialog, correspond to the materials you have defined in the *Define Material Properties* dialog.
3. To define a Custom Load Split for a material, select the *Custom Load Split* checkbox, for the desired material, and enter the *Split Factor* and *Stress Behaviour* information for the material.
4. Repeat Step 3, for each material which requires a Custom Load Split.

NOTE: the default Load Split information, in the main (upper) part of the **Load Split** dialog, will still apply to all materials which have NOT had a Custom Load Split specified.

Typical Use of Load Split

A typical use of Load Splitting, is to simulate the delayed installation of support. For example, in a simple two stage model of an excavation:

- You could set the *Split Factor* = 0.3 in Stage 1 and *Split Factor* = 0.7 in Stage 2.
- Install support (eg. bolts and / or liners) in Stage 2.

When you run the analysis, only 30% of the total Field Stress load will be applied in the first stage. Effectively, this allows some displacement to take place in the first stage, without support, simulating the delayed installation of support at the face of an advancing tunnel, for example. Then, support installed in the second stage can react to the remainder (70%) of the field stress load.

An example of the use of load splitting can be found in the "Support Tutorial" in the PHASE2 User's Guide.

Other uses of the **Load Split** option are also possible, due to the general formulation of the Load Split / Custom Load Split option in PHASE2.

Seismic Loading

The **Seismic Loading** option in the **Loading** menu allows the user to include the effect of pseudo-static earthquake loading in the finite element analysis.

In the Seismic Load dialog, the user may define seismic coefficients for the *Horizontal* and / or *Vertical* directions.

The Seismic Coefficients are dimensionless coefficients which represent the (maximum) earthquake acceleration as a fraction of the acceleration due to gravity. Typical values are in the range of 0.1 to 0.3.

Seismic Force

If seismic coefficients are defined, a seismic Body Force will be applied to *each finite element in the mesh*. The seismic force which is applied to each finite element is as follows:

$$\text{Seismic Force} = \text{Seismic Coefficient} * \text{area of element} * \text{Unit Weight of element material}$$

Note that the acceleration due to gravity ($g = 9.81 \text{ m/s}^2$) is already incorporated into the *Unit Weight* of Material (remember that the dimensions of *Unit Weight* are MN / m^3), and therefore does not explicitly appear in the above equation. (The *Unit Weight* of material is the *Unit Weight* which is defined for each material in the Define Material Properties dialog, when Body Force is included with the *Initial Element Loading*, see below for details). Alternatively, the seismic force equation can be expressed as:

$$\text{Seismic Force} = \text{Seismic Coefficient} * \text{Body Force (due to gravity)}$$

NOTE:

- Body Force (due to gravity) simply represents the self-weight of a finite element.
- It is very important to note that Seismic Force only applies to materials which have an *Initial Element Loading* which includes Body Force. *Initial Element Loading* is specified in the **Define Material Properties** dialog. See the **Initial Element Loading** topic for more information.
- The Seismic Force will be applied as an additional Body Force to each element, and is vectorially added to the downward Body Force which exists due to gravity.
- If Body Force is NOT included in the *Initial Element Loading*, then the applied Seismic Force will be ZERO, even if seismic coefficients are defined in the Seismic Load dialog. You must remember this if you want to use Seismic Load in combination with **Constant Field Stress**, since by default, the *Initial Element Loading* for Constant Field Stress is *Field Stress Only*.

Finally note the direction conventions –

-
- a POSITIVE Horizontal coefficient represents a force applied to the RIGHT.
 - a NEGATIVE Horizontal coefficient represents a force applied to the LEFT.
 - A POSITIVE Vertical coefficient represents a force applied UPWARDS.
 - A NEGATIVE Vertical coefficient represents a force applied DOWNWARD.

When carrying out a finite element analysis with Seismic Loading, you should run multiple analyses using different combinations and directions of Horizontal and Vertical seismic coefficients, in order to evaluate the effect of seismic force direction on the analysis results (eg. stresses, displacements, strength factor etc).

Tractions Overview

In PHASE2 you can apply tractions (ie. distributed surface loads) to EXCAVATION, EXTERNAL, MATERIAL or STAGE boundaries. In a multi-stage model, *tractions can be staged*.

There are three different options available for applying tractions in PHASE2:

- NORMAL / SHEAR
- HORIZONTAL / VERTICAL
- ANGLE

These options allow the user to choose the most convenient way of applying a load to a boundary. However, note that *a given load could be specified using any one of the three options*.

Traction Units

Although traction magnitudes are entered as MN / m, in reality they represent surface loads in MN / m², when considering the out-of-plane dimension. For a Plane Strain model, each traction therefore represents an infinite strip load.

NOTE:

- Tractions cannot be applied until the finite element mesh has been generated.
 - If the mesh is reset, all existing tractions will be automatically deleted. If you accidentally reset the mesh, remember that you can always Undo any actions in PHASE2 with the **Undo** option in the **Edit** menu.
-

Normal / Shear Traction

In the majority of cases, you will simply want to add a NORMAL surface load to a boundary. The easiest way to do this is with the **Normal / Shear** tractions option:

1. Select **Normal / Shear** from the toolbar, or the **Tractions** sub-menu of the **Loading** menu.
2. In the dialog you will see, enter the magnitudes of the *Normal* and *Shear* components of your traction, and note the sign conventions. In most cases you will probably want to add a *Normal* traction only. To do this, enter a *Normal* magnitude, and leave the *Shear* component = 0.
3. In a multi-stage model, tractions can be staged by selecting the *Stage traction* checkbox, and entering a *Factor* for the load magnitude at each stage. If you do NOT stage the traction on a multi-stage model, then by default it will be applied, with the same magnitude, at all stages. See below for details.
4. Select **OK** in the dialog, and you will be prompted to **select** the boundary line segment(s) on which to apply the traction. Remember you can select EXCAVATION, EXTERNAL, MATERIAL or STAGE boundary segments.
5. When finished selecting, press *Enter* or right-click and select *Done Selection*, and your traction will be applied to all selected segment(s), as indicated by the display of red arrows and the magnitude.
6. Check the direction of the arrows and the magnitude, to make sure that this is the traction you wished to apply. If there is a mistake, repeat steps 1 to 5 and make sure the *Normal* and *Shear* magnitudes are correct. Note that minus signs can be used in front of the magnitudes, to change the direction of the *Normal* or *Shear* components.

Staging Traction

On a multi-stage model, tractions can easily be staged by selecting the *Stage traction* checkbox in the Traction dialog (Step 3 above), and entering appropriate factors for the load magnitude at each stage. NOTE:

- *Factor* = 1 means that the load magnitude will be equal to the magnitude(s) entered in the Traction dialog.
- You can increase or decrease the load magnitude of the traction at any stage, by entering a *Factor* greater than or less than 1 (eg. a *Factor* = 2 would double the load magnitude).
- You can remove the traction at any stage, by entering a *Factor* = 0. The traction will not exist at any stage in which the *Factor* = 0, and will not appear on the model.

If your traction is staged, it is a good idea to select the Stage tabs, after adding the traction, to check that the traction is applied at the correct stages, and that the magnitudes are correct. If not, then repeat Steps 1 to 5 (above), and make sure that the correct *Factors* have been applied at the correct stages.

NOTE:

- The display of tractions (ie. the red arrows) can be turned off in the [Display Options](#) dialog. Whenever you add or delete tractions, all existing tractions will be automatically displayed, even if they were turned off in **Display Options**.

Horizontal / Vertical Traction

If you want to apply HORIZONTAL or VERTICAL tractions to boundaries, *regardless of the boundary orientation*, then use the **Horizontal / Vertical** tractions option.

To apply a HORIZONTAL / VERTICAL traction, see the information in the [Normal / Shear Traction](#) topic, as the procedure for applying the traction is the same. However note the following points about the **Horizontal / Vertical** traction option:

1. Enter a *Vertical* component of zero if you only want to apply a HORIZONTAL traction.
2. Enter a *Horizontal* component of zero if you only want to apply a VERTICAL traction (this can be useful for modeling a gravity load on a non-horizontal ground surface, for example).
3. If both components are non-zero, loads at any orientation can be defined.

Angle Traction

You can use the **Angle** tractions option if your load is in terms of a magnitude and an angle, rather than NORMAL / SHEAR or HORIZONTAL / VERTICAL components.

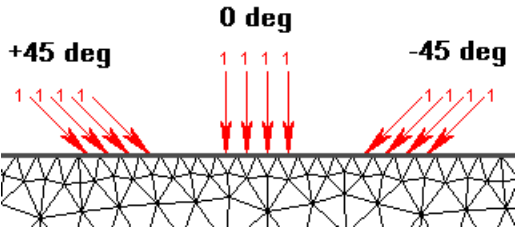
To apply an **Angle** traction, see the information in the [Normal / Shear Traction](#) topic, as the procedure for applying the traction is the same. However note the following points about the **Angle** traction option:

1. Only one load magnitude and an angle are required.
2. The angle can be specified as either *Global* or *Local* with the *Angle Definition Type* in the Angle Tractions dialog.
 - For a *Global* angle, the angle is specified with respect to the global axes, and the convention is degrees counter-clockwise from the "down" axis.
 - For a *Local* angle, the angle is specified with respect to the boundary orientation, and the convention is degrees counter-clockwise from the normal "into" the rock.
3. It is particularly important when using the **Angle** traction option, to verify that the correct traction has been applied, since errors can easily be made when deciding on the angle to use for the correct orientation. Check the direction and magnitude of the red arrows displayed after the traction is applied, and make sure it is what you intended.

Traction Examples

Note that any of the three Traction options – **Normal / Shear**, **Horizontal / Vertical** or **Angle** – can be used to define any given traction. Which option is used is a matter of convenience, or user preference. Depending on the situation, it will generally be more convenient to use one option over the others.

To illustrate this, refer to the figure below, which shows three loads of equal magnitude, at varying angles.



The input values which could be used to create these loads are listed below, for each of the three traction options.

	+ 45 deg. load	0 deg. load	- 45 deg. load
Normal / Shear	Normal = .707	Normal = 1	Normal = .707
	Shear = .707	Shear = 0	Shear = -.707
Horizontal / Vertical	Horiz = .707	Horiz = 0	Horiz = -.707
	Vertical = .707	Vertical = 1	Vertical = .707
Angle	Magnitude = 1	Magnitude = 1	Magnitude = 1
	Angle = + 45	Angle = 0	Angle = - 45

Note: because the surface in the above example is horizontal, either *Global* or *Local* angle conventions would produce the same results with the **Angle** traction option. This would NOT be the case for any other angle of surface.

Modifying Traction

To modify an existing traction (or correct a mistake), simply add a new traction to the same line segment(s). You do not have to delete the existing traction, since every time you add a traction to a line segment, the previous one is automatically erased and the new one is in effect.

If you wish to completely remove a traction, then use the **Delete Traction** option.

Deleting Traction

To delete tractions from the model:

1. Select **Delete Traction** from the toolbar, or the **Traction** sub-menu of the **Loading** menu.
2. Traction are deleted by selecting the boundary line segments on which they are applied. **Select** the line segments which have the tractions you wish to delete, and press *Enter* or right-click and select *Done Selection*. Traction on all selected line segments will be removed.

Tips

- To delete a traction on a multi-stage model, you can select a boundary line segment at any stage, and the traction will be deleted for ALL stages.
- If you want to modify an existing traction, simply add a new traction to the same line segment(s). You do not have to delete the existing traction, since every time you add a traction to a line segment, the previous one is automatically erased and the new one is in effect.

Add Nodal Force

You can add a point load to any node of the finite element mesh using the **Add Nodal Forces** option. To add a nodal force:

1. Select **Add Nodal Forces** from the **Forces** sub-menu of the **Loading** menu.
2. You will see a dialog allowing you to enter x and y force components. Enter the required forces, and note the units (MN) and sign conventions. A minus sign can be used to specify a force in the negative x or y directions.
3. In a multi-stage model, forces can be staged by selecting the *Stage Nodal Forces* checkbox, and entering a *Factor* for the force magnitude at each stage. If you do NOT stage the force on a multi-stage model, then by default the force will be applied, with the same magnitude, at all stages. See below for details.
4. Now select the node(s) on which to apply the force. You can select *any node of the mesh*, including nodes within the rockmass. Note that multiple nodes can be selected with a window. As soon as node(s) are selected, the force is applied, as you will see by the arrow symbol, and the force component magnitudes.
5. When you are finished applying the forces, press *Escape* or right-click and select *Done*.
6. Whenever you add a nodal force, all existing nodal forces will be automatically displayed. The display of nodal forces (ie. the arrow symbols) can be turned off in the **Display Options** dialog.

Staging Nodal Forces

On a multi-stage model, forces can be staged by selecting the *Stage Nodal Forces* checkbox in the Nodal Forces dialog (Step 3 above), and entering appropriate factors for the force magnitude at each stage. NOTE:

- *Factor* = 1 means that the force magnitude will be equal to the magnitude(s) entered in the Forces dialog.
- You can increase or decrease the magnitude of the force at any stage, by entering a *Factor* greater than or less than 1 (eg. a *Factor* = 2 would double the force magnitude).
- You can remove the force at any stage, by entering a *Factor* = 0. The force will not exist at any stage in which the *Factor* = 0, and will not appear on the model.

If your force is staged, it is a good idea to select the Stage tabs, after adding the force, to check that the force is applied at the correct stages, and that the magnitudes are correct. If not, then repeat Steps 1 to 5 (above), and make sure that the correct *Factors* have been applied at the correct stages.

Force Units

Although force magnitudes are entered as MN, in reality they represent line loads in MN / m, when considering the out-of-plane dimension. For a **Plane Strain** model, each force actually represents an infinite line load.

Delete Nodal Force

To delete nodal forces:

1. Select **Delete Nodal Forces** from the **Forces** sub-menu of the **Loading** menu.
2. Select the nodes on which the forces are applied, and the forces will be immediately deleted. Note that multiple nodes can be selected with a window, for the purposes of deleting nodal forces.
3. When you are finished deleting the forces, press *Escape* or right-click and select *Done*.

Tips

- To modify existing nodal forces, simply add a new nodal force to a node. The previous one will be automatically erased and the new one will be in effect.
- On a multi-stage model, nodal forces will be deleted at all stages, regardless of the stage at which the node was selected.
- All nodal forces are automatically displayed when you select **Delete Nodal Forces**. The display can be toggled on or off at any time, in the **Display Options** dialog.

Add Springs

Springs can be added to any node of the finite element mesh using the **Add Springs** option.

Springs can be used, for example, to model the effect of struts used for support.

To add springs:

1. Select **Add Springs** from the **Springs** sub-menu of the **Loading** menu.
2. In the Spring Properties dialog, enter the following information, and select *OK*:
 - *Initial Forces* in the x and y directions.
 - *Spring Stiffness* in the x and y directions
 - *Stage In / Stage Out* (staged models only), if you want to specify the stage at which springs are installed / uninstalled.
3. Now select the node(s) on which to apply the spring(s). You can select *any node of the mesh*, including nodes within the rockmass. Note that multiple nodes can be selected with a window. As soon as node(s) are selected, the spring is applied, as you will see by the spring symbol, and the force and stiffness values.
4. When you are finished applying the springs, press *Escape* or right-click and select *Done*.
5. Whenever you add springs, all existing springs will be automatically displayed. The display of springs can be turned on or off with the *Forces* checkbox in the **Display Options** dialog.

Note that if you used *Stage In / Stage Out* in the Spring Properties dialog to customize the staging of the spring installation, the springs are only displayed at stages where they exist (ie. a spring will NOT be displayed if the viewed stage is LESS THAN *Stage In* or GREATER THAN OR EQUAL TO *Stage Out*).

Delete Springs

To delete springs:

1. Select **Delete Springs** from the **Springs** sub-menu of the **Loading** menu.
2. Select the nodes on which the springs are applied, and the springs will be immediately deleted. Note that multiple nodes can be selected with a window, for the purposes of deleting springs.
3. When you are finished deleting springs, press *Escape* or right-click and select *Done*.

Tips

- To modify existing springs, simply add a new spring to a node. The previous one will be automatically erased and the new one will be in effect.
- On a multi-stage model, a spring will be deleted from a node even if the spring does not exist at the stage being viewed (ie. if the viewed stage is $< \textit{Stage In}$ or $\geq \textit{Stage Out}$, as defined in the Spring Properties dialog when the spring was **added** to the model). You should keep this in mind when using **Delete Springs** on multi-stage models with staged spring installation.
- All springs are automatically displayed when you select **Delete Springs**. The display of springs can be toggled on or off with the *Forces* checkbox in the **Display Options** dialog.

Restraints Overview

The **Restraints** menu options allow the user to apply the following boundary conditions:

- Fixed XY (ie. pinned)
- Fixed X (ie. free to move in the Y direction only)
- Fixed Y (ie. free to move in the X direction only)
- Free (ie. free in both X and Y directions)

anywhere on the model. In addition:

- A fixed, non-zero displacement can be applied as a boundary condition using the **Add Displacement** option.
- **Infinite Elements** can be used on the EXTERNAL boundary. This is set in the **Restraint Options** dialog.

Restraint Options

The **Restraint Options** dialog has two purposes:

1. The *Apply Restraints* options, allow the user to choose the most appropriate method for applying the restraint boundary conditions, when using the **Restrain X**, **Restrain Y**, **Restrain XY**, **Free**, or **Displacement** restraint options:
 - *Pick by boundary segment* – Restraints are applied to entire boundary segments (ie. only boundary segments can be selected, and boundary conditions will be applied to all nodes of the selected segments). This is the default method.
 - *Pick by boundary node* – Individual nodes on boundaries can be selected. This is useful, for example, when you want to select multiple nodes on a boundary with a selection window, without selecting internal mesh nodes.
 - *Pick by any node* – Any node of the mesh can be selected, including internal nodes.
2. The *Default External Boundary* option determines what boundary conditions will be in effect, by default, on the EXTERNAL boundary:
 - *Fixed / Pinned* – all EXTERNAL boundary nodes will be given Fixed XY conditions (this can then be customized using the **Restrain X**, **Restrain Y** or **Free** options, if necessary).
 - *Infinite Elements* – the EXTERNAL boundary conditions will be modeled using **Infinite Elements**

Infinite Elements

In the **Restraint Options** dialog, you may choose to model the EXTERNAL boundary conditions by the use of *Infinite Elements*.

Infinite Elements are an alternative to the use of fixed boundary conditions on the EXTERNAL boundary. They allow the user to solve a problem with a potentially much smaller finite element mesh, and still obtain good results.

In two dimensions, infinite elements extend to infinity in one direction, and allow the displacement to decay gradually from the external boundary toward the infinite domain – the shape functions for these elements are chosen such that the displacement at the node located at infinity is zero.

A discussion of infinite elements can be found in the book "Numerical Methods in Rock Mechanics" by G.N. Pande, G. Beer & J.R. Williams, John Wiley and Sons, Ltd., 1990.

NOTE:

- *Infinite Elements* are represented by arrows radiating from the EXTERNAL boundary nodes, when *Restraints* are displayed on the model, in the **Display Options** dialog.
-

Applying Restraints

To apply restraints to a model:

1. If necessary, use the **Restraint Options** dialog to set the selection mode for applying the restraints. (Selection mode can also be toggled in the right-click menu, see below).
2. Select the restraint option you would like to apply (**Restrain X** , **Restrain Y** , **Restrain XY** or **Free**) from the toolbar or the **Restraints** menu.
3. If the *Apply Restraints* option in the **Restraint Options** dialog is set to *Pick by boundary segment*, pick the desired boundary line segments, and when you are finished, press *Enter* or right-click and select *Done Selection*. The restraint option you are using will be applied to all nodes of the selected segments.
4. If the *Apply Restraints* option in the **Restraint Options** dialog is set to *Pick by boundary node* or *Pick by any node*, then pick the desired nodes (individually with the cursor, or pick multiple nodes with a selection window), and the restraint option you are using will be applied immediately to the selected nodes. When you are finished, press *Escape*, or right-click and select *Done*.
5. Whenever you use **Restrain X**, **Restrain Y**, **Restrain XY** or **Free**, all existing restraints (ie. roller and pin symbols) will be displayed on the model, if they are not already displayed. The display of restraints can be toggled on or off at any time in the **Display Options** dialog.

Tips

While you are applying restraints, the selection mode can be toggled in the right-click menu (ie. *Pick by boundary segment*, *Pick by boundary node*, *Pick by any node*). This allows you to change the selection mode on the fly, without having to use the **Restraint Options** dialog.

Restrain X

The **Restrain X** option is used to apply zero X-displacement boundary conditions.

The nodes will be free to move in the Y direction only. The X displacement will be fixed at zero throughout the analysis.

Nodes restrained in the X direction are indicated by vertical "roller" symbols, when *Restraints* are displayed, as configured in the [Display Options](#) dialog.

See the [Applying Restraints](#) topic for information about applying restraints.

Restrain Y

The **Restrain Y** option is used to apply zero Y-displacement boundary conditions.

The nodes will be free to move in the X direction only. The Y displacement will be fixed at zero throughout the analysis.

Nodes restrained in the Y direction are indicated by horizontal "roller" symbols, when *Restraints* are displayed, as configured in the [Display Options](#) dialog.

See the [Applying Restraints](#) topic for information about applying restraints.

Restrain XY

The **Restrain XY** option is used to apply pinned boundary conditions.

The nodes will not be free to move in the X or Y directions, and will maintain zero displacement throughout the analysis.

Nodes restrained in the X and Y directions are indicated by a triangular "pin" symbol, when *Restraints* are displayed, as configured in the [Display Options](#) dialog.

See the [Applying Restraints](#) topic for information about applying restraints.

Tips

By default, the entire EXTERNAL boundary is given a "pinned" boundary condition when the **mesh** is generated. It is not necessary for the user to use the **Restrain XY** option on the EXTERNAL boundary, unless they wish to change previously applied Fixed X, Fixed Y or Free boundary conditions.

Free

The **Free** (or **Delete Restraints**) option is used to free any nodes which are restrained in the X, Y or X and Y directions. For example, if you are modeling a surface excavation, in most cases you will want to apply a Free boundary condition to all segments of the EXTERNAL boundary representing the ground surface.

No symbol appears at nodes which are free, since they are completely free to move with no restraints.

See the [Applying Restraints](#) topic for information about applying restraints.

Add Displacement

The **Add Displacement** option allows you to specify a **non-zero**, fixed displacement as a boundary condition. To use this option:

1. Select **Add Displacement** from the **Displacements** sub-menu of the **Restraints** menu.
2. You will see a dialog allowing you to enter X and / or Y displacements. If you only wish to specify a displacement in the X or Y direction, then de-select the appropriate checkbox. Enter the displacement(s) and select **OK**.

Note: if you de-select the X or Y displacement checkbox, then the node(s) will be free to move in the direction which you de-selected, and the displacement will only be fixed in one direction.

3. The displacement boundary condition is then applied in the same manner as the **Restrain X**, **Restrain Y**, **Restrain XY** or **Free** options. See the [Applying Restraints](#) topic for information. An arrow symbol and the displacement magnitude(s) will indicate the presence of a displacement boundary condition.
4. Note that the *Apply Restraints* option in the [Restraint Options](#) dialog also applies to the **Add Displacement** option, so the displacement boundary condition can be applied to boundary segments or individual nodes, as required.
5. When you are finished, press *Escape* or right-click and select *Done*.

To remove displacement boundary conditions applied with the **Add Displacement** option, you can use the [Delete Displacements](#) option or the [Free](#) option, as both have the same effect.

Significance of a non-zero displacement boundary condition

A non-zero displacement boundary condition means that the displacement you enter is fixed throughout the analysis. For example, if you entered 0 , -.5 in the dialog in Step 2 above, the displacement at the selected node(s) would be fixed at $x = 0$, $y = -.5$ for the entire analysis. Experiment with this option to see how a non-zero displacement boundary condition affects a model.

Delete Displacement

You can use **Delete Displacement** to delete displacement boundary conditions added with the **Add Displacement** option.

1. Select **Delete Displacement** from the **Displacements** sub-menu of the **Restraints** menu.
2. Select the displacement boundary conditions to delete (you can select multiple nodes with a selection window).
3. When finished, press *Escape* or right-click and select *Done*.

Tips

You can also use the **Free** option for this purpose, it has the same effect as **Delete Displacement**.

Bolts Overview

Adding Bolts

Bolts are added to a model using either the [Add Pattern Bolts](#) option or the [Add Spot Bolts](#) option. They can also be [added from a DXF file](#) . Note:

- Bolts can only be added to a PHASE2 model, after the [mesh](#) has been generated.
- Bolts can be added to EXCAVATION, STAGE or EXTERNAL boundaries (they cannot be added on MATERIAL or JOINT boundaries).

Bolt Properties

Four bolt models are currently available in PHASE2:

- End Anchored
- Fully Bonded
- Plain Strand Cable
- Split Set / Swellex

Bolt properties are defined with the [Define Bolts](#) option, and then assigned to the bolts with the [Assign Properties](#) option.

Bolts & Staging

When adding bolts ([Add Pattern Bolts](#) or [Add Spot Bolts](#)) to a multi-stage model, it is useful to remember that they are automatically installed at the stage on which you are adding them.

- For example, if you had a three stage model, and you added bolts at stage two (ie. the Stage 2 tab was selected at the bottom of the view), they would automatically be UNINSTALLED at stage 1, and INSTALLED at stages 2 and 3.

If the bolts are not initially installed at the desired stage, the staging can be changed at any later time using the **Assign** option. A complete discussion of assigning bolt properties and staging is found in the [Assign Bolts](#) topic.

Add Pattern Bolts

The **Add Pattern Bolts** option allows the user to quickly add a uniformly spaced pattern of bolts to a boundary. To add a pattern of bolts to a model:

1. Select **Add Pattern Bolts** from the toolbar or the **Support** menu.
2. You will see the Pattern Bolts dialog, in which you may enter:
 - *Bolt Length* and *In-Plane Spacing*
 - Select the type of boundary you want to apply the pattern to (*Excavation*, *Stage* or *External*)
 - Select the *Pattern Type* (*Normal to Boundary* or *Radial*)
3. Select **OK** and follow the instructions in the prompt line, for *Normal Pattern Bolting* or *Radial Pattern Bolting*, as described below.

Normal Pattern Bolting

If your *Pattern Type* was *Normal to Boundary*, you will be prompted:

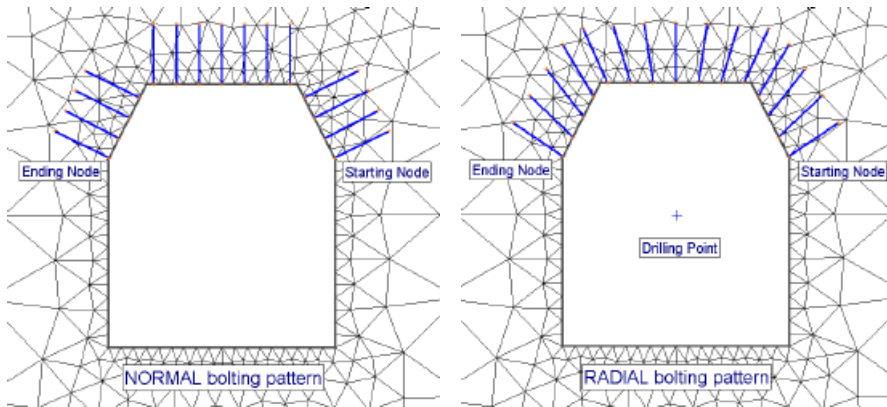
Select boundary vertex to start bolt pattern [enter=done, esc=quit]:

Select boundary vertex to end bolt pattern [enter=done, esc=quit]:

Use the mouse to select two vertices on the same boundary (excavation, stage or external), and the pattern will be generated between the two points after the second point is entered.

NOTE:

- Only boundary *vertices* can be selected as the start / end points of a bolt pattern. Boundary vertices are the points which define the geometry of your boundaries. You cannot select mesh nodes on a boundary for this purpose, unless a node is coincident with a boundary vertex.
- Be careful to note the counter-clockwise entry convention for the starting and ending vertices on EXCAVATION or EXTERNAL boundaries. If you make a mistake with the counter-clockwise convention, use the **Undo** option to undo the incorrect pattern, and then re-install the pattern, but reverse the order of the two vertices which you previously entered.
- Similarly, on a STAGE boundary, if the pattern bolts appear on the wrong side of the boundary, then use the **Undo** option, and re-install the pattern with the order of the two vertices reversed.



Radial Pattern Bolting

If your *Pattern Type* was *Radial*, you must first respond to the prompt:

Select drilling point [enter=done, esc=quit]:

Use the mouse to select a point within an excavation from which the *Radial* bolt pattern is to be drilled. You can also use the keyboard to enter an x-y location in the prompt line.

After the drilling point is established, the procedure is the same as described for the *Normal* bolt installation (ie. you must pick starting and ending vertices on the boundary).

Add Spot Bolt

The **Add Spot Bolt** option allows the user to add individual bolts to a model.

To add spot bolts:

1. Select **Add Spot Bolt** from the toolbar or the **Support** menu.
2. You will be prompted to enter two points for each bolt:

Enter vertex[esc=quit]:

Enter vertex[u=undo, esc=quit]:

Use the mouse to enter the points, or you can enter x-y coordinates in the prompt line.

3. Each spot bolt is added to the model as soon as the second point is entered.
4. You can enter as many spot bolts as you wish, by continuing to enter points.
5. When you are finished, press *Escape* or right-click and select *Done*.

Spot bolts are automatically intersected with EXCAVATION, EXTERNAL and STAGE boundaries. To ensure that the bolt actually intersects the intended boundary, you should make sure that the point on the boundary is either exactly ON the boundary, or extends slightly past it. If it extends past the boundary, then it will be automatically and correctly intersected with the boundary.

Adding Bolts with Faceplates

The following bolt models in PHASE² allow the user to attach face plates to one end of the bolt:

Plain Strand Cable

Split Set / Swellex

The face plate option is one of the bolt properties in the **Define Bolt Properties** dialog. It is important to note the following:

- When adding bolts with faceplates, PHASE² will automatically locate the face plate on the end of the bolt which intersects a boundary. Bolt face plates are indicated by small circular markers displayed at one end of a bolt.
- However, when bolts are installed between two excavations, PHASE² cannot automatically determine which end of the bolt requires the face plate. In this case, the following rule is in effect – the **FIRST** point of each bolt, must be the end with the face plate. If you make a mistake with this convention, then use the **Undo** option (or delete the bolt). Then re-enter the bolt, but reverse the order of the two points defining the bolt.

Delete Bolts

To delete bolts:

1. Select **Delete Bolts** from the toolbar or the **Support** menu.
2. Select the bolts to delete. You can **select** individual bolts by clicking on them with the left mouse button, or multiple bolts by enclosing them in a selection window.
3. When you are finished selecting, press *Enter* or right-click and select *Done Selection*, and all selected bolts will be deleted.

Tips

- If you accidentally select bolts that you don't want to delete, they can be un-selected by simply re-selecting them with the left mouse button.
- If you are deleting bolts from a staged model, you can select the bolts *at any stage*, even if they are not installed at that stage. Bolts are still visible when not installed, but are displayed in a lighter colour.

Stretch Bolts

You can use the **Stretch Bolts** option to modify existing bolts, to make them longer, shorter or change their position. To stretch bolts:

1. Select **Stretch Bolts** from the **Support** menu.
2. You will be prompted to pick the bolt vertices that you wish to stretch. You can select them individually, or select multiple bolt vertices with a selection window.
3. When you are finished selecting, press *Enter* or right-click and select *Done Selection*.
4. You will then be prompted to enter a base point and a second point defining the stretch.
5. As soon as you enter the second point, the selected bolt vertices will be moved to their new positions.

Tips

- Depending on the situation, it may be easier to **delete bolts** and re-enter new ones, than use the **Stretch Bolts** option. For example, if you wanted to change the length of pattern bolts around an excavation, it would generally be easier to delete the existing pattern and enter a new one with the desired bolt length.
- To define a "relative" stretch, enter 0 , 0 as the base point in the prompt line, and the relative x , y stretch at the second point, at step 4) above. This is an easy way of stretching by a known amount.

Bolt DXF Import

Bolts can be imported into a PHASE² model from a DXF file using the **Import DXF** option in the **File** menu. The mesh must exist before bolts can be imported. See the [Import DXF](#) topic for details.

Bolts can also be exported to a DXF file using the [Export DXF](#) option.

Liner Overview

Liners in PHASE² allow the user to model reinforcement which is applied in the form of a liner to excavation boundaries. Liners are typically used to model shotcrete. However, liners can also be used to model concrete or steel linings, for example. Composite Liners can be defined, which represent liners consisting of multiple layers of material.

Adding Liners

Liners are added to a model using the **Add Liner** option. Note:

- Liners can only be added to a PHASE2 model, after the **mesh** has been generated.
- Liners can be added to any type of boundary in PHASE2, with the exception of JOINT boundaries (ie. liners can be added to EXCAVATION, STAGE, MATERIAL or EXTERNAL boundaries).

Liner Properties

Liner properties are defined with the **Define Liners** option, and then assigned to the liners with the **Assign Properties** option.

In the most general case, Composite Liners can be defined using the **Define Composite** option. A Composite Liner can represent up to 4 different layers of material. The properties of each individual layer of a Composite Liner, are defined using the **Define Liners** option. The effect of slip between the excavation boundary and the first liner layer, may be accounted for. Composite Liner properties are assigned in the same manner as regular liners, using the **Assign Properties** option. See the **Define Composite** topic for complete details.

Add Liner

To add liners to a model:

1. Select **Add Liner** from the toolbar or the **Support** menu.
2. Select the boundary line segments that you wish to line. Liners can be added to EXCAVATION, STAGE, MATERIAL or EXTERNAL boundary line segments. (They may NOT be applied to JOINT boundaries).

NOTE: To select multiple adjoining line segments to be lined (for example, the curved roof of an excavation), a selection window is useful. All line segments *within* or *crossing* the selection window will be selected. This is important because if segments are selected individually, some may be missed.

3. When you are finished selecting, press *Enter* or right-click and select *Done Selection*.
4. All selected segments will display thick blue line segments along each finite element edge. Each of these blue line segments represents a "beam" element on the boundary.

Liners & Staging

When adding liners to a multi-stage model, it is useful to remember that a liner is automatically installed at the stage in which you are adding it.

- For example, if you had a three stage model, and you added a liner at stage two (ie. the Stage 2 tab was selected at the bottom of the view), the liner would automatically be UNINSTALLED at stage 1, and INSTALLED at stages 2 and 3.

If the liners are not initially installed at the desired stage, the staging can be changed at any later time using the **Assign** option. A complete discussion of assigning liner properties and staging is found in the [Assign Liners](#) topic.

Delete Liner

To delete liners:

1. Select **Delete Liner** from the toolbar or the **Support** menu.
2. Select the boundary line segments from which to delete liners.
3. When you are finished selecting, press *Enter* or right-click and select *Done Selection*, and all selected line segments will have the liners removed.

Tips

If you are deleting liners from a staged model, you can select the boundary line segments *at any stage*, even if the liner is not installed at that stage. Liners are still visible when not installed, but are displayed in a lighter colour.

Properties Overview

The final step in creating a PHASE² model is to:

- DEFINE all of the required material properties, and then
- ASSIGN these properties to the various elements of your model.

Properties are defined using the **Define** options in the toolbar or the **Properties** menu (ie. **Define Materials**, **Define Bolts** etc.) Properties can also be imported from other PHASE² files, using the **Import Properties** option in the **Import** sub-menu of the **File** menu.

Properties are assigned with the **Assign Properties** option in the toolbar or the **Properties** menu. The **Assign** option is also used to assign the staging sequence of excavations and support installation.

Default Properties

The first tab of each **Define** dialog (ie. **Define Materials** , **Define Bolts** , **Define Liners** , **Define Joints**) represents the DEFAULT property assignment for that option. The significance of the DEFAULT property assignment is as follows:

- All entities of your model (eg. finite elements, bolts, liners, joints) are initially assigned the DEFAULT properties when the entities are created or added to the model. The default properties are in effect until you use the **Assign** options to assign other properties.

One advantage of having a DEFAULT assignment, is that if you are only working with one type of a given entity (ie. your model only contains one material, or one bolt type, or one liner type, etc), then you can **Define** your properties using the first tab in the **Define** dialogs, without having to Assign them, since the default properties are automatically in effect.

Define Materials

To define your material (eg. rock or soil) properties, select **Define Materials** from the toolbar or the **Properties** menu.

You can define up to 40 different material types in the Define Material Properties dialog. See the following topics for details.

Name

Material Colour

Initial Element Loading

Unit Weight

Elastic Properties

Strength Properties

Staged Material Properties

NOTE: Properties can also be imported from other PHASE2 files, using the **Import Properties** option in the **Import** sub-menu of the **File** menu.

Material Name

The *Material Name* you enter in the Define Material Properties dialog will appear in the **Assign** dialog when you are assigning the material properties. Therefore use a meaningful name which you can associate with your rock type (eg. "sandstone" instead of "rock1"). It will also appear in the tabs at the top of the Define Material Properties dialog.

Material Colour

The *Material Colour* you choose in the Define Material Properties dialog will appear in the **Assign** dialog, beside the **Material Name**.

- The user can customize any material colour.
- To restore the original colour used by PHASE², select *Material Colour* and select *Default*.

Initial Element Loading

In the Define Material Properties dialog, *Initial Element Loading* allows the user to specify whether *Field Stress* and / or *Body Force* will be used for the initial element loading of a given material type. Four options are available:

- *Field Stress Only*
- *Field Stress & Body Force*
- *Body Force Only*
- *None*

Field Stress loading is derived from the values entered in the **Field Stress** dialog.

Body Force loading represents the self-weight of the elements, and is derived from the **Unit Weight** of the material entered in the Define Material Properties dialog.

If you are using a **CONSTANT** field stress, then the default *Initial Element Loading* for each material, will be *Field Stress Only*.

If you are using a **GRAVITY** field stress, then the default *Initial Element Loading* for each material, will be *Field Stress & Body Force*.

Tips

Users often are confused by the concept of Initial Element Loading, and how it relates to the Field Stress defined in the Loading menu. Here are some further comments:

The initial element loading is one of the most difficult concepts to grasp in Finite-Element (FE) modeling. Basically, in FE an element can have two initial internal loadings, initial stress and body force. Body force is just self weight. If you use just body force in a model then you will notice that a component of the displacements is due to your model settling under it's own weight. If you use materials with initial element loading set to body force only then the Field Stress defined in the Loading menu is NOT used at all. This brings us to the case of initial stress. An initial stress locks in a particular stress in an element. The stress that is locked into the element is defined in the Field Stress dialog. Think of a finite element as a sponge, applying an initial stress is like compressing the sponge. If you release confinement on an edge of the sponge it will expand in that direction. This is basically what happens when you open up an excavation in a material with an initial stress. Now how do body force and initial stress relate. They work to balance each other out. Think of an element with initial stress, then put a layer of material on top of it such that the material has enough body force to balance the initial stress. As a result the element is in equilibrium and won't expand or contract. As an example, let's think of a bucket (a rectangular region) of material represented by finite elements. The boundary conditions are free on the top surface and rollers on the left/right/bottom sides. If the material has just body force then it settles under

it's own weight and the top surface moves down. If the material has just initial stress then it expands and the top surface moves up. If you have both defined then the material is in equilibrium and there is no displacement of the top surface. To have the body force and field stress balanced you must use a gravitational Field Stress with unit weight equal to the materials unit weight and a ground surface elevation equal to the top of the bucket.

Unit Weight

The *Unit Weight* in the Define Material Properties dialog works in conjunction with the **Initial Element Loading**, and is not applicable unless *Body Force* is included in the *Initial Element Loading*. That is, *Unit Weight* is only enabled if *Initial Element Loading* is either:

- *Field Stress & Body Force* or
- *Body Force Only*

Unit Weight is disabled if *Initial Element Loading* = either *Field Stress* or *None*

If applicable, the *Unit Weight* is used to determine the *Body Force* applied to each finite element.

Unit Weight vs. Unit Weight of Overburden (Gravity Field Stress)

If you are using **GRAVITY** field stress, you should understand the distinction between the *Unit Weight* of a material entered in the Define Material Properties dialog, and the *Unit Weight of Overburden* entered in the Field Stress dialog.

- The *Unit Weight* of a material in the Define Material Properties dialog, is used to determine the Body Force applied to each finite element of a given material type.
- The *Unit Weight of Overburden* in the Field Stress dialog, is used to determine the Gravity Field Stress for all finite elements in the model.
- For a multiple material model, these two unit weights will not necessarily be the same.
- For a single material model, then the *Unit Weight* in the Define Material Properties dialog should be the same as the *Unit Weight of Overburden* in the Field Stress dialog.

Elastic Properties

Three elastic models are available for defining your rock elastic properties, in the Define Material Properties dialog:

- ISOTROPIC
- TRANSVERSELY ISOTROPIC
- ORTHOTROPIC

Note that if a given material is defined as PLASTIC (in **Strength Parameters**), then you must use ISOTROPIC elastic properties for that material. In PHASE², you cannot combine plasticity with TRANSVERSELY ISOTROPIC or ORTHOTROPIC elastic properties.

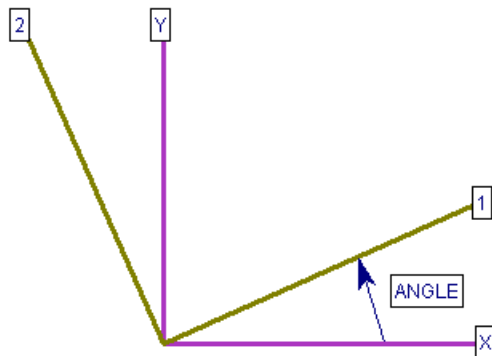
Isotropic

An ISOTROPIC material only requires Young's Modulus and Poisson's Ratio.

Transversely Isotropic

A TRANSVERSELY ISOTROPIC material requires the following parameters:

- E1 and E2 are the in-plane elastic moduli. (Ez is the out-of-plane elastic modulus, and is equal to E1)
- The required Poisson's ratios are ν_{12} and $\nu_{13} = \nu_{31} = \nu$. Note that ν_{ij} implies strain in the j-direction due to strain in the i-direction. Also note that $E_i / E_j = \nu_{ij} / \nu_{ji}$. Therefore, the plane 1-2 is the isotropic plane
- G12 is the in-plane shear modulus
- The ANGLE is measured between the x-direction and the direction of E1, as shown in the figure below.



Definition of ANGLE for TRANSVERSELY ISOTROPIC or ORTHOTROPIC materials

Orthotropic

The required elastic parameters are now E1, E2 and Ez, where the out-of-plane elastic modulus Ez is no longer equal to E1. Three separate Poisson's ratios are required, ν_{12} , ν_{1z} and ν_{2z} . The in-plane shear modulus G12 and the ANGLE are the same as for a TRANSVERSELY ISOTROPIC material.

Strength Parameters

In the Define Material Properties dialog, three failure criteria are available in PHASE2 for defining the strength of your rockmass:

- MOHR-COULOMB
- HOEK-BROWN
- DRUCKER-PRAGER

Remember that the strength parameters you enter should reflect the strength of the rockmass, and not necessarily the strength of an intact sample.

Elastic Material

The purpose of defining failure criterion parameters for an ELASTIC material, is to allow the calculation and plotting of strength factors. Even though an ELASTIC material in PHASE2 does not "fail", the failure criterion allows a degree of overstress to be calculated.

Plastic Material

For PLASTIC materials, the strength parameters you enter will be used in the analysis if yielding occurs. This is unlike ELASTIC materials, where the strength parameters are only used to obtain values of strength factor, but do not affect the analysis results.

If you define a material to be PLASTIC then you must define residual strength parameters and a dilation.

- If the residual strength parameters are equal to the peak parameters, then you are defining an "ideally" elastic-plastic material.
- The dilation is a measure of the increase in volume of the material when sheared.

NOTE that if you define a given material as PLASTIC, then you are restricted to ISOTROPIC elastic properties for that material. You cannot combine plasticity with TRANSVERSELY ISOTROPIC or ORTHOTROPIC elastic properties.

Staged Material Properties

In PHASE2, the properties of a material can be modified at different stages of a multi-stage model, by using the *Stage Material Properties* option in the Define Material Properties dialog. This allows the user to simulate, for example:

- an increase or decrease in material stiffness
- an increase or decrease in material strength

as staging progresses. Any of the material parameters entered in the Define Material Properties dialog, can be increased or decreased by user-defined factors at different stages, as described below. To stage material properties:

1. In the Define Material Properties dialog, select the *Apply Factors* checkbox in the *Stage Material Properties* area at the bottom of the dialog.
2. The *Define Factors* button will now be enabled. Select this button, and the Stage Material Properties dialog will appear.
3. In the Stage Material Properties dialog, select the *Add Stage* button, until the number of rows in the dialog, is equal to the number of stages at which you need to modify the material properties. For example:
 1. if you need to modify material properties at 3 different stages, then select *Add Stage* TWICE, so that a total of 3 rows appears in the dialog.
4. In each row, enter the Stage number at which the properties are to be modified.
5. In each row, enter the required Factor(s) which will be used to modify the indicated material parameter(s). NOTE:
 - the Factors are multiplied by the parameter values in the Define Material Properties dialog, in order to obtain the actual value to be used for a parameter, at a given stage of the analysis
 - for example, if you want to increase Young's Modulus by 20%, enter a Factor of 1.2 for Young's Modulus, at the desired stage(s)
 - leave the Factor = 1 for any parameter which does not change value
6. It is important to note that you only need to define the stage and factor information, for the stages at which you want to modify properties. You DO NOT need to define this information at stages where the properties do not change.
7. If you accidentally add too many rows with the *Add Stage* button, click the mouse in a row and select the *Delete Stage* button, to delete the row.
8. When you are finished entering all stage numbers and factors, select *OK* in the Stage Material Properties dialog. Select *OK* in the Define Material Properties dialog when you are finished defining all materials.

When the analysis is run, PHASE2 will use the factors in the Stage Material Properties dialog, to determine the material properties to be used at each stage of the analysis, for a given material.

Define Groundwater Properties

The **Define Groundwater Properties** option, in the toolbar or the **Properties** menu, is used to specify the groundwater information for each material in the **Define Material Properties** dialog. This is applicable only if your PHASE2 model incorporates groundwater (ie. you have defined Piezometric Lines or a Water Pressure Grid, in order to include pore pressure in the analysis). NOTE:

- The material names and colours which appear in the **Define Groundwater Properties** dialog, are those which are specified in the **Define Material Properties** dialog.
- The options available in the **Define Groundwater Properties** dialog, correspond to the *Groundwater Method* specified in the **Project Settings** dialog:
 - If the *Groundwater Method* = *Piezometric Lines*, then you will be able to assign Piezometric Lines to each material.
 - If the *Groundwater Method* = *Water Pressure Grid*, then you will simply be able to turn the effect of the grid on or off, for each material.

Piezo to Use

Piezometric Lines are created with the with the **Add Piezometric Line** option. Each Piezometric Line has an ID number associated with it. These ID numbers are automatically created by PHASE2, when the Piezometric Line is added to the model.

The *Piezo to Use* option in the Define Groundwater Properties dialog, will contain a list of ID numbers of all Piezometric Lines which currently exist in the model.

- By default, the *Piezo to Use* option is initially set to *None*. This means that NO Piezometric Line will be assigned to that material, and pore pressure at all points within such materials, will be zero.
- In order to assign a Piezometric Line to a given material, simply select the ID number of the desired Piezometric Line, from the *Piezo to Use* drop-down list. The selected Piezometric Line will then be used for the pore pressure calculations for that material.

NOTE: the *Piezo to Use* option is applicable if you want to use ONE Piezometric Line for the pore pressure calculation for a given material, AT ALL STAGES OF THE MODEL (for staged models). If you require different piezo lines to be applied at different stages, for a given material, then use the *Stage Piezo Lines* option described below.

Stage Piezo Lines

If your model consists of multiple stages, and you require different piezo lines to be applied at different stages, for a given material, then use the *Stage Piezo Lines* option. This allows you to simulate, for example, the draw down of a water table at different stages.

To assign the staging of Piezometric Lines:

1. In the Define Groundwater Properties dialog, select the *Stage Piezo Lines* checkbox.
2. You will notice that the *Piezo to Use* option becomes disabled, and the area below the *Stage Piezo Lines* checkbox becomes enabled.
3. Now select the *Add Stage* button, until the number of rows is equal to the number of stages at which you need to specify a Piezometric Line. For example:
 - if you need to assign 3 Piezometric Lines at 3 different stages, then select *Add Stage* TWICE, so that a total of 3 rows appears in the data entry area below the *Stage Piezo Lines* checkbox.

NOTE: if the *Add Stage* button is disabled, this means that you have not set the *Number of Stages* in **Project Settings**. In this case, go to the **Project Settings** dialog, and enter the correct *Number of Stages* (greater than 1) for the model.

4. In each row, enter the *Stage* number at which the Piezometric Lines are to be assigned.
5. In each row, select the Piezo ID number representing the Piezometric Line that you wish to assign at the specified stage.
6. It is important to note that you only need to define the *Stage* and *Piezo ID* information, for the stages at which the Piezometric Line will change. You DO NOT need to define this information at stages where the Piezometric Line does not change.
7. If you accidentally add too many rows with the *Add Stage* button, click the mouse in a row and select the *Delete Stage* button, to delete the row.
8. When you are finished entering the *Stage* and *Piezo ID* numbers, if your model consists of multiple materials with the same staging of Piezo Lines, you can use the *Copy to All* button, to automatically copy the Stage / Piezo ID information, to all materials in the dialog (see below).
9. When the analysis is run, PHASE2 will use the *Stage* and *Piezo ID* information in the Define Groundwater Properties dialog, to determine the correct Piezometric Line to be used at each stage of the analysis, for each material.

Copy To All

The *Copy To All* option allows you to define the groundwater information for one material, and then automatically copy the same information, to all materials in the dialog. This is useful when all or most of the materials in a model, have the same groundwater conditions (for example, they all use the same water table).

The *Copy to All* option is particularly useful, if you are using the *Stage Piezo Lines* option, and your model has multiple materials and piezo lines. The staging information can then be defined only once, and automatically copied to all materials. For example:

- Define your Piezometric Line staging information for ONE material.
- Select *Copy to All*, and the same information will be automatically entered for ALL materials in your model.

-
- NOTE that the *Copy to All* option also copies the value of the Hu coefficient for the current material, to all materials.

Of course, this assumes that you wish to define the same groundwater conditions, for all materials in your model. If the same information applies to MOST but not all of your materials, then you can still use the *Copy to All* option, and then individually edit any materials which require different information.

Hu Value

The *Hu Value* is simply a factor by which the vertical distance from a point in the model, to a Piezometric Line is multiplied to obtain the pressure head. The *Hu Value* may range between 0 and 1. $Hu = 1$ would indicate hydrostatic conditions. $Hu = 0$ would indicate a dry material, and intermediate values are used to simulate head loss due to seepage. See the [Pore Pressure Calculation Using Piezometric Lines](#) topic for details.

Use Grid Values

If the *Groundwater Method* in **Project Settings** is set to one of the 3 water pressure grid options – *Grid (Total Head)*, *Grid (Pressure Head)* or *Grid (Pore Pressure)*, then the *Use Grid Values* option will be enabled in the Define Groundwater Properties dialog.

The *Use Grid Values* option simply allows the user to toggle the effect of a Water Pressure Grid, ON or OFF for a given material.

- If the *Use Grid Values* checkbox is selected, then pore pressure will be calculated from the Water Pressure Grid.
- If the *Use Grid Values* checkbox is NOT selected, then pore pressure for that material will be set to ZERO throughout the analysis, and the Water Pressure Grid will have no effect, for that material.

NOTE that currently in PHASE2 it is NOT possible to stage Water Pressure Grids, in the same manner as Piezometric Lines. Only ONE Water Pressure Grid may be defined for a model, and used at all stages. This may be improved in a future PHASE2 upgrade, to allow multiple water pressure grids to be read into a model, and applied at different stages.

Pore Pressure Calculation

Piezometric Lines are used for pore pressure calculation if the *Groundwater Method* in **Project Settings** is set to *Piezometric Lines*.

Pore pressure is calculated for a Piezometric Line using:

- the *Pore Fluid Unit Weight* (entered in the **Project Settings** dialog)
- the vertical distance from a point to a Piezometric Line and
- the *Hu Value* for the material (entered in the **Define Groundwater Properties** dialog), as follows:

$$\text{Pore Pressure} = \text{Pore Fluid Unit Weight} * \text{Vertical Distance to Piezometric Line} * \text{Hu Value}$$

If the distance is negative, (ie. Piezometric Line is below a given point) then the pore pressure is set to zero at that point.

What is the Hu Value?

The *Hu Value*, as defined in PHASE2, is simply a factor between 0 and 1, by which the vertical distance from a point in a material to a Piezometric Line is multiplied to obtain the pressure head. An *Hu Value* can be defined for each material in the **Define Groundwater Properties** dialog. $Hu = 1$ would indicate hydrostatic conditions. $Hu = 0$ would indicate a dry material, and intermediate values are used to simulate head loss due to seepage.

Define Bolts

To define your bolt properties, select **Define Bolts** from the toolbar or the **Properties** menu.

You can define up to 20 different bolt types in the Define Bolt Properties dialog.

There are currently four different bolt models available in PHASE2:

End Anchored

Fully Bonded

Plain Strand Cable

Split Sets / Swellex

End Anchored Bolts

The *End Anchored* bolt model in PHASE2 works as follows:

- Each bolt behaves as a single element. Interaction with the finite element mesh is through the endpoints of the bolt only.

Failure Mechanism

- An *End Anchored* bolt fails in tension, when the peak capacity is exceeded.
- An *End Anchored* bolt has no residual capacity after failure, the entire bolt is considered to have failed.

Fully Bonded Bolts

The *Fully Bonded* bolt model in PHASE2 works as follows:

- *Fully Bonded* bolts are divided into "bolt elements" according to where the bolts cross the finite elements. These bolt elements act independently of each other – neighbouring *Fully Bonded* bolt elements do not influence each other directly, but only indirectly through their effect on the rock mass.
- The strength and stiffness of the rock/grout and bolt/grout interfaces are NOT accounted for – the bolt is assumed to remain fully bonded to the rock, and no interface effects are considered.

Failure Mechanism

- Failure of a *Fully Bonded* bolt element occurs in tension, when the axial force on the bolt element exceeds the axial capacity. Failure of an element of a *Fully Bonded* bolt does not necessarily imply failure of adjacent elements on the same bolt. If you view yielded bolt elements in the PHASE2 INTERPRETER, you will see that *Fully Bonded* bolts can show yielding in some elements and not in others.
- Unlike an *End Anchored* bolt, a *Fully Bonded* bolt element can still have a residual capacity (ie. it can still carry load) after the peak capacity is exceeded.

Plain Strand Cable

The *Plain Strand Cable* model in PHASE2 works as follows:

- The entire bolt behaves as a single element. Although for purposes of the algorithm, the bolt is discretized according to the intersections with the finite elements, the behaviour of each segment of the bolt has a direct effect on adjacent segments.
- The stiffness of the grout, and the strength and stiffness of the bolt/grout interface is taken into account.
- The effect of confining stress on the bolt due to excavation induced stress change is taken into account.

Failure Mechanisms

- At present, the only failure mechanism of a *Plain Strand Cable* bolt is by tensile rupture of the cable.
- Failure of the cable/grout interface also occurs, but it is not a failure mechanism as such, since this interface is always assumed to be in a plastic state as the rock moves. The amount of relative slip at this interface, and the stiffness of the interface, determines how much shear force is generated at the cable.
- Failure of the grout/rock interface is NOT currently considered, but may be in future.

The following properties define a *Plain Strand Cable* bolt in PHASE2.

Borehole Diameter

The diameter of the borehole used to install the bolts.

Cable Diameter

The diameter of the cable.

Cable Modulus

The effective Young's Modulus of the cable.

Cable Peak

The peak tensile capacity of the cable.

Water-Cement Ratio

The Water to Cement ratio of the grout used to install the cable.

Out-of-Plane Spacing

The spacing between bolts in the out-of-plane direction (ie. normal to the analysis plane).

Face Plates

The *Face Plates* option allows the user to simulate the effect of face plates used on bolts.

- If the *Face Plates* checkbox is selected, then the first vertex of each bolt will be "fixed" to the rock mass, allowing the bolt to develop load starting at the face plate.
- If the *Face Plates* checkbox is NOT selected, then the load at the beginning of the bolt will be zero.

Add Pull Out Force

The *Add Pull Out Force* option allows the user to simulate the effect of a pull-out test on a bolt, by applying a tensile force to the beginning of the bolt. To use this option, select the *Add Pull Out Force* checkbox, and enter the desired value of pull out force. The force will be applied as a tensile load to the first vertex of the bolt.

Constant Shear Stiffness

In the *Plain Strand Cable* model in PHASE2, the shear stress vs. shear displacement graph for a bolt, will in general be non-linear. Therefore the shear stiffness will be variable, and will change according to the shear stress. The user may impose a Constant Shear Stiffness, by selecting the Constant Shear Stiffness checkbox, and entering a value. The shear stiffness will then be independent of the shear stress on the bolt. Note that the units of Shear Stiffness are FORCE / LENGTH / LENGTH.

References

For information about the development of the *Plain Strand Cable* model, see the following references.

Moosavi, M. (1997), "Load distribution along fully grouted cable bolts based on constitutive models obtained from modified Hoek cells". Ph.D. Thesis, Queen's University, Kingston, Ontario, Canada.

Moosavi, M. , Bawden, W.F. and Hyett, A.J., (1996), "A comprehensive laboratory test programme to study the behaviour of modified geometry cable bolt support", Rock mechanics, tools and technics, Proceedings of the 2nd North American Rock Mechanics Symposium, Montreal.

Hyett, A.J., Moosavi, M and Bawden, W.F., (1996), "Load distribution along fully grouted bolts, with emphasis on cable reinforcement", Int. J. of Num. & Anal. Meth. in Geomech. Vol 20, pp 517-544.

Hyett, A.J, Bawden, W.F., MacSporran, G.R., and Moosavi M., (1995), "A constitutive law for bond failure of fully grouted cable bolts using a modified Hoek cell", Intl. J. Rock Mech. and Mining Sci. & Geomechanics Abstracts, Vol 32, No.1, pp 11-36.

Swellex / Split Sets

The *Swellex / Split Set* bolt model (also called Frictional bolts or Shear bolts) in PHASE2 works as follows:

- The entire bolt behaves as a single element. For purposes of the algorithm, the bolt is discretized according to the intersections with the finite elements, however, the behaviour of each segment of the bolt has a direct effect on adjacent segments.
- The stiffness of the bolt / rock interface is taken into account.
- Unlike the **Plain Strand Cable** model, the confining stress level in the rock does NOT affect the shear strength or stiffness of the bolt.

Failure Mechanisms

A *Swellex / Split Set* bolt in PHASE2 can fail in two modes:

- In tension, if the *Tensile Capacity* is exceeded.
- In shear, if the *Shear Capacity* is exceeded.

The following properties define a *Swellex / Split Set* bolt in PHASE2.

Tensile Capacity

This is the maximum tensile (axial) force which the bolt is capable of sustaining.

NOTE:

- The *Tensile Capacity* option is only enabled, if the *Bolt Model* = *Plastic* (see below). If the *Bolt Model* = *Elastic*, then by definition the bolt behaves in an elastic manner only, and tensile (and shear) capacity (strength) is not considered, and is not applicable.
 - *Tensile Capacity* is equal to the (Yield Stress of the steel) * (*Tributary Area*)
-

Tributary Area

Swellex / Split Set bolts are hollow. The *Tributary Area* is the effective cross-sectional area of the bolt (ie. the cross-sectional area of the annulus of steel which makes up the bolt). It is NOT the total cross-sectional area of the bolt (ie. it does NOT include the hollow area of the bolt). The *Tributary Area* is used to calculate the axial stiffness of the bolt (see below).

Bolt Modulus

The *Bolt Modulus* is the Young's Modulus of the bolt steel.

NOTE:

- The *Bolt Modulus*, *Tributary Area* and bolt length determine the axial stiffness of a bolt, ie. $\text{Axial Stiffness} = (\text{Bolt Modulus}) * (\text{Tributary Area}) / (\text{bolt length})$
-

Shear Capacity

This is the maximum shear force capacity of the bolt / rock interface. The *Shear Capacity* is expressed as a FORCE per unit BOLT LENGTH.

NOTE:

- The *Shear Capacity* option is only enabled, if the *Bolt Model = Plastic* (see below). If the *Bolt Model = Elastic*, then by definition the bolt behaves in an elastic manner only, and shear (and tensile) capacity (strength) is not considered, and is not applicable.
 - *Shear Capacity* can be determined from empirical methods such as pull-out tests.
-

Shear Stiffness

This is the shear stiffness of the bolt / rock interface. Note that the units of Shear Stiffness are FORCE / LENGTH / LENGTH. The Shear Stiffness can be thought of as follows:

-
- On a shear force vs. shear displacement graph, the Shear Stiffness is equal to the slope of the ELASTIC portion of the graph. Remember shear force is expressed as a FORCE / LENGTH therefore the units of the Shear Stiffness are FORCE / LENGTH / LENGTH.
-

Out-of-Plane Spacing

This is the spacing between bolts in the out-of-plane direction (ie. normal to the analysis plane).

Bolt Model

- If the *Bolt Model = Elastic*, then the tensile and shear capacity of the bolt is not considered. The *Tensile Capacity* and *Shear Capacity* options will be disabled. The bolt will behave in an elastic manner only, and the forces in the bolt will be determined by the Axial and Shear Stiffness.
- If the *Bolt Model = Plastic*, then the tensile and shear capacity of the bolt is considered, and the *Tensile Capacity* and *Shear Capacity* options will be enabled. If the Tensile Capacity or Shear Capacity of the bolt is reached, then the bolt will behave in a perfectly plastic manner, and the axial or shear force in the bolt will not vary with further axial or shear displacement.

Face Plates

The *Face Plates* option allows the user to simulate the effect of face plates used on bolts.

- If the *Face Plates* checkbox is selected, then the first vertex of each bolt will be "fixed" to the rock mass, allowing the bolt to develop load starting at the face plate.
- If the *Face Plates* checkbox is NOT selected, then the load at the beginning of the bolt will be zero.

Add Pull Out Force

The *Add Pull Out Force* option allows the user to simulate the effect of a pull-out test on a bolt, by applying a tensile force to the beginning of the bolt. To use this option, select the *Add Pull Out Force* checkbox, and enter the desired value of pull out force. The force will be applied as a tensile load to the first vertex of the bolt.

Add Pre-Tensioning Force

If the *Add Pre-Tensioning Force* checkbox is selected, this allows the user to include a pre-tensioning force in the analysis. A pre-tensioning force is implemented as an initial axial force assigned to each bolt element of a bolt. (Bolt elements are determined by the intersections of the bolt with the finite element mesh).

Define Liners

To define liner properties, select **Define Liners** from the toolbar or the **Properties** menu.

You can define up to 20 different liner types in the Define Liner Properties dialog.

Liners in PHASE² allow the user to model reinforcement which is applied in the form of a liner to excavation boundaries. Liners are typically used to model shotcrete. However, liners can also be used to model concrete or steel linings, for example.

NOTE: Composite Liners can be defined, which represent liners consisting of multiple layers of material. The properties of individual layers of Composite Liners, are defined with the **Define Liners** option. See the [Define Composite Liners](#) topic for details.

Beam Formulation

Liners are comprised of "beam" elements, corresponding to the edges of finite elements. Two different Beam Formulations are available in PHASE2 for modeling liners:

- Timoshenko (allows for transverse shear deformation effects)
- Bernoulli (the classical Euler-Bernoulli formulation, which takes no account of transverse shear deformation)

The Timoshenko Beam Formulation is recommended if you are using finite elements with [midside nodes](#) (ie. 6-noded triangles or 8-noded quadrilaterals), since in this case PHASE2 will automatically use 3-noded Timoshenko beam elements, resulting in displacement compatibility between the finite elements and the beam elements. Bernoulli beam elements are always 2-noded, and give less accurate results when used with 6-noded triangles or 8-noded quadrilaterals.

Liner Elastic Properties

Liners are assumed to have isotropic elastic properties, therefore only a single Young's modulus and a single Poisson's ratio can be entered.

Liner Strength Parameters

If you define the liner Material Type to be ELASTIC, then strength parameters are not applicable, since there is currently no facility for visualizing a "strength factor" for elastic liners in PHASE2.

If you define the liner Material Type to be PLASTIC, then you may enter peak and residual compressive and tensile strength parameters, which will be used in the PHASE2 analysis. Liners defined as PLASTIC will be able to yield, if the peak strength is reached. The

plasticity calculation uses the "layering" method presented in reference 1 (see below), in which the cross-section of the beam is partitioned into layers. If the stress in a given layer exceeds the peak strength (compressive or tensile), the layer yields.

Stage Liner Properties

In PHASE2, the properties of a liner can be modified at different stages of a multi-stage model, by using the *Stage Liner Properties* option in the Define Liner Properties dialog. This could be used, for example, to model the increase in strength and stiffness of shotcrete or concrete liners after initial placement.

Any of the parameters entered in the Define Liner Properties dialog, can be increased or decreased by user-defined factors at different stages. For details about staging liner properties, see the [Stage Material Properties](#) topic, as the procedure for staging properties is the same.

References

Further information about the liner model used in PHASE2, can be found in the following references.

1. D.R.J. OWEN and E. HINTON (1986), Finite elements in Plasticity theory and practice, Department of Civil Engineering, University College of Swansea, UK, Pineridge Press Limited, Swansea, UK. (Chapter 5.5)
2. I.M. SMITH and D.V. GRIFFITHS, Programming the finite element method, Second ed., University of Manchester, UK, John Wiley and Sons, New York. (Chapter 4).

Define Joints

To define joint properties, select **Define Joints** from the toolbar or the **Properties** menu.

You can define up to 20 different joint types in the Define Joint Properties dialog.

Normal & Shear Joint Stiffness

Each joint element has a normal and a shear spring stiffness which relate the normal and shear stress on the element to the normal and shear displacement. The stiffness of the joint can be derived from the thickness and modulus of the infilling material (if these quantities are known) by the following equations:

$$K_n = E_o / h$$

$$K_s = G_o / h$$

Where:

K_n = joint normal stiffness

K_s = joint shear stiffness

E_o = Young's modulus of the infilling material

G_o = shear modulus of the infilling material

h = joint thickness or aperture

Slip Criterion

If the *Slip Criterion* is *None*, then joints will behave elastically, according to the normal and shear joint stiffness, and cannot slip.

If Slip Criterion is *Mohr-Coulomb* or *Barton-Bandis*, then local joint slip can occur if the shear stress on a joint element exceeds the shear strength as defined by the *Slip Criterion* parameters.

Mohr-Coulomb Slip Criterion parameters are:

- Cohesion
- Friction Angle
- Tensile Strength

Barton-Bandis Slip Criterion parameters are:

- JCS
- JRC

-
- Residual Friction Angle

Initial Joint Deformation

Initial Joint Deformation refers to how the joint behaves in regard to the far field stresses. If the *Initial Joint Deformation* is ON, then the joint will deform based on *both* the far field stresses *and* the induced stresses due to any excavations. As a result, the stress field in the vicinity of the joint will be altered from the initial far field distribution. This will be evident if you run an analysis with a **joint only** (ie. no excavations).

If *Initial Joint Deformation* is OFF, then a single joint, with no excavation, has no effect at all (try running this). With no Initial Joint Deformation, only the induced stresses from the presence of an excavation or an external load, can cause joint deformation.

Stage Joint Properties

The properties of a joint can be modified at different stages of a multi-stage model, by using the *Stage Joint Properties* option in the Define Joint Properties dialog. Any of the parameters entered in the Define Joint Properties dialog, can be increased or decreased by user-defined factors at different stages. For details about staging joint properties, see the [Stage Material Properties](#) topic, as the procedure for staging properties is the same.

Allow slip starting from stage

The *Allow slip starting from stage #* option allows the user to specify that joint slip (ie. inelastic yield) cannot occur before a specified stage. To use this option, select the checkbox, and enter the stage at which the joint is allowed to slip. Note that elastic displacements may still occur before this stage.

Define Composite Liner

A Composite Liner in PHASE2, is a liner which may consist of multiple layers of material. The different layers of a Composite Liner may have different material properties, and may be applied at different stages.

- A Composite Liner may consist of up to 4 different layers. The properties of each layer of a Composite Liner, are defined using the **Define Liner Properties** dialog.
- A joint may also be included in the Composite Liner, which will exist between the rockmass and the FIRST liner layer. The properties of the joint, are defined using the **Define Joint Properties** dialog.

The Composite Liner Properties dialog allows the user to define a Composite Liner, by selecting individual liners (defined in the Define Liner Properties dialog), and a joint (defined in the Define Joint Properties dialog), in the desired sequence and combination. To define a Composite Liner:

1. You must first define the properties of the individual layers of the Composite Liner, using the **Define Liners** option.
2. If you are including a joint in the Composite Liner, then you must also define the properties of the joint, using the **Define Joints** option.
3. Now select the **Define Composite** option from the toolbar or the **Properties** menu.
4. In the Define Composite Properties dialog, select the *Composite Type* that you would like to define.
 - if you are NOT including a joint, then you can define a Composite Liner with 2, 3 or 4 layers (ie. *Composite Type = 2 liners, 3 liners or 4 liners*)
 - if you wish to include a joint, then you can define a Composite Liner with 1, 2, 3 or 4 layers (ie. *Composite Type = 1 liner (with slip), 2 liners (with slip), 3 liners (with slip) or 4 liners (with slip)*).
5. Now use the drop-down lists, to select the desired sequence of liners which will be used, and also the joint, if included.

NOTE: The liner (and joint) names which appear in the drop-down lists, are the liner (and joint) names, which you entered in the Define Liner Properties and Define Joint Properties dialogs, when you defined the liner (and joint) properties. Therefore it is important that you use meaningful names for the liners, so that you can easily identify the correct liners that you wish to use in the Define Composite Properties dialog.

6. If your Composite Liner consists of 2, 3 or 4 liners, then you can now assign the stage of application of liners 2, 3 and 4, relative to the stage of application of the FIRST liner layer, using the drop-down list boxes at the right of the dialog. NOTE:
 - The stage of application of the FIRST liner layer, is determined with the **Assign Properties** option (see below).
7. When you are finished defining the Composite Liner, select **OK**.

Composite Liners are assigned their initial stage application, in exactly the same manner as regular (single) liners, using the **Assign Properties** option. This determines the stage of application of the FIRST layer of the Composite Liner. The stage of application of all subsequent layers of the Composite Liner, is determined in the Define Composite Properties dialog, as described above.

Assign Properties Overview

After you Define the material and support properties, you must Assign these properties to the various elements of your model. **Default properties** are initially in effect, but in general the user will have to make use of the **Assign Properties** option.

When you select **Assign Properties**, you will see the Assign Properties dialog. At the bottom of this dialog is a drop-down list, which allows you to select which type of entity you wish to assign.

- This list will always include Materials, which is the default selection.
- All other entity types (ie. Bolts, Liners, Composite Liners, Joints) will only appear in the list, if these entities are currently part of your model.

To assign properties or the staging sequence for a given type of entity, you must first select the desired entity from this list at the bottom of the Assign dialog. See the following topics for details:

[Assign Materials](#)

[Assign Bolts](#)

[Assign Liners](#)

[Assign Joints](#)

[Assign Composite](#)

NOTE:

- The finite element mesh must exist before the **Assign Properties** option is enabled.
- The **Assign Properties** 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, to minimize/maximize it without closing it. The **X** button will close the dialog.

Assign Materials

The **Assign Materials** option is used to assign:

- MATERIAL properties to the different material zone(s) of your model
- The staging sequence of the EXCAVATIONS for STAGED models.

To assign materials:

1. In the **Assign Properties** dialog, make sure that *Materials* is selected in the drop-down list at the bottom of the dialog.
2. In the dialog, notice that the material names and colours are those which you defined in the **Define Materials** option. Use the mouse to select a material you would like to assign.
3. Now as you move the cursor over the model, notice that a small cross-hair appears at the end of the cursor arrow. This indicates that you are in the mode to assign materials.
4. Now simply locate the cross-hair cursor inside a material zone in your model, and click the left mouse button. You will see that all finite elements in the zone will change to the colour of the assigned material.
5. Repeat step 4, to assign the same material to other zones of the model if necessary. You may need to zoom or pan the view, in order to accurately select regions to assign.
6. If your model has multiple material types, repeat steps 2 to 5 to assign the desired materials to all the zones of the model.
7. If you are finished with the Assign dialog, you can close it by selecting the X in the dialog, or by pressing *Escape*.

NOTE that a material "zone" is any closed region defined by the intersection of any of the model boundaries (ie. EXCAVATION, EXTERNAL, MATERIAL, STAGE or JOINT boundaries). If a region is not "closed" you will discover this when you are assigning materials (the assignment will "leak" around the edges of any boundaries which do not intersect. If this occurs, you will have to reset the mesh, and edit the necessary boundaries, so that they intersect).

To assign the staging of excavations:

1. In the **Assign Properties** dialog, make sure that *Materials* is selected in the drop-down list at the bottom of the dialog.
2. Select the *Excavate* button at the bottom of the Assign dialog.
3. Make sure the stage you are viewing is the stage at which you want to excavate. Use the Stage tabs to view the desired stage.

-
4. Click inside the region you want to excavate, and notice that the finite elements "disappear" in that region. This indicates that the region is excavated (see note below for more information).
 5. Repeat steps 3 and 4 to excavate your model in the desired sequence.

NOTE: As of PHASE2 version 5, any closed region of the model, formed by the intersection of any of the model boundaries, can be excavated. In earlier versions of PHASE2, only the material within EXCAVATION boundaries could be excavated. This restriction has now been removed, and gives the user more freedom to define a model. For example, closed regions formed by the intersection of STAGE boundaries and the EXTERNAL boundary, can now be excavated.

To assign backfill:

You can backfill an excavated region at a later stage, by assigning another material to the region at any stage after the excavated stage – you will see that the finite elements reappear in the excavated region in this case.

Tips

- For a multi-stage model, material assignment and excavation applies to all stages AFTER and INCLUDING the stage at which the assignment was made. Stages BEFORE the assigned stage are NOT affected (ie. the material type will remain unchanged). Therefore, in most cases you should do all of your material assignments in the FIRST stage.
- Any material assigned after the first stage, should, practically speaking, represent the backfill of an excavated region. However, since there is no restriction in PHASE2, you could actually change material properties from stage to stage.
- The maximum number of times that a given region would normally be assigned properties is THREE – the initial material assignment, the EXCAVATED assignment, and a final material assignment, representing backfill.

Assign Bolts

The **Assign Bolts** option is used to assign the bolt type(s) and installation sequence to the bolts in your model.

To assign a bolt type:

1. In the **Assign Properties** dialog, make sure that *Bolts* is selected in the drop-down list at the bottom of the dialog.
2. In the dialog, notice that the bolt names and colours are those which you defined in the **Define Bolts** option. Use the mouse to select a bolt type you would like to assign.
3. Use the mouse to select the desired bolts on the model (you can select multiple bolts with a selection window). When selected, bolts are highlighted by a dotted line. TIP – if you accidentally select bolts that you did not intend to select, they can be **UNSELECTED**, by clicking on them again with the left mouse button.
4. When all desired bolts are selected, press *Enter* or right-click and select *Done Selection*. All selected bolts will be assigned the bolt type you chose, as indicated by their colour.
5. If your model has multiple bolt types, repeat steps 2 to 4 to assign the correct bolt types to all the bolts in the model.
6. If you are finished with the Assign dialog, you can close it by selecting the X in the dialog, or by pressing *Escape*.

NOTE:

- Bolts are initially assigned the properties of the first bolt type (ie. the first window in the Define Bolts dialog) when you added the bolts to the model. If you are only using one bolt type in your model, then you do not necessarily need to assign a bolt type (just make sure you define your bolt properties using the first tab of the Define Bolts dialog).
- For a multi-stage model, the assigned bolt type applies to all stages – ie. the bolt type (of a given bolt) cannot change at different stages.

To assign the staging of bolts:

1. In the **Assign Properties** dialog, make sure that *Bolts* is selected in the drop-down list at the bottom of the dialog.
2. Select the *Install* button at the bottom of the Assign dialog.
3. Make sure the stage you are viewing is the stage at which you want to install the bolt(s). Use the Stage tabs to view the desired stage.
4. Use the mouse to select the desired bolts (you can select multiple bolts with a selection window), and press *Enter* or right-click and select *Done Selection* when you are finished. All selected bolts will be:
 - **INSTALLED** at the stage you are viewing and all subsequent stages

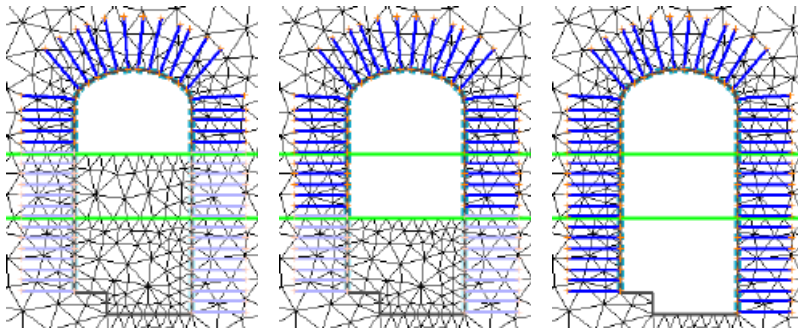
- NOT INSTALLED prior to that stage.

This applies regardless of any previous staging of the bolt(s) – ie. THE *Install* OPTION ALWAYS OVERRIDES ALL PREVIOUS STAGING INFORMATION.

5. Repeat steps 3 and 4 as necessary for all of the bolts in your model.

NOTE:

- When bolts are added to a multi-stage model (using the *Add Pattern* or *Add Spot* options), they are automatically installed at the stage on which you are adding them. So if you plan ahead, you do not necessarily have to assign the bolt staging. Of course if you do not initially add them at the desired stage, you can always use the above procedure to install them at the correct stage.
- Bolts which are NOT INSTALLED at a given stage are displayed in a lighter shade of the bolt type colour. This is illustrated in the figures below.



Three stage model with staging of bolt installation. Note lighter colour of uninstalled bolts

The *Remove* button in the Assign Bolts dialog, allows the user to "uninstall" a bolt AFTER it has been installed (for example, to simulate support loss due to corrosion of bolts). To *Remove* bolts after the installation stage:

1. In the **Assign Properties** dialog select the *Remove* button.
2. Make sure the stage you are viewing is the stage at which you want to uninstall the bolt(s). Use the Stage tabs to view the desired stage.
3. Select the bolts to be "uninstalled", and press *Enter* or right-click and select *Done Selection* when you are finished. All selected bolts will be uninstalled *from that stage forward*, as indicated by the lighter shade of the bolt type colour.

NOTE:

- The most complicated staging any given bolt can receive is therefore UNINSTALLED, INSTALLED, and UNINSTALLED again, using the *Install* and *Remove* options as described above.

-
- After assigning bolt properties and staging, the correct staging of the bolts should always be verified by selecting the stage tabs, and observing the shading of the bolt colour – a light shade of the bolt colour indicates that a bolt is UNINSTALLED at a given stage.
 - If temporary support is installed within areas to be later excavated, it will be automatically mined out (ie. uninstalled) when the material is excavated. The user does not have to use the *Remove* option in this case – it is automatically taken care of.

Assign Liners

Liners are assigned properties and staged in exactly the same manner as bolts. See the [Assign Bolts](#) topic for information.

NOTE that when you are selecting liners, you are actually selecting the boundary line segments (EXCAVATION, STAGE, MATERIAL or EXTERNAL) to which the liner has been applied. To make sure that you select all segments of a given liner, it is advisable to use a selection window, so that you do not accidentally miss portions of a liner when you are assigning properties and staging.

Assign Joints

Joints are assigned properties in the same manner as bolts or liners – see the [Assign Bolts](#) topic for information.

Joints and Staging

Joints cannot be staged in the way that bolts and liners can be staged, therefore there are no *Install* or *Remove* buttons in the Assign Properties dialog, when *Joints* is selected at the bottom of the dialog. However, note the following:

- Segments of a joint which pass through an excavation, are automatically "removed" or "installed" when the area is excavated, according to whether the joint was defined as *Natural* or *Artificial*, when the joint was initially added to the model. See the [Add Joint](#) topic for details.
- Also, in the [Define Joints](#) dialog, the user can specify that the joint is allowed to slip beginning at a given stage. This can be considered "staging" of the joint, however, this is done through the **Define Joints** dialog, and NOT the **Assign Properties** dialog.

Assign Composite Liners

For the purposes of assigning properties and staging (of the initial liner layer), Composite Liners are equivalent to regular Liners, and the assignment procedure is exactly the same.

See the [Assign Liners](#) and [Assign Bolts](#) topic for details.

Staging of Composite Liner layers after the initial layer, is done through the [Define Composite Properties](#) dialog, and not with the **Assign Properties** dialog, which only establishes the installation stage of the FIRST layer of a Composite Liner.

Reset Assign

To reset property assignments, you can use the options in the **Reset Assign** sub-menu in the **Properties** menu. You can reset assignments for materials only, bolts only, joints only, liners only, or ALL entities of your model.

When property assignments are reset, this means that entities will have the [default properties](#), and all staging assignment (ie. excavation and support sequencing etc.) will be reset.

NOTE: The **Reset Assign** options only reset property **assignments** (ie. assignments made through the use of the **Assign Properties** option). The actual properties which you have defined with the **Define** options are NOT affected and will NOT be reset.

Selecting

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

- Boundaries, for editing, applying tractions, restraints, liners, etc.
- Bolts, for editing
- Vertices, for editing

Here are some useful tips to remember when selecting:

General Rules

1. When an entity is selected, it will temporarily change colour or line style, to indicate that it is selected. For example, boundaries change to dashed red lines when selected.
2. IMPORTANT!!! Selection works as a toggle. That is, whenever an entity is selected, it can be de-selected by simply re-selecting it with the left mouse button. This is useful any time that you select an entity by accident, that you did not intend to select.
3. The toggle described in step 2 above, only applies when selecting individual entities with the left mouse button. If you are selecting entities with a window (see below), they will NOT be de-selected if you re-select them with a window. You can always de-select them individually with the left mouse button, even if they were originally selected with a window.

Selecting with a Window

Entities can be selected by drawing a rectangular window around them with the mouse. This is useful for selecting multiple entities.

1. When selecting entities with a window, there are two modes available:
 - regular WINDOW mode, which requires that entities must be COMPLETELY within the window to be selected.
 - CROSSING window mode, which will select entities within or crossing the window (not applicable to vertices, see note below).
 - The two window modes can be accessed with the right-mouse button, or by typing **w** (for a regular window) or **c** (for a crossing window) in the prompt line while you are selecting. You will find the two window selecting modes to be useful in different situations, experiment with them to see how they work.

NOTE: the CROSSING window mode is only applicable when selecting line segments, or entire entities. It is NOT applicable if you are selecting vertices (for moving or deleting).

Selecting All Entities

To select ALL entities of a given type (eg. all loads, or all bolts), here are some useful shortcuts:

-
- right-click the mouse, and choose *Select All* from the popup menu
 - type the asterisk " * " character in the prompt line, followed by *Enter*
 - draw a selection window (see above) around all entities to be selected.

Selection Filter

The **Selection Filter** allows the user to choose which boundary types can be selected for editing. By default, all boundaries are selectable. The main purpose of the **Selection Filter** is to allow the user to select boundaries with a window, but filter out the selecting of some boundary types which may be within the window. See the [Selection Filter](#) topic for more information.

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.

Default right mouse

When no action is being performed, the default right-click menu makes available:

Zoom Window

Zoom Mouse

Zoom All

Pan

Display Options

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.
- While applying **restraints**, the method of applying restraints can be changed (pick by line segment / pick by node).
- While entering boundaries, the **snap** mode can be toggled (vertex snap / grid snap / no snap)

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 key
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 Displacement, 115
- Add Excavation, 46
- Add External, 47
- Add Joint, 54
- Add Liner, 125
- Add Material, 50
- Add Nodal Force, 105
- Add Pattern Bolts, 118
- Add Piezometric Line, 57
- Add Spot Bolt, 120
- Add Springs, 107
- Add Stage, 52
- Add Vertices, 62
- Advanced Discretization, 71, 80
- Analysis Type, 9
 - Axisymmetric, 14, 15
 - Plain Strain, 13
- Angle Traction, 102
- Arc Segments and Circles, 44
- Artificial Joint, 55
- Assign Bolts, 158
- Assign Composite Liners, 160
- Assign Joints, 159
- Assign Liners, 159
- Assign Materials, 155
- Assigning Properties, 154
- Automatic Boundary Intersection, 45
- Axisymmetric Analysis, 14

B

- Bolt DXF Import/Export, 123
- Bolt Properties, 140
- Bolts
 - Assigning, 159
 - End Anchored, 141
 - Fully Bonded, 141
 - Plain Strand Cable, 142
 - Properties, 140
 - Stretch, 122
 - Swelllex / Split Sets, 145
- Bolts Overview, 117
- Boundaries
 - Add Excavation, 46
 - Add External, 47
 - Add Joint, 54
 - Add Material, 50
 - Add Piezometric Line, 57
 - Add Stage, 52

- DXF Import / Export, 60
- Export DXF, 22
- Import DXF, 19
- Boundary Types, 40

C

- Chugh's Method, 9
- Circles
 - Drawing, 44
- Composite Liners, 152, 160
- Compute, 17
- Constant Field Stress, 92
- Coordinates
 - Entering, 42
- Copy, 25
- Copy Excavation, 67
- Correcting Mistakes, 43
- Custom Discretization, 80
- Custom Discretize, 82
- Custom Field Stress, 95

D

- Define Bolts, 140
- Define Composite Liner, 152
- Define Groundwater Properties, 136
- Define Joints, 150
- Define Liners, 148
- Define Materials, 128
- Delete Bolts, 121
- Delete Boundaries, 60
- Delete Displacement, 116
- Delete Liner, 126
- Delete Nodal Force, 106
- Delete Piezometric Line, 69
- Delete Springs, 108
- Delete Traction, 104
- Delete Vertices, 64
- Discretization, 80
- Discretize, 81, 82
- Display Options, 34
- Drawing Arc Segments and Circles, 44
- DXF Import / Export, 123
- DXF Import/Export, 60

E

- Editing Boundaries
 - Add Vertices, 62
 - Delete Boundaries, 60
 - Delete Vertices, 64
 - Move Boundaries, 61

- Move Vertices, 63
- Rotate, 66
- Scale, 65
- Selection Filter, 68
- Elastic Properties, 132
- Element Numbers, 35
- Element Type, 71, 79
- End Anchored Bolts, 141
- Entering Coordinates, 42
- Excavation Boundaries, 46
- Expansion Factor, 48
- Export DXF, 22, 60
- Exporting Images, 23, 25
- External Boundary, 47, 48, 49

F

- Field Stress, 90
 - Constant, 92
 - Custom, 95
 - Gravity, 94
 - Load Split, 96, 97
- File
 - Opening, 8
 - Saving, 8
- Files, 17
 - Output, 17
- Forces, 36
- Free, 114
- Fully Bonded Bolts, 141

G

- Gradation Factor, 71, 74
- Graded Mesh, 73
- Gravity Field Stress, 94
- Grid, 37
- Grid Interpolation, 9
- Ground Level, 39
- Groundwater, 9, 11, 12
 - Piezometric Lines, 57
 - Water Pressure Grid, 58, 59
- Groundwater Method, 9
- Groundwater Properties, 136, 137, 138

H

- Horizontal / Vertical Traction, 102
- Hu Value, 139

I

- Import DXF, 19, 60
- Import Properties, 20
- Infinite Elements, 111
- Info Viewer, 16
- Initial Element Loading, 130
- Initial Joint Deformation, 151

- Interpret, 18

J

- Joints, 54
 - Artificial, 54, 55
 - Assigning, 159
 - Natural, 54
 - Properties, 150, 151

L

- Limits, 26
- Liner Overview, 124
- Liner Properties, 148
- Liners, 125, 126, 159
 - Adding, 125
 - Assigning, 159
- Load Splitting, 96, 97
- Load Steps, 10

M

- Mapped Meshing, 86
- Material Boundaries, 50, 51
- Material Colour, 129
- Material Name, 129
- Material Properties, 128
 - Assigning, 155, 156
 - Colour, 129
 - Elastic Properties, 132
 - Initial Element Loading, 130
 - Name, 129
 - Staged, 134
 - Strength, 133
 - Unit Weight, 131
- Maximum Number of Iterations, 10
- Mesh, 83
 - Graded, 73, 74
 - Radial, 78
 - Reset, 89
 - Uniform, 76
- Mesh Overview, 70
- Mesh Quality, 87, 88
- Mesh Refinement, 84
- Mesh Setup, 71
- Mesh Type, 71, 72
- Meshing, 85
 - Mapped, 85, 86
- Mistakes
 - Correcting, 43
 - Undo / Redo, 24
- Modifying Traction, 104
- Move Vertices, 63
- Moving Boundaries, 61
- Multiple Files
 - Opening, 8

N		Rotate Boundaries, 66
Natural Joint, 54		Ruler, 39
New Project, 7		
Nodal Forces, 105, 106		S
Deleting, 106		Saving a File, 8
Node Numbers, 35		Scale Boundaries, 65
Normal Traction, 101		Seismic Loading, 99
Number of Excavation Nodes, 71		Selecting Entities, 161
Number of External Boundary Nodes, 74		Selection Filter, 68
Number of Stages, 9		Shortcut Keys, 164
O		Shrink Elements, 35
Opening, 8		Slip Criterion, 150
File, 8		Snap, 38
Multiple Files, 8		Solver Type, 10
Output Files, 17		Splits Sets / Swellex, 145
P		Spot Bolts, 120
Pan, 33		Springs
Pattern Bolts, 118		Adding, 107
Phase2		Deleting, 108
Introducing, 5		Stage Boundaries, 52, 53
Phase2 Modeler, 6		Staged Material Properties, 134
Piezometric Lines, 11, 57, 69, 139		Starting a new project, 7
Deleting, 69		Strength Parameters, 133
Plain Strand Cable, 142		Stress Block, 39
Plane Strain Analysis, 13		Stretch Bolts, 122
Pore Fluid Unit Weight, 12		Support
Pore Pressure Calculation, 139		Add Liner, 125
Project Name, 9		Add Pattern Bolts, 118
Project Settings, 9		Add Spot Bolt, 120
Properties, 127		Assigning, 157
Bolts, 140		Assigning Liners, 159
Groundwater, 136, 137, 138		Bolt Properties, 140
Importing, 20		Bolts, 117
Joints, 150		Composite Liners, 153
Liners, 148, 149		Delete Bolts, 121
Materials, 128		Delete Liners, 126
Properties Overview, 127		Liner Properties, 148, 149
R		Liners, 124, 160
Radial Mesh, 78		Swellex / Split Sets, 145
Reset Assign, 160		
Reset Mesh, 89		T
Restraining X, 113		Thin Plate Spline, 9
Restraining XY, 114		Tolerance, 9
Restraining Y, 113		Traction Examples, 103
Restraint Options, 110		Tractions
Restrains, 109		Angle, 102
Applying, 112		Deleting, 104
Displaying, 36		Displaying, 36
Free / Delete, 114		Horizontal / Vertical, 102
Restrains Overview, 109		Modifying, 104
Right-click Menus, 163		Normal, 101
		Staging, 101
		Tractions Overview, 100

	U		Z
Undo / Redo, 24		Zoom All, 27	
Uniform Mesh, 76		Zoom Excavation, 32	
Unit Weight, 131		Zoom In, 28	
		Zoom Mouse, 31	
	V	Zoom Out, 29	
View Limits, 26		Zoom Window, 30	
	W		
Water Pressure Grid, 12, 58			