Phase^2

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

User's Guide

© 1998 - 2001 Rocscience Inc.

Table of Contents

Support Tutorial – Step 2   75

Surface Excavation Tutorial   99

Joint Tutorial   127

Axisymmetry Tutorial

143

Introduction

The PHASE[2] User's Guide consists of the following tutorials. It is recommended that the user follow the step-by-step instructions to create the models themselves. If the user wishes to skip the modeling process, the finished product of each tutorial can be found in the EXAMPLES folder in your PHASE[2] installation folder, in the files indicated below.

FILES	DESCRIPTION
tut1.fea	**Quick Start Tutorial** – a simple tutorial which will get the user familiar with some of the basic modeling and data interpretation features of PHASE[2]
tut2.fea	**Materials & Staging Tutorial** – introduction to the use of multiple materials and staging.
tut3a.fea tut3b.fea	**Support Tutorial** – this tutorial is presented in two steps; the model is analyzed without support, and then bolts and a liner are installed.
tut4.fea	**Surface Excavation Tutorial** – a surface excavation using gravity field stress is modeled. A traction load is included in the third stage.
tut5.fea	**Joint Tutorial** – a joint is added near an excavation, and the effect of inelastic joint slip is examined.
tut6.fea	**Axisymmetry Tutorial** – how to use the axisymmetric modeling option of PHASE[2], and comparison with plane strain results

PHASE^2 Documentation

The documentation for the PHASE^2 program is organized as follows:

Tutorials

Tutorials are found in the PHASE^2 User's Guide, the manual you are now reading.

For information on any PHASE^2 options which are not covered in the PHASE^2 tutorials, consult the PHASE^2 Help system.

Reference

Detailed reference information on all of the options in the PHASE^2 program is found in the PHASE^2 Help system. To access the Help system:

Select: Help → Help Topics

in the PHASE^2 Model or PHASE^2 Interpret programs.

If you wish to have a paper copy of the PHASE^2 reference information, PDF documents are available, which can be printed. See below for details.

Verification

Verification examples are documented in the PHASE^2 Verification Manual, which is available as a PDF file. See below for details. The PHASE^2 files used for verification, can be found in the VERIFICATION sub-folder in the EXAMPLES folder in your PHASE^2 installation folder.

PDF Files

The PHASE² Tutorial, Reference and Verification documents are all available as PDF (portable document format) files. After you install PHASE², you will find them in the Manuals folder in your PHASE² installation folder. The PDF documents can also be downloaded from our website www.rocscience.com.

PDF files are viewed with Adobe Acrobat reader. The PDF documents can be printed, if you wish to have paper copies of the PHASE² Reference or Verification documentation.

About this Manual

Instructions in the following format:

Select: Boundaries → Add Excavation

are used to navigate the menu selections.

When a toolbar button is displayed in the margin, as shown above, this indicates that the option is available in a PHASE² toolbar. *This is usually the recommended and quickest way to use the option.* If no toolbar button is shown, then the option is only available through the PHASE² menus.

Text in "courier" font, enclosed by a box (eg.):

```
Enter vertex [a=arc,esc=quit]: -5 10
Enter vertex [a=arc,u=undo,esc=quit]: -5 0
Enter vertex [a=arc,u=undo,esc=quit]: 5 0
```

indicates prompt line instructions found at the bottom right of the PHASE² screen. The **bold italic** text at the end of a prompt line indicates the user input for the prompt line. In most cases, the user must press *Enter* to enter the data (eg. co-ordinate pairs).

Text in "`courier`" font enclosed by a box is also used to indicate status bar information (eg.):

```
Maximum Total Displacement = 0.00226428 m
```

✓ *Enter:*

  Project Name = (optional)

✓ Number of Stages = 3

  Analysis = Plane strain

  Max. # of iterations = 500

  Tolerance = 0.001

  # Load Steps = Auto

  Solver Type = Gauss. Elim.

When dialog input is required, in most cases the dialog will be displayed, along with a listing in the margin, as shown at left. Parameters requiring user input are marked by a checkmark (✓). Parameters NOT marked by a checkmark, should already be at the correct values, however, the user should always check ALL input, to make sure it is correct.

Quick Start Tutorial
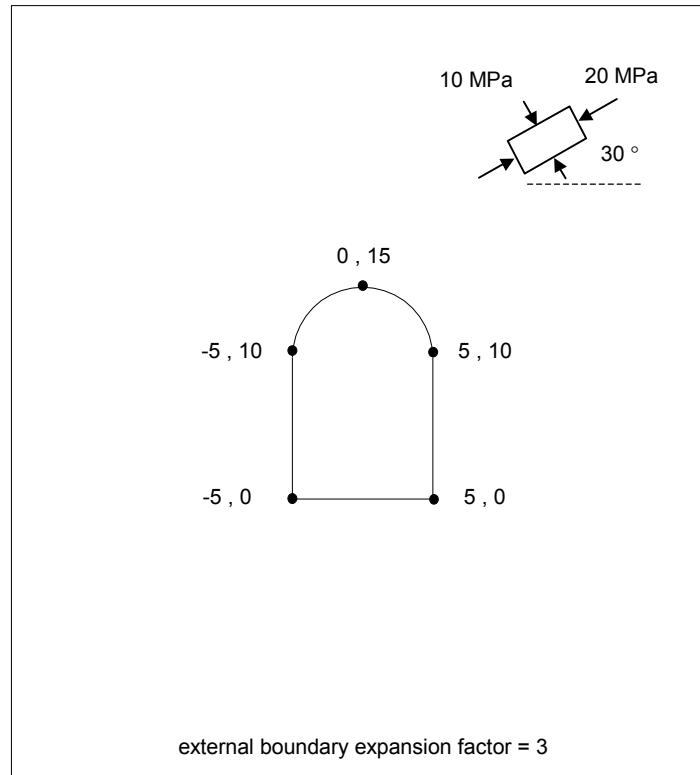
This "quick start" tutorial will demonstrate some of the basic features of PHASE[2] using the simple model shown above. You will see how quickly and easily a model can be created and analyzed with PHASE[2].

NOTE: the finished product of this tutorial can be found in the **tut1.fea** data file, located in the EXAMPLES folder in your PHASE[2] installation folder.

Model

If you have not already done so, run the PHASE² MODEL
program by double-clicking on the PHASE² icon in your
installation folder. Or from the Start menu, select
Programs → Rocscience → Phase2 → Phase2.

If the PHASE² application window is not already
maximized, maximize it now, so that the full screen is
available for viewing the model.
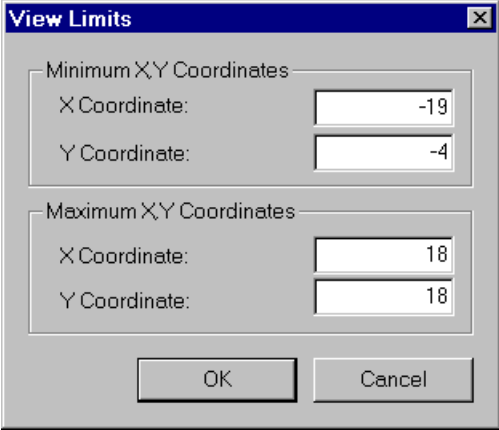
Note that when the PHASE² MODEL program is started,
a new blank document is already opened, allowing you to
begin creating a model immediately.

Limits

Let's first set the limits of the drawing region, so that we
can see the model being created as we enter the geometry.

Select: View → Limits

Enter the following minimum and maximum x-y
coordinates in the View Limits dialog, and select OK.

These limits will approximately center the excavation in the drawing region, when you enter it as described below.

Entering Boundaries

First create the excavation as follows:

Select: Boundaries → Add Excavation

Enter the following coordinates in the prompt line at the bottom right of the screen. Note – press *Enter* at the end of each line, to enter each coordinate pair.

```
Enter vertex [a=arc,esc=quit]: -5 10
Enter vertex [a=arc,u=undo,esc=quit]: -5 0
Enter vertex [a=arc,u=undo,esc=quit]: 5 0
Enter vertex [a=arc,c=close,u=undo,esc=quit]: 5 10
Enter vertex [a=arc,c=close,u=undo,esc=quit]: a
Number of segments in arc <20>: press Enter
Enter second arc point [u=undo,esc=quit]: 0 15
Enter third arc point [u=undo,esc=quit]: c
```

Note the series of prompts used for creating the arched roof. First the "a" command is entered, to begin entering the arc. Then we accepted the default number of arc segments, in this case 20, by pressing *Enter* at the next prompt (although we could have entered a different number). Then an intermediate point on the arc is entered, (0,15), and by entering "c" at the last prompt, the arc closes on the first point of the excavation. In this case, we formed a semi-circle, although a flatter arch could have been formed by lowering the intermediate (second) arc point.

Now we will create the external boundary. In PHASE[2], the external boundary may be automatically generated, or user-defined. We will use one of the 'automatic' options.

Select: Boundaries → Add External

You will see the Create External Boundary dialog. We will use the default settings of Boundary Type = Box and Expansion Factor = 3, so just select OK, and the external boundary will be automatically created.

Create External Boundary

Boundary Type: Box

Expansion Factor: 3

OK   Cancel
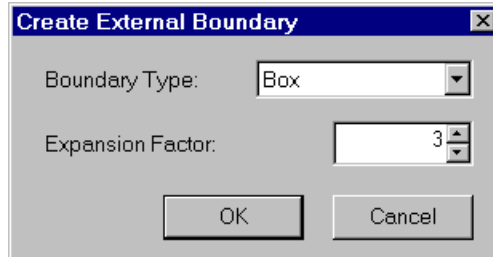
The boundaries for this example have now been entered.

Meshing

The next step is to generate the finite element mesh. In PHASE^2, meshing is a simple two-step process. First you must DISCRETIZE the boundaries, and then the MESH can be generated. You can also configure various Mesh Setup parameters before generating the mesh. We will do this first, although default parameters are in effect if you do not use the Mesh Setup option.

Select: Mesh → Setup

Mesh Setup

Mesh and Discretization Settings

Mesh Type: Graded

Element Type: 3 Noded Triangles

Gradation Factor: 0.1

Default Number of Nodes on All Excavations: 60

☐ Advanced Discretization Settings

Discretize   Mesh   OK   Cancel

✓ *Enter:*

Mesh Type = Graded

Elem. Type = 3 Noded Tri.

Gradation Factor = 0.1

✓ # Excavation Nodes = 60
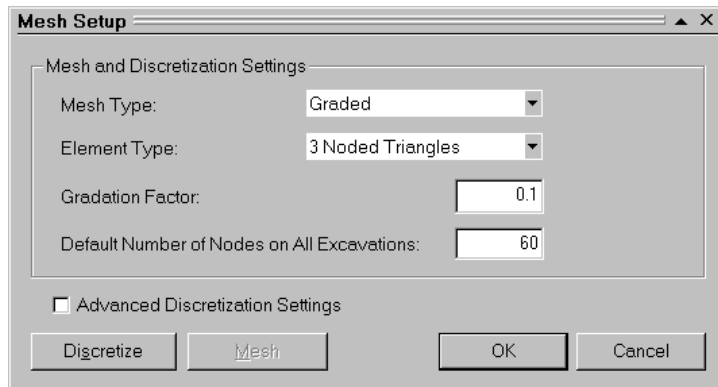
Enter the # of Excavation Nodes = 60, and select OK.

Now discretize the boundaries.

Select: Mesh → Discretize

The discretization of the boundaries, indicated by red crosses, will form the framework for the finite element mesh. Notice the summary of discretization shown in the status bar, indicating the actual number of discretizations for each boundary type.

```
Discretizations: Excavation=59 External=49
```

Note that the number of excavation discretizations is 59, but we entered 60 in the Mesh Setup dialog. Don't worry, this is normal. Due to the nature of the discretization process, the actual number will not always be the same as the number you entered. If you are not happy with a given discretization, it can always be customized using the Custom Discretize option (this is covered in later tutorials), or with the Advanced Discretization option in the Mesh Setup dialog.

Now generate the finite element mesh, by selecting the Mesh option within the Mesh menu.

Select: Mesh → Mesh

The finite element mesh is generated, with no further intervention by the user. When finished, the status bar will indicate the number of elements and nodes in the mesh:

```
ELEMENTS = 981 NODES = 516
```

If you have followed the steps correctly so far, you should get exactly the same number of nodes and elements as indicated above.

Boundary Conditions

For this tutorial, no boundary conditions need to be specified by the user. The default boundary condition will therefore be in effect, which is a fixed (ie. zero displacement) condition for the external boundary.

Field Stress

In PHASE[2] you can define either a Constant field stress or a Gravity field stress. For this tutorial we will use a Constant field stress.

Select: Loading → Field Stress

Field Stress Properties			
Field Stress Type:	Constant ▼		OK
Sigma 1 (MPa, Comp. +):	20		Cancel
Sigma 3 (MPa, Comp. +):	10		
Sigma Z (MPa, Comp. +):	10		
Angle (deg. from horizontal, CCW):	30		
Locked-in horizontal stress (in plane) :	0		
Locked-in horizontal stress (out-of-plane) :	0		Advanced >>

✓ *Enter:*

Fld. Str. Type = Constant

✓ Sigma 1 = 20

Sigma 3 = 10

Sigma Z = 10

✓ Angle = 30

Enter Sigma 1 = 20, Angle = 30, and select OK.

Notice that the small "stress block" in the upper right corner of the view now indicates the relative magnitude and direction of the field stress you entered. *Note the definition of the Constant Field Stress Angle in PHASE[2] – the Angle is the counter-clockwise angle between the Sigma 1 direction and the horizontal axis.*

Properties

We will now define the properties of the rockmass.

Select: Properties → Define Materials

With the first tab selected, enter the following properties:

> ✓ *Enter:*
>
> Name = Material 1
> Init.El.Ld.=Fld Stress Only
> Material Type = Isotropic
> Young's Modulus = 20000
> Poisson's Ratio = 0.2
> Failure Crit. = Mohr Coul.
> Material Type = Elastic
> Tens. Strength = 0
> Fric. Angle (peak) = 35
> ✓ Cohesion (peak) = 12

Enter a cohesion of 12 MPa, and select OK.

Since you entered properties with the first (Material 1) tab selected, you do not have to Assign these properties to the model. PHASE² automatically assigns the Material 1 properties for you.

If you define properties with the Material 2, Material 3, Material 4 etc. tabs (eg. for a multiple material model), then you will have to use the Assign option to assign these properties. We will deal with assigning properties in Tutorial 2.

Excavating

We have one last thing to do to complete our simple model. Although we do not have to assign material properties, we do have to use the Assign Properties option, in order to excavate the material from within the excavation boundary. This is easily done with a few mouse clicks.

Select: Properties → Assign Properties

You will see the Assign Properties dialog, shown in the margin.
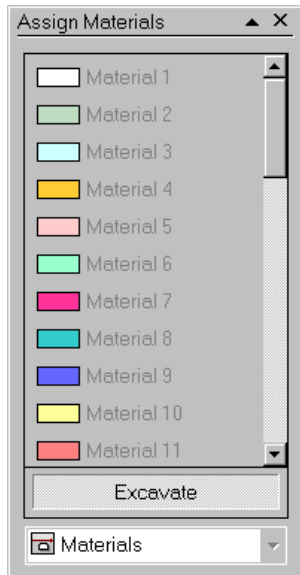
1. Use the mouse to select the Excavate button at the bottom of the Assign Properties dialog.

2. A small cross-hair icon (+) will appear at the end of the cursor. Place the cross-hair anywhere within the excavation boundary, and click the left mouse button.

3. The elements within the excavation boundary will disappear, indicating that the region within the boundary is now "excavated".

4. That is all that is required. Select the X button at the upper right corner of the Assign dialog (or press Escape twice, once to exit the "excavate" mode, and once to close the dialog). The Assign dialog will be closed, and the excavation will be complete.

We are now finished with the modeling, your finished model should appear as shown below.
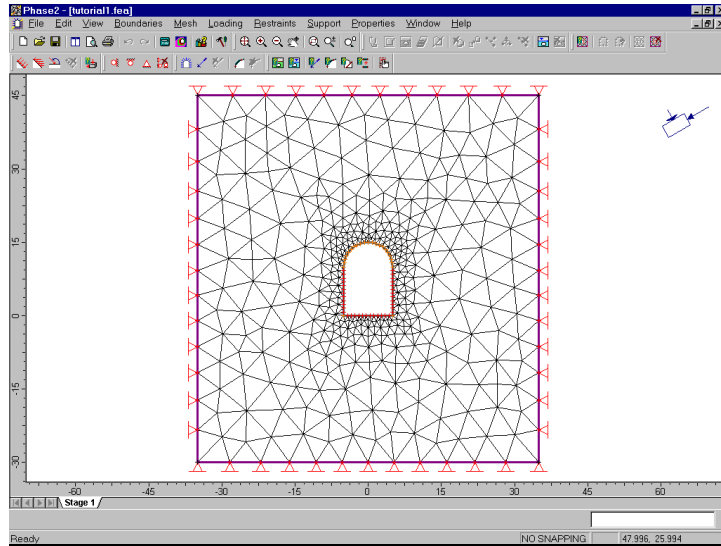
Figure 1-1: Finished model – PHASE^2 Quick Start Tutorial

Compute

Before you analyze your model, save it as a file called **quick.fea**. (PHASE^2 model files have an .FEA filename extension.)

Select: File → Save

Use the Save As dialog to save the file. You are now ready to run the analysis.

Select: File → Compute

The PHASE^2 COMPUTE engine will proceed in running the analysis. When completed, you will be ready to view the results in INTERPRET.

Interpret

To view the results of the analysis:

Select: File → Interpret

This will start the PHASE² INTERPRET program. By default, you will always see a contour plot of the major principal stress Sigma 1, when a file is opened in INTERPRET. This is shown in the figure below.
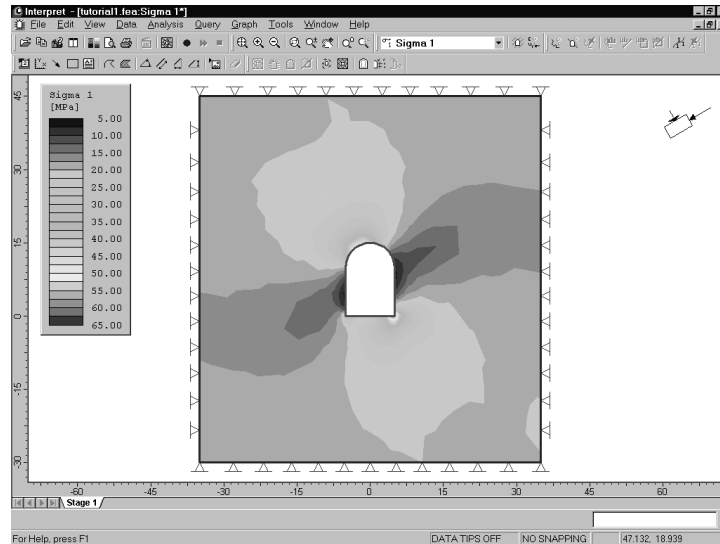
Figure 1-2: Contours of Major Principal Stress

Notice the effect of the field stress orientation (30 degrees from horizontal) on the Sigma 1 contours.

Now let's zoom in to get a closer look at the stress contours around the excavation. An easy shortcut to zoom in to your excavation(s), is to use the Zoom Excavation option.

Select: View → Zoom → Zoom Excavation

Notice the stress buildup at the upper left and lower right of the excavation. The maximum Sigma 1 is at the sharp corner at the lower right.

Now toggle the display of principal stress trajectories on.

Select: View → Display Options

In the Display Options dialog, toggle on Stress Trajectories, and select Done.

The principal stress trajectories are shown as small cross icons where the long axis of the cross is oriented in the direction of the major principal stress (Sigma 1) and the short axis is the direction of the minor principal stress (Sigma 3).

Toggle off the display of stress trajectories by selecting the Stress Trajectories toolbar button. (Frequently used Display Options can be toggled on or off in the Display toolbar).

To display the entire model again (after zooming), select Zoom All (or you can use the F2 function key for the same purpose).

Select: View → Zoom → Zoom All

Let's now look at the Strength Factor contours. Select Strength Factor from the data list in the toolbar.

Select: Strength Factor

Let's change the number of contour intervals, so that we get even numbered intervals.

Select: View → Contour Options

In the Contour Options dialog, enter the Number (of contour intervals) = 7, and select Done.
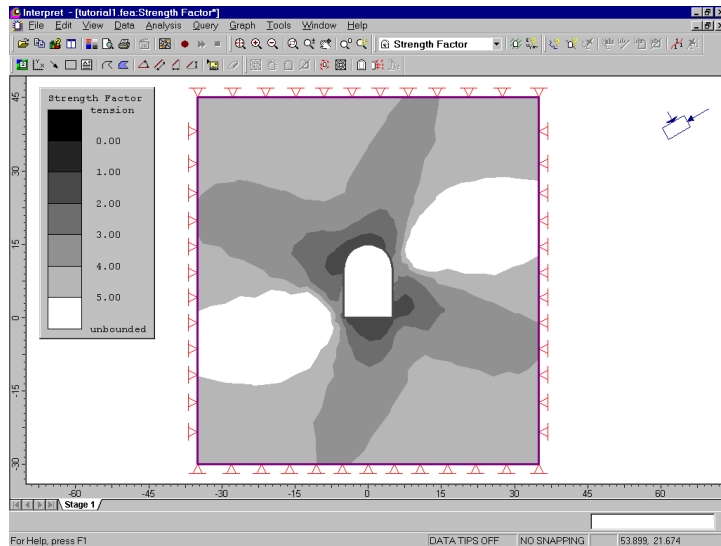
Figure 1-3: Strength Factor Contours.

Notice that the minimum strength factor contour interval is between 1 and 2. Therefore, based on this elastic analysis, no failure is to be expected for this model.

No additional information would be gained from a plastic analysis of the same model. It is left as an exercise for the user to define the material as plastic, and re-run the analysis.

Finally, let's look at the displacements. Select Total Displacement from the data list in the toolbar.

Select: **Total Displacement**

The total displacement contours will be plotted, and the status bar will indicate the maximum displacement for the entire model (about 11 mm).

```
Maximum Total Displacement = 0.01151 m
```

Now select Zoom Excavation again.

Select: View → Zoom → Zoom Excavation

As can be seen from the contours, the maximum displacement is occurring at the excavation walls. Now let's display the deformation vectors and the deformed boundaries.

This time, we will use the buttons in the Display toolbar. Select the Deformed Boundaries and Deformation Vectors buttons in the Display toolbar.

The deformed shape of the excavation boundaries is graphically illustrated by the use of these options. Note that a default scale factor was applied to magnify the deformations. This scale factor can be user-defined in the Display Options dialog.

Frequently used Display Options can be toggled on or off using the buttons in the Display toolbar.
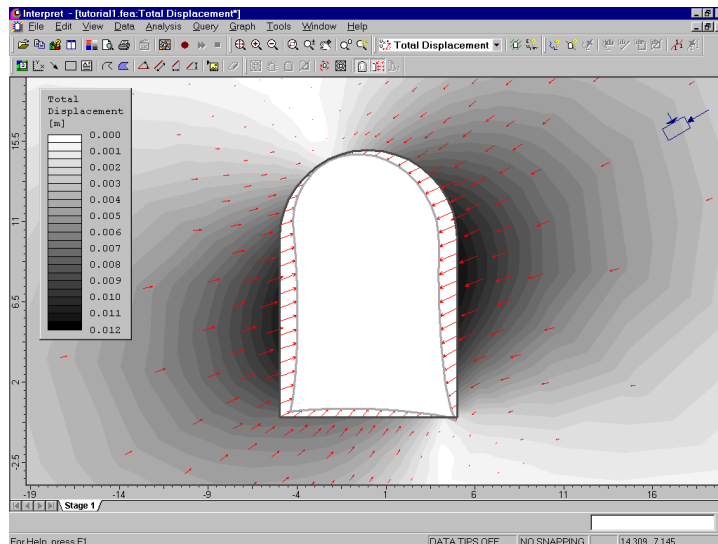
Figure 1-4: Total displacement contours, with deformation vectors and deformed boundaries displayed.

Toggle off the deformation vectors and deformed boundaries by re-selecting the corresponding buttons in the Display toolbar.

Now we will change the number of contour intervals, and add some contour labels. Right-click the mouse and select Contour Options.

In the Contour Options dialog, change the Number (of contour intervals) to 6. Select Done.

Now let's add some labels to the contours, to identify the values represented by each contour boundary.

Select: Tools → Label Contour

A cross-hair cursor will appear on the screen. Click the left mouse button anywhere on a contour boundary, and a contour label will be placed at that point. The following figure illustrates what the display might look like after you have added some contour labels to the model. When you have added all of the labels that you wish, press the *Esc* key or right-click and select Cancel.

> *Contour Options is available in the right-click menu.*
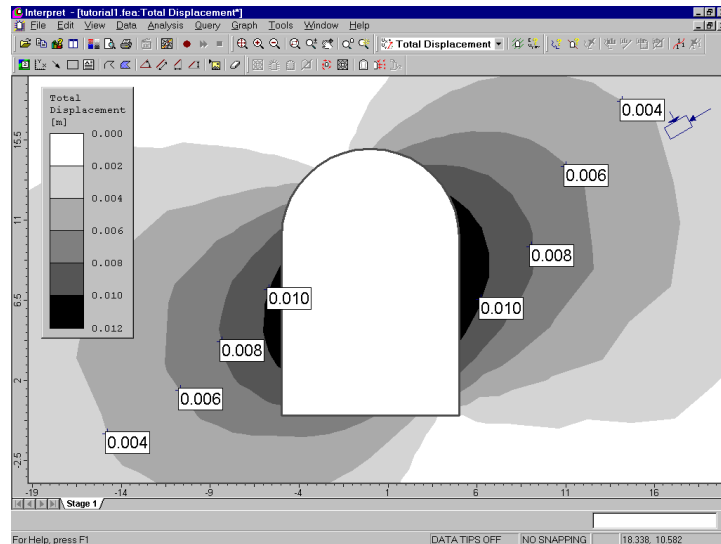
Figure 1-5: Contour labels added to displacement contour plot.

Data Tips

A useful feature of the PHASE² Interpreter, are the popup Data Tips which allows the user to obtain model and analysis information, by simply placing the mouse cursor over any model entity or location on the screen.

To enable Data Tips, click on the box on the Status Bar (at the bottom of the PHASE² application window), which says Data Tips. By default, it should indicate Data Tips Off. When you click on this box, it will toggle through 3 different Data Tip modes. Click on this box until it displays Data Tips Max.

DATA TIPS MAX   SNAP GRID

Now move the mouse cursor over the model, and you will see that the material properties of the rock mass are displayed when the cursor is placed anywhere within the material. Place the cursor over the Stress Block in the upper right corner of the screen, and the Field Stress parameters will be displayed.

Click on the Data Tips box in the Status Bar, until it displays Data Tips Query. This mode allows you to obtain exact interpolated values of data at any point on the contour plots. Move the mouse around the contour plot, and notice that the exact value of the currently contoured variable, will be displayed, as well as the exact location coordinates.

DATA TIPS QUERY SNAP GRID

Click on the Status Bar and toggle Data Tips Off. Data Tips can also display a variety of other information, including support properties etc. The user is encouraged to experiment with this option in later tutorials.

NOTE – Data Tips can also be toggled using the Data Tips sub-menu in the View menu.

Info Viewer

The Info Viewer option in the File menu or the toolbar, displays a summary of PHASE² model parameters, and analysis results, in its own view.

Select: File → Info Viewer
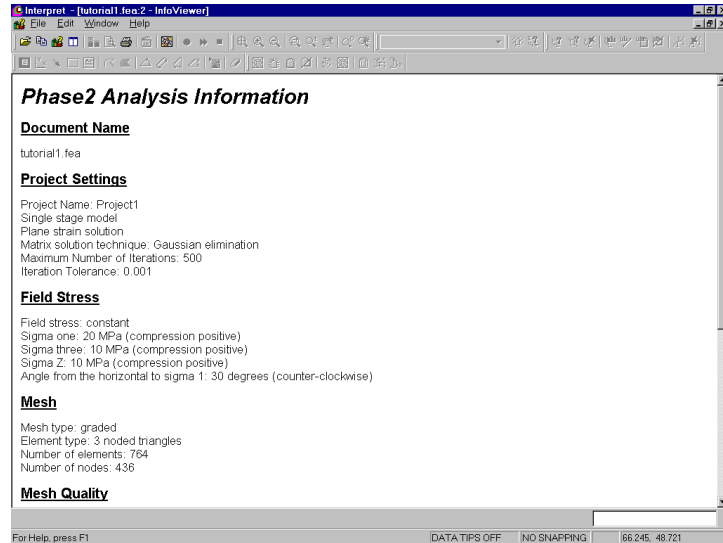
Figure 1-6: PHASE² Info Viewer listing.

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.

The Info Viewer information can also be saved to a text file. The Save As text file options are available in the File menu, (while the Info Viewer is the active view), or by right-clicking in the Info Viewer view.

Close the Info Viewer view, by selecting the X in the upper right corner of the view.

Drawing Tools

In the Tools menu or the toolbar, a wide variety of options are available for customizing views. We will briefly demonstrate some of these options.

First, let's delete the contour labels we added previously.

Select: Tools → Delete Drawings

Right-click the mouse and select Delete All from the popup menu. Select OK in the dialog which appears, and all contour labels will be deleted.

Now press F2 to Zoom All.

Let's add an arrow to the view. Select the Arrow option from the toolbar or the Tools menu.

Select: Tools → Arrow

Click the mouse at two points on the screen, to add an arrow pointing anywhere within the rockmass. Now let's add some text.

Select: Tools → Text Box

Click the mouse at a point near the tail of the arrow. You will see the Add Text dialog. The Add Text dialog allows you to type any text and add it to the screen. The convenient Auto-Text option can be used to annotate the model with pre-formatted input and output data.

For example:

1. In the Add Text dialog, select the Material Properties "+" box (NOT the checkbox). Then select the Material: Material 1 "+" box. Then select the Material: Material 1 checkbox.

2. Now select the Insert Auto-text button. The Material Properties for Material 1, will be added to the editing area at the left of the Add Text dialog.

3. Now select OK. The text is added to the view, and your screen should look similar to Figure 1-7.
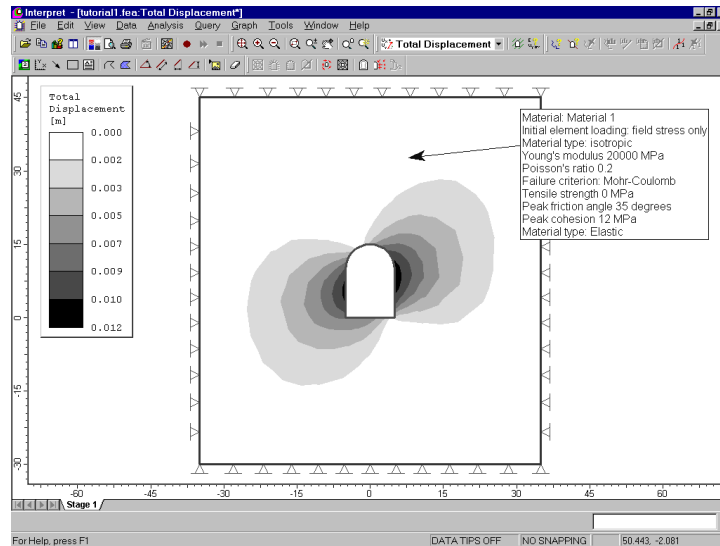
Figure 1-7: Auto-text and arrow added to view.

TIP – After adding a drawing tool, you can easily change the position, size or formatting style, by clicking on the tool with the mouse. This is described in the next section.

Many other drawing tools are available in PHASE², including options which allow the user to add a variety of dimensioning notations on the model, calculate areas of polygons, etc. The user is encouraged to experiment with the many different capabilities of the Drawing Tools in PHASE².

Editing Drawing Tools

We will now describe the following properties of all drawing tools added through the Tools menu options:

Right-click

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

For example:

- right-click on the arrow. Delete, Format and Copy options are available in the popup menu.

- right-click on the text box. Various options are available, including Delete, Format and Edit Text.

Single-click

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

- You can click and drag the control points, to re-size the tool.

- If you hover the mouse over any part of the drawing tool, but NOT on a control point, you will see the four-way arrow cursor, allowing you to click and drag the entire drawing tool to a new location.

- You can delete the tool by pressing Delete on the keyboard.

- You can create a copy of the tool by pressing Ctrl-C on the keyboard, or by selecting Copy from the toolbar or the Edit menu.

Double-click

If you *double-click the mouse* on a drawing tool, you will see the Format Tool dialog. The Format Tool dialog allows the user to customize styles, colours etc. Only the options applicable to the clicked-on tool, will be enabled in the Format Tool dialog. (Note: this is the same Format option available when you right-click on a tool).

It is left as an optional exercise, for the user to experiment with the various editing options that are available for each Tools option.

Saving Drawing Tools

All drawing tools added to a view through the Tools menu, can be saved, so that you do not have to re-create drawings each time you open a file.

Drawing Tools are saved with the Save Tools File option in the Tools menu. These files have a *.pht filename extension. *Only drawing tools of the current (active) view are saved.*

- If you save a tools file with the same name as the corresponding PHASE^2 file, then this tools file will automatically be opened when the PHASE^2 file is opened in INTERPRET, and you will immediately see the saved drawing tools on the opening view.

- If you save a tools file with a DIFFERENT name from the original PHASE^2 file, then you will have to use the Open Tools File option in the Tools menu, to display the tools on the model. This allows you, for example, to save different tools files, corresponding to various views of a model.

Exporting Images

In PHASE[2], various options are available for exporting image files.

Export Image File

The Export Image File option in the File menu or the right-click 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 toolbar or the Edit menu. This will place a bitmap image on the clipboard which can be pasted directly into word or image processing applications.

Black and White Images (Grayscale)

The Grayscale option, available in the toolbar or the View menu, will automatically convert the current view to Grayscale, suitable for black and white image requirements. This can be useful when sending images to a black and white printer, or for capturing black and white image files.

That concludes this 'quick start' tutorial. To exit the INTERPRET program:

Select: File → Exit

Materials & Staging Tutorial
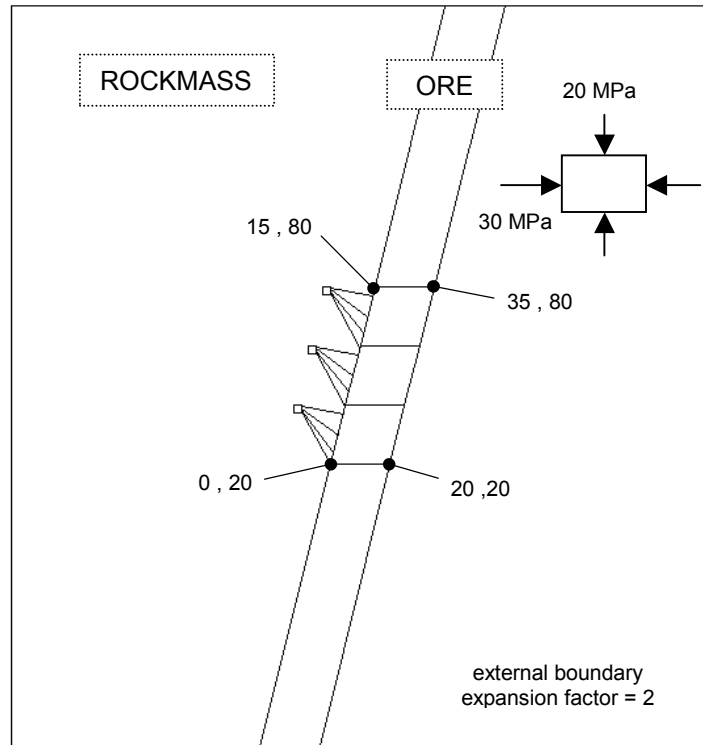
This tutorial will demonstrate the use of multiple materials and staging in PHASE[2], using material and stage boundaries. The model represents a longhole stope in an orebody which has different properties than the surrounding rockmass.

The model will consist of a total of four stages – the stope will be excavated in the first three stages, and will be backfilled in the fourth stage. Support (cables) will also be installed from the access drifts to the hangingwall, however support installation is covered in more detail in the PHASE[2] Support tutorial.

NOTE: the finished product of this tutorial can be found in the **tut2.fea** data file, located in the EXAMPLES folder in your PHASE² installation folder.
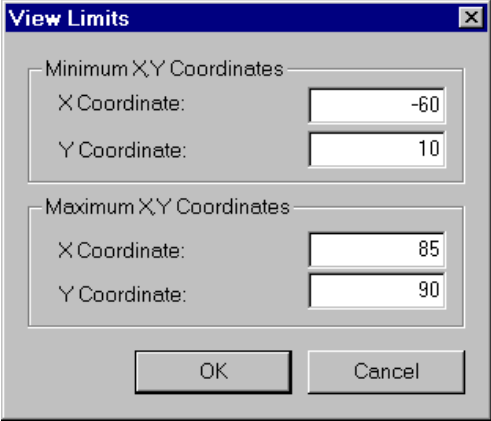
Model

If you have not already done so, run the PHASE² MODEL program by double-clicking on the PHASE² icon in your installation folder. Or from the Start menu, select Programs → Rocscience → Phase2 → Phase2.

Limits

Let's first set the drawing limits so that they encompass the excavations you will be entering.

Select: View → Limits

Enter the following minimum and maximum x-y coordinates in the View Limits dialog, and select OK.

These limits will approximately center the excavations in the drawing region, when you enter them as described below.

Project Settings

Whenever we are creating a staged model, the first thing we should always remember to do is to set the Number of Stages in Project Settings, since this affects subsequent modeling options. That is, some modeling options behave differently if your model is single stage (Number of Stages = 1) or multi-stage (Number of Stages > 1).

Select: File → Project Settings

> ✓ *Enter:*
> Project Name = (optional)
> ✓ Number of Stages = 4
> Analysis = Plane strain
> Max. # of iterations = 500
> ✓ Tolerance = 0.01
> # Load Steps = Auto
> Solver Type = Gauss. Elim.

Project Settings

Project name: Materials & Staging Tutorial

Analysis
Number of Stages: 4
Analysis Type: Plane Strain
Max. Number of Iterations: 500
Tolerance: 0.01
Number of Load Steps: Auto
Solver Type: Gaussian Elimination

Groundwater
Method:
Piezometric Lines

Grid Interpolation:
○ Thin Plate Spline
○ Chugh's Method

Pore Fluid Unit Weight:
0.00981   MN/m3

OK   Cancel

In the Project Settings dialog, enter Number of Stages = 4, and the Tolerance = .01. A Project name is always optional.

We set the Tolerance = 0.01 for this example, to save us time when we run the analysis. The tolerance controls how far the plastic iteration is allowed to proceed, and therefore controls the accuracy of the final solution. A 0.01 tolerance will give us a sufficiently accurate solution for this tutorial.

Entering Boundaries

Let's first enter the stope and the three access drifts. Remember that EXCAVATION boundaries always represent the **final** stage of an excavation for staged models.

Select: Boundaries → Add Excavation

```
Enter vertex [a=arc,esc=quit]: 35 80
Enter vertex [a=arc,u=undo,esc=quit]: 15 80
Enter vertex [a=arc,u=undo,esc=quit]: 10 60
Enter vertex [a=arc,c=close,u=undo,esc=quit]: 5 40
Enter vertex [a=arc,c=close,u=undo,esc=quit]: 0 20
Enter vertex [a=arc,c=close,u=undo,esc=quit]:20 20
Enter vertex [a=arc,c=close,u=undo,esc=quit]:25 40
Enter vertex [a=arc,c=close,u=undo,esc=quit]:30 60
Enter vertex [a=arc,c=close,u=undo,esc=quit]: c
```

Select: Boundaries → Add Excavation

```
Enter vertex [a=arc,esc=quit]: 0 80
Enter vertex [a=arc,u=undo,esc=quit]: -2.5 80
Enter vertex [a=arc,u=undo,esc=quit]: -2.5 77.5
Enter vertex [a=arc,c=close,u=undo,esc=qui]:0 77.5
Enter vertex [a=arc,c=close,u=undo,esc=quit]: c
```

Select: Boundaries → Add Excavation

```
Enter vertex [a=arc,esc=quit]: -5 60
Enter vertex [a=arc,u=undo,esc=quit]: -7.5 60
Enter vertex [a=arc,u=undo,esc=quit]: -7.5 57.5
Enter vertex [a=arc,c=clos,u=undo,esc=qui]:-5 57.5
Enter vertex [a=arc,c=close,u=undo,esc=quit]: c
```

Select: Boundaries → Add Excavation

```
Enter vertex [a=arc,esc=quit]: -10 40
Enter vertex [a=arc,u=undo,esc=quit]: -12.5 40
Enter vertex [a=arc,u=undo,esc=quit]: -12.5 37.5
Enter vertex [a=arc,c=clos,u=und,esc=qui]:-10 37.5
Enter vertex [a=arc,c=close,u=undo,esc=quit]: c
```

Now let's add the two stage boundaries so that the stope can be excavated in three stages. STAGE boundaries can be used **within** excavations for defining intermediate excavation boundaries.

Select: Boundaries → Add Stage

Before we start, right-click the mouse and select Vertex Snap, so that we can snap the stage boundary vertices to the existing excavation vertices.

```
Enter vertex [esc=quit]: use the mouse to click on
the excavation vertex at 10 60
Enter vertex [u=undo,esc=quit]: use the mouse to
click on the excavation vertex at 30 60
Enter vertex [enter=done,u=undo,esc=quit]: right-
click and select Done
```

Notice that when you are in "snap" mode, if you hover the cursor over a vertex, the box cursor changes to a circle, to indicate that you will snap exactly to a vertex, when you click the mouse.

Select: Boundaries → Add Stage

```
Enter vertex [esc=quit]: use the mouse to click on
the excavation vertex at 5 40
Enter vertex [u=undo,esc=quit]: use the mouse to
click on the excavation vertex at 25 40
Enter vertex [enter=done,u=undo,esc=quit]: right-
click and select Done
```

Since we planned ahead and added extra vertices to the stope where the stage boundaries would be, all we had to do was snap to these vertices to add the stage boundaries.

If the stage boundary vertices were not there, we could have still added the stage boundaries using the automatic boundary intersection capability of PHASE[2], which would automatically add the required vertices. This is demonstrated below with the material boundaries.

Next, let's add the external boundary.

Select: Boundaries → Add External

Create External Boundary

Boundary Type: Box

Expansion Factor: 2

OK   Cancel

✓ *Enter:*

   Boundary Type = Box
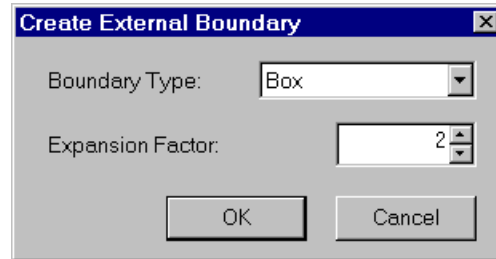
✓ Expansion Factor = 2

Enter an Expansion Factor of 2. Select OK, and the external boundary will be automatically created.

We will now add the material boundaries, which will define the rest of the orebody outside of the excavation.

Select: Boundaries → Add Material

You should still be in vertex snapping mode, if not, then right-click the mouse and select Vertex Snap.

```
Enter vertex [esc=quit]: use the mouse to click on
the excavation vertex at 15 80
Enter vertex [u=undo,esc=quit]: enter the point 40
180 in the prompt line
Enter vertex [enter=done,u=undo,esc=quit]: press
Enter
```

Select: Boundaries → Add Material

```
Enter vertex [esc=quit]: use the mouse to click on
the excavation vertex at 35 80
Enter vertex [u=undo,esc=quit]: enter the point 60
180 in the prompt line
Enter vertex [enter=done,u=undo,esc=quit]: press
Enter
```

Select: Boundaries → Add Material

```
Enter vertex [esc=quit]: use the mouse to click on
the excavation vertex at 0 20
Enter vertex [u=undo,esc=quit]: enter the point
-25 -80 in the prompt line
Enter vertex [enter=done,u=undo,esc=quit]: press
Enter
```

Select: Boundaries → Add Material

```
Enter vertex [esc=quit]: use the mouse to click on
the excavation vertex at 20 20
Enter vertex [u=undo,esc=quit]: enter the point –5
-80 in the prompt line
Enter vertex [enter=done,u=undo,esc=quit]: press
Enter
```

You have just added four material boundaries, representing a continuation of the orebody above and below the excavation. Note the following important point:

The second point you entered for each of the four material boundaries was actually slightly outside of the external boundary. PHASE^2 automatically intersected these lines with the external boundary, and added new vertices. This capability of PHASE^2 is called 'automatic boundary intersection', and is useful whenever exact intersection points are not known, or whenever new boundaries cross existing boundaries where vertices were not previously defined.

Since we knew the slope of the material boundaries but not the exact intersection with the external boundary, we just picked a point outside of the external boundary and PHASE^2 calculated the exact intersection.

We are finished defining the boundaries for this model, so let's move on to the meshing.

Meshing

As usual, we will discretize and mesh the model. We will use the default Mesh Setup parameters this time, so just proceed directly to Discretize.

Select: Mesh → Discretize

All of the model boundaries will be discretized, and the status bar will show a summary of the total number of discretizations for each boundary type.

```
Discretizations: Excavation=76, External=47,
Material=48, Stage=16
```

Now generate the mesh by selecting the Mesh option within the Mesh menu.

Select: Mesh → Mesh

The finite element mesh will be generated, based on the discretization of the boundaries, and the status bar will show the total number of elements and nodes in the mesh.

```
ELEMENTS = 1297 NODES = 673
```

The mesh appears satisfactory, so we will proceed with the modeling. (Note: the mesh quality can always be inspected with the Show Mesh Quality option in the Mesh menu. This is left as an optional exercise to explore after completing this tutorial, and is described in the PHASE^2 Help system).

Boundary Conditions

For this tutorial, no boundary conditions need to be specified by the user. The default boundary condition will therefore be in effect, which is a fixed (ie. zero displacement) condition for the external boundary.

Support

We will support the hangingwall of the stope with cable bolts installed from the access drifts. To save some time, we will import the bolt geometry from a DXF file, since support installation (pattern bolting and liners) is covered in more detail in the PHASE2 Support Tutorial.

Select: File → Import → Import DXF

Read in the bolt coordinates from a DXF file.

Dxf Options

- ☐ Excavations
- ☐ External Boundary
- ☐ Materials
- ☐ Stages
- ☐ Joints
- ☐ Piezo Lines
- ☑ Bolts
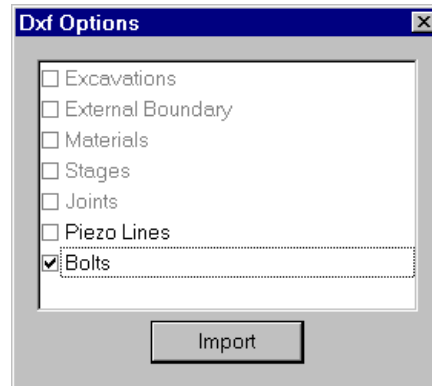
Import

In the DXF Options dialog, select only the Bolts checkbox and select Import.

You will now see an Open file dialog. Open the bolts.dxf file which you should find in the EXAMPLES folder of your PHASE2 installation folder.

Twelve cables (thick blue lines) should now be installed from the access drifts to the hangingwall. Normally, these bolts would be installed using the Add Spot Bolt option, but that is left as an optional exercise for the user to experiment with after completing this tutorial.

To get a better look at the bolts:

Select: View → Zoom → Zoom Excavation

When finished, press F2 to Zoom All.

Field Stress

For this tutorial we will use a constant field stress.

Select: Loading → Field Stress

✓ *Enter:*

   Fld. Str. Type = Constant

✓ Sigma 1 = 30

✓ Sigma 3 = 20

✓ Sigma Z = 20

   Angle = 0

In the Field Stress dialog, enter a constant field stress of Sigma 1 = 30 MPa and Sigma 3 = Sigma Z = 20. Leave the Angle = 0 degrees. Select OK.
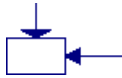
Notice that the stress block now indicates the relative magnitude and direction of the in-plane principal stresses you entered. The angle in this case is zero, so Sigma 1 is horizontal.

Properties

This is where most of the 'action' will be in this tutorial, as far as the modeling is concerned. First we will define the material properties (rockmass, ore, and backfill) and the bolt properties, and then we will assign these properties and the staging sequence to the various elements of our model.

Defining Properties

Select: Properties → Define Materials

With the first tab selected at the top of the Define Material Properties dialog, enter the rockmass properties.

✓ *Enter:*

✓ Name = rockmass
   Init.El.Ld.=Fld Stress Only
   Material Type = Isotropic
✓ Young's Modulus = 64000
✓ Poisson's Ratio = 0.25
✓ Failure Crit. = Hoek-Brown
✓ Material Type = Plastic
✓ Comp. Strength = 110
✓ m (peak) = 10
✓ s (peak) = 0.05
✓ Dilation = 2.5
✓ m (residual) = 10
✓ s (residual) = 0.02

Select the second tab and enter the ore properties, and select the third tab and enter the backfill properties. Select OK when you are finished.

✓ *Enter:*

✓ Name = ore
   Init.El.Ld.=Fld Stress Only
   Material Type = Isotropic
✓ Young's Modulus = 35000
✓ Poisson's Ratio = 0.25
✓ Failure Crit. = Hoek-Brown
✓ Material Type = Plastic
✓ Comp. Strength = 54
✓ m (peak) = 2
✓ s (peak) = 0.02
   Dilation = 0
✓ m (residual) = 2
✓ s (residual) = 0.01

Define Material Properties

☐ rockmass   ☐ ore   ☐ Material 3   ☐ Material 4   ☐ Material 5   ☐ Material 6   ◄ ►

Name: ore   Initial Element Loading: Field Stress Only

Material Colour: [   ]   Unit Weight (MN/m3): 0.027

Elastic Properties
Material Type: Isotropic

Young's Modulus (MPa): 35000   Poisson's Ratio: 0.25

E1 (MPa): 2000   E2 (MPa): 2000   Ez (MPa): 2000
v12: 0.2   v1z: 0.2   v2z: 0.2

Strength Parameters
Failure Criterion: Hoek-Brown   Material Type: Plastic

Comp. Strength (MPa): 54   Dilation Param.: 0

m Parameter (peak): 2   m Parameter (resid): 2

s Parameter (peak): 0.02   s Parameter (resid): 0.01

Stage Material Properties
☐ Apply Factors   Define Factors...

OK   Cancel

✓ *Enter:*

✓ Name = backfill
✓ Init.El.Ld.=Body Force Only
✓ Unit Weight = 0.023
   Material Type = Isotropic
✓ Young's Modulus = 2000
✓ Poisson's Ratio = 0.025
✓ Failure Crit. = Hoek-Brown
✓ Material Type = Plastic
✓ Comp. Strength = 7.5
✓ m (peak) = 6
✓ s (peak) = 1
✓ Dilation = 1.5
✓ m (residual) = 6
✓ s (residual) = 1

Define Material Properties

☐ rockmass   ☐ ore   ☐ backfill   ☐ Material 4   ☐ Material 5   ☐ Material 6   ◄ ►

Name: backfill   Initial Element Loading: Body Force Only

Material Colour: [   ]   Unit Weight (MN/m3): 0.023

Elastic Properties
Material Type: Isotropic

Young's Modulus (MPa): 2000   Poisson's Ratio: 0.025

E1 (MPa): 2000   E2 (MPa): 2000   Ez (MPa): 2000
v12: 0.2   v1z: 0.2   v2z: 0.2

Strength Parameters
Failure Criterion: Hoek-Brown   Material Type: Plastic

Comp. Strength (MPa): 7.5   Dilation Param.: 1.5

m Parameter (peak): 6   m Parameter (resid): 6

s Parameter (peak): 1   s Parameter (resid): 1

Stage Material Properties
☐ Apply Factors   Define Factors...

OK   Cancel

Notice the properties we gave to the ore and the backfill. The orebody has a significantly lower stiffness and strength than the rockmass. The backfill has very low stiffness and strength. In addition, the 'Initial Element Loading' for the backfill was toggled to 'Body Force Only' – *the field stress component of initial element loading for a backfill material should always be zero*. 'Body Force Only' implies that the initial element loading is due to self-weight only.

We are finished defining the material properties. Select OK to close the Define Material Properties dialog, and we will now define the bolt properties.

Select: Properties → Define Bolts

✓ *Enter:*

✓ Name = cables

✓ Bolt Type = Plain Strd Cbl

   Borehole Diameter = 48

   Cable Diameter = 19

   Cable Modulus = 200000

   Cable Peak = 0.1

   Water/Cement Ratio = 0.35

✓ Out-of-Plane Spacing = 2

   Attached Face Plates = ✓

Enter the bolt properties with the first tab selected and select OK.

If you zoom in to the access drifts, you will notice circular markers now appear at the upper end of each cable, in the access drifts. These markers represent the cable face plates. For details about the Plain Strand Cable model see the Help system and references in the PHASE[2] MODEL program. Select F2 to Zoom All.

You have now defined all the necessary material and bolt properties. We will now proceed to the final part of our modeling, the *assigning* of the properties and staging sequence.

Assigning Properties

Select: Properties → Assign Properties

The Assign Properties dialog allows us to assign the properties we defined to the various elements of our model. In conjunction with the Stage Tabs at the bottom left of the view, it also allows us to assign the staging sequence of the excavations and support.
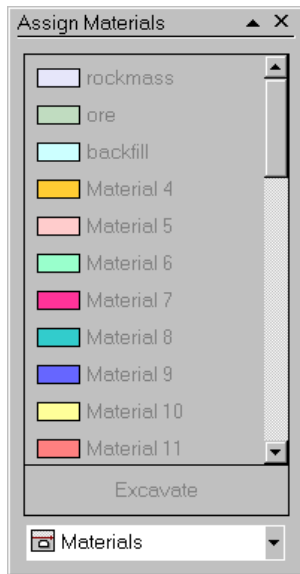
- In the first stage, we will assign the ore properties, and also excavate the bottom section of the stope, and the three access drifts.

- In the second stage we excavate the middle section of the stope.

- In the third stage we excavate the top section of the stope.

- In the fourth stage, we backfill the entire stope.

Assign Materials

1. Make sure the Stage 1 tab is selected (at the bottom left of the view).

2. Make sure the Materials option is selected at the bottom of the Assign dialog.

3. Select the "ore" button in the Assign dialog. (Notice that the material names are the names you entered when you defined the three materials -- ie. rockmass, ore and backfill).

4. Click the left mouse button in the orebody zones above and below the excavation, as well as the two upper sections of the stope. Notice that these elements are now filled with the colour representing the 'ore' property assignment.

5. Select the "Excavate" button in the Assign dialog.

6. Place the cursor in the bottom section of the stope and click the left mouse button. Notice that the elements in this zone disappear, indicating that they are 'excavated'.

NOTE: since we defined the "rockmass" properties using the *first* tab in the Define Materials dialog, the "rockmass" properties do not need to be assigned by the user. The properties of the first material in the Define Materials dialog, are always automatically assigned to all elements of the model. Therefore the rockmass, on either side of the orebody, is already assigned the correct properties, and it is not necessary for the user to assign properties.

Since the access drifts are so small, we'll have to zoom in so we can accurately select them for excavating.

Select: View → Zoom → Zoom Excavation

Now press the F5 function key twice, to zoom in a bit closer. (F5 is equivalent to using the Zoom In option.)

Again, notice the circles which appear at the ends of the bolts, at the access drifts we want to excavate – these represent the cable faceplates.

7. You should still be in "Excavate" mode (if not, select the "Excavate" button in the Assign dialog.)

8. Place the cursor in each of the three access drifts, and left click to excavate them.

9. Select the Stage 2 tab.

10. Place the cursor in the middle section of the stope and click the left mouse button, and the elements will disappear.

11. Select the Stage 3 tab.

12. Place the cursor in the top section of the stope and click the left mouse button, and the elements will disappear.

13. Select the Stage 4 tab.

14. Select the "backfill" button in the Assign dialog.

15. Click in each of the three sections of the stope, and the elements will reappear, with the colour representing the 'backfill' property assignment.

16. You are now finished assigning materials. As an optional step, select each Stage Tab, starting at Stage 1, and verify that the excavation staging and material property assignment is correct.

Assign Bolts

Since we defined our bolt properties with the *first* bolt property tab selected in the Define Bolt Properties dialog, we don't have to assign properties (since they are automatically assigned), but we do have to assign the staging sequence of the bolt installation.

1. At the bottom of the Assign dialog, select the Bolts option from the drop down combo box.

2. Select the Stage 2 tab.

3. Select the "Install" button in the Assign dialog.

4. Use the mouse to select the middle 4 set of bolts. Right click and select Done Selection.

5. Select the Stage 3 tab.

6. Use the mouse to select the top 4 set of bolts. Right click and select Done Selection.
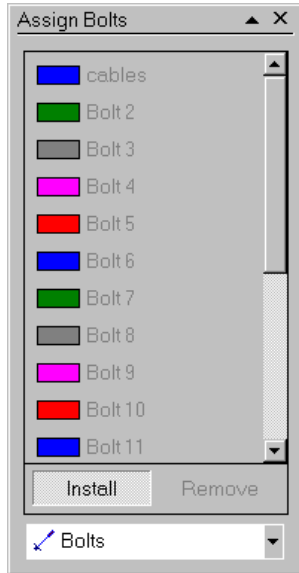
That is all that is required, the bolts should now be installed at the correct stages. Verify your input -- when bolts are NOT installed at a given stage, they are a displayed in a lighter shade of colour.

7. Select the Stage 1 tab. Only the lower set of four bolts should be installed.

8. Select the Stage 2 tab. Both the lower and middle sets of bolts should be installed.

9. Select the Stage 3 tab. All 12 bolts should now be installed.

So we see that the effect of Step 4 above, was to install the middle set of bolts at Stage 2 (and all subsequent stages). The effect of Step 6 was to install the top set of bolts at Stage 3 (and all subsequent stages).

Close the Assign dialog, and press F2 to Zoom All.

You have now completed the modeling phase of the analysis, the model should appear as in the following figure.

Assign Bolts

- cables
- Bolt 2
- Bolt 3
- Bolt 4
- Bolt 5
- Bolt 6
- Bolt 7
- Bolt 8
- Bolt 9
- Bolt 10
- Bolt 11

Install   Remove

Bolts
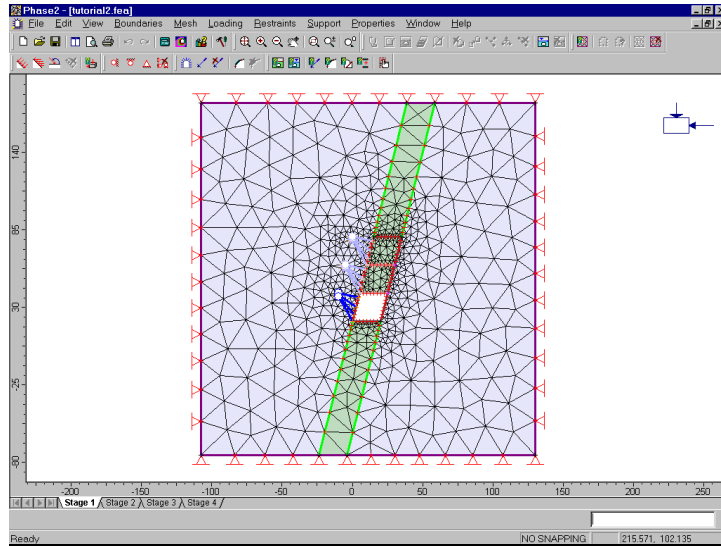
Figure 2-1: Finished model – PHASE^2 Material & Staging Tutorial

Compute

Before you analyze your model, save it as a file called **matstg.fea**.

Select: File → Save

Use the Save As dialog to save the file. You are now ready to run the analysis.

Select: File → Compute

The PHASE^2 COMPUTE engine will proceed in running the analysis. Since we are using PLASTIC materials and bolts, the analysis may take a bit of time, depending on the speed of your computer.

When completed, you will be ready to view the results in INTERPRET.

Interpret

To view the results of the analysis:

Select: File → Interpret

This will start the PHASE[2] INTERPRET program.

Viewing Stages

By default, you will always see the Stage 1 results when a multi-stage model is opened in INTERPRET.

Stage 1 Stage 2 Stage 3 Stage 4
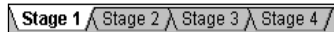
Viewing results at different stages in PHASE[2] is simply a matter of selecting the desired stage tab at the lower left of the view.

Sigma 1

Let's first zoom in.

Select: View → Zoom → Zoom Excavation

You are now viewing the Sigma 1 Stage 1 results. Select the Stage 2, 3 and 4 tabs and observe the changing stress distribution.

Toggle on the principal stress trajectories, using the button provided in the Display toolbar. Again select the stage tabs 1 to 4, and observe the stress flow around the excavation.

If you want to compare results at different stages on the same screen, it can easily be done as follows.

1. Select Window→New Window TWICE, to create two new views of the model.

2. Select the Tile Vertically button in the toolbar, to tile the three views vertically.

3. Select Zoom Excavation in each view.

4. Select the Stage 1 tab in the left view, the Stage 2 tab in the middle view, and the Stage 3 tab in the right view.

5. Display the stress trajectories in each view.

6. Hide the legend in the right and middle views (use View → Legend Options).

7. Right-click in any view and select Contour Options. Click in each view, and select Auto-Range (all stages), to ensure that the same contour range is used for all stages. Close the Contour Options dialog.

Your screen should appear as shown below.
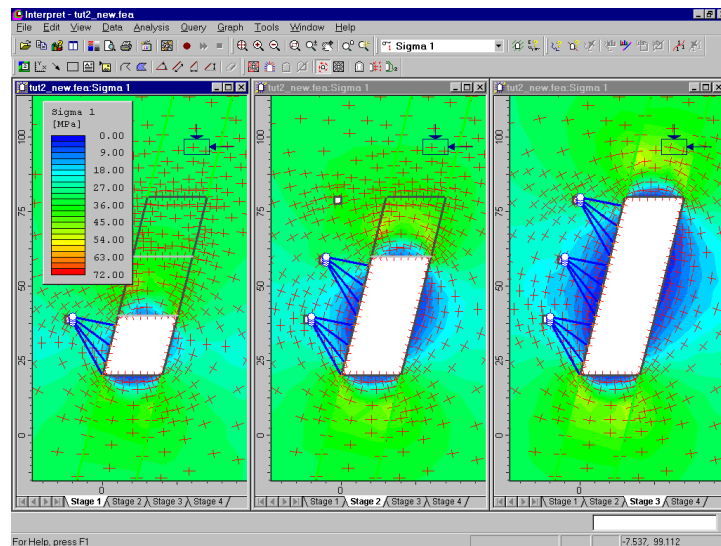
Figure 2-2: Sigma 1 contours, stages 1, 2 and 3. Principal stress trajectories are displayed.

Strength Factor

While we have the three views displayed, let's look at the Strength Factor contours.

Strength Factor ▾

1. Display the Strength Factor in each view.

2. Toggle Stress Trajectories OFF, and Yielded Elements ON, using the Display toolbar buttons, in each view.

Observe the development of strength factor and yielding around the excavation. Note that:

*   The orebody has different strength factor contours than the surrounding rockmass, since we assigned it weaker strength parameters than the rockmass.

*   Most of the yielding is in the back and floor of the stope (ie. in the orebody), although there is some yielding in the rockmass as well.

Let's view the model full screen again. Maximize one of the views (it doesn't matter which one). Re-display the legend if necessary (View → Legend), and select the Stage 3 tab, if necessary.

Zoom in to get a closer look at the yielded elements in the stope back.

Select: View → Zoom → Zoom Window

```
Enter first window point[esc=quit]: 0 100
Enter second window point[esc=quit]: 50 60
```

You will see that there are actually two symbols used for the yielded point markers – failure in shear is indicated by an × marker, and failure in tension is indicated by a ○ marker. This is indicated in the Legend. In most cases, tensile failure is accompanied by shear failure, so the symbols overlap in this case.
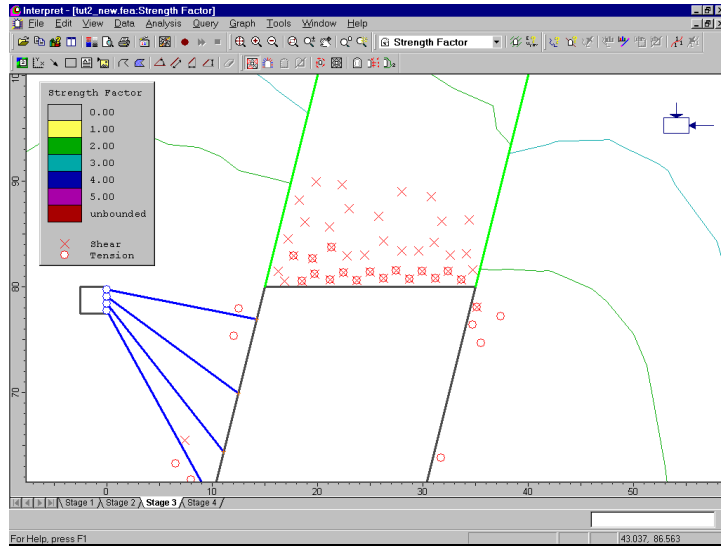
Figure 2-2: Strength Factor contours and yielded elements, Stage 3.

(Note: in the above figure, the following Contour Options were used – Number of Intervals = 7, and Mode = Lines)

As an optional step, use the arrow keys (up / down / left / right), to pan the model around the view. View the contours and yielded elements around the entire excavation. Or pan with the mouse using the Pan option in the Zoom toolbar.

 Display the mesh by selecting the Elements button in the Display toolbar. Notice that each Yielded Element symbol does in fact correspond to a single finite element.

Toggle off the Mesh and select Zoom All. Select the Stage tabs 1 to 4, and observe the strength factor contours on the whole model.

Toggle off the Yielded Elements.

Displacement

Total Displacement ▼

Now look at the total displacements.

Select: Data → Total Displacement

Select the Stage 1 tab. The maximum total displacement for Stage 1 is about 15 mm, as indicated in the status bar.

```
Maximum Total Displacement = 0.01533 m
```

Select the Stage 2 tab.

```
Maximum Total Displacement = 0.02053 m
```

Select the Stage 3 tab.

```
Maximum Total Displacement = 0.02720 m
```

Select the Stage 4 tab.

```
Maximum Total Displacement = 0.02726 m
```

The Stage 3 and Stage 4 maximum displacements are almost identical. Thus far we have not discussed the Stage 4 results. This is discussed in the next section.

Zoom in again.

Select: View → Zoom → Zoom Excavation

Right-click the mouse and select Display Options.

In the Display Options dialog, toggle on Deform Boundaries, enter a Scale Factor of 100, and select Done.

Select the Stage tabs 1 to 4 again, and observe the displacement contours with the deformed boundaries displayed.

The deformed boundaries graphically illustrate the inward movement of the excavation boundaries. It is also interesting to observe the shifting of the access drifts towards the hangingwall – if you did not notice this, select the Stage tabs 1 to 3 and observe the displaced outlines of the access drifts.

Note – the Deform Boundaries option is also available in the Display toolbar. However, if you want to customize the scale factor (as we did here, with a scale factor = 100), you will have to use the Display Options dialog.
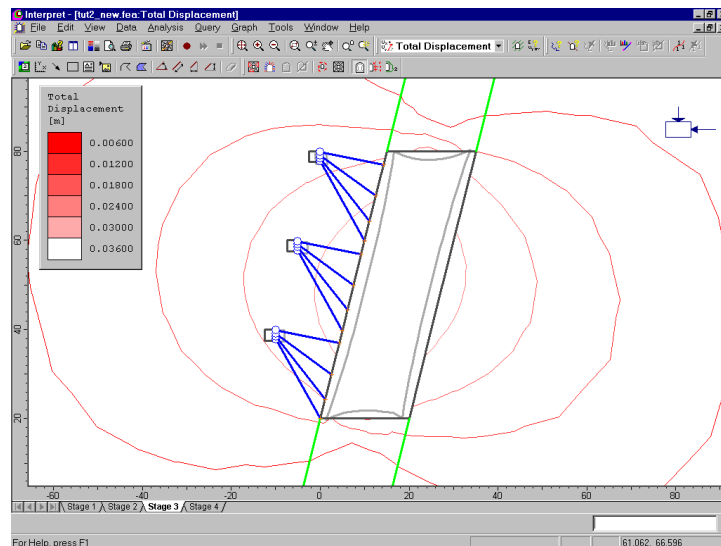
Figure 2-5: Total displacement contours, third stage. Deformed Boundary option toggled ON, Scale Factor = 100

To prepare for the last part of the tutorial, let's close two of the views we created. First tile the views with the Tile option in the Standard toolbar. Then close two of the views. Then maximize the remaining view, and select Zoom Excavation, if necessary.

Stage 4

Remember that in the fourth stage of this model we backfilled the entire stope with a material having representative backfill properties. Except for this, nothing else was changed.

Practically speaking, the backfill has no effect on the results for this model, compared to the third stage results. It is left as an exercise for the user to verify that the contour plots in the third and fourth stage are essentially identical.

- *The purpose of the backfill in this tutorial was to demonstrate how it could be modeled.* A practical use of backfill modeling would be a staged model with several excavations that were excavated and then backfilled in sequence. In this case, the stiffness of the backfill would serve to limit displacements in the backfilled excavations. However, that is beyond the scope of this tutorial, and is left for the user to demonstrate for themselves.

- One final note – remember we specified the Initial Element Loading for the backfill material as Body Force Only. This effectively gives the backfill an active force resisting the excavation deformation, in addition to the passive material stiffness. However, compared to the field stress in this model, this body force is negligible and its effects on the model are minimal. If we were dealing with a surface excavation and gravity field stress, then the body force loading would be more significant. (If we had specified the Initial Element Loading as 'None', then only the backfill stiffness would resist deformation.) See the PHASE[2] Help system for more information about Initial Element Loading.

Bolts

Now let's see what's going on with our bolts. They don't seem to have any obvious effect on the stress or strength contours, so let's see what other information we can gather, using the Graph Bolt Data option. First, select the Stage 3 tab.

Select: Graph → Graph Bolt Data

```
Pick bolts to graph[enter=done,*=all, esc=quit]:
use the mouse to select the lower set of 4 bolts
```

When the four bolts are selected (they are highlighted by a white line when selected), right-click the mouse and select Graph Selected, and you will see the following dialog:
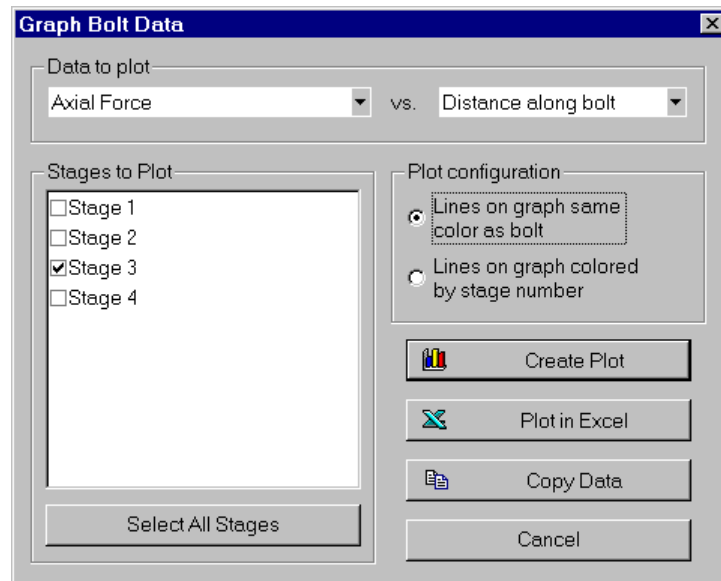
In the Graph Bolt Data dialog, change the Plot Configuration to 'Lines on graph same color as bolt'. Select Create Plot, and a graph of Axial Force for the selected bolts will be generated.

Now repeat the above procedure for the middle and the top sets of four bolts, to generate two more graphs.

Let's tile the graphs we have created, so that we can view them all on one screen.

Select: Window → Tile Vertically

On each graph you will notice a legend, with a bolt number and a stage number. On the model, you will notice numbers on each bolt. The bolt numbers on the model correspond to the bolt numbers on the graphs, allowing you to identify the bolts.

Graph ranges and titles can be changed with the Chart Properties option in the Graph menu or the right-click menu.
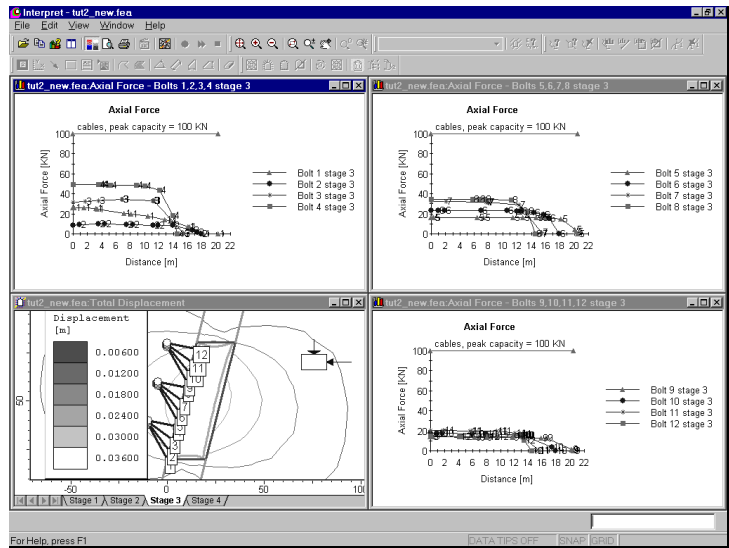
Figure 2-6: Axial force in cables vs. distance along each cable.

Furthermore, the numbers also identify the **end** of each bolt and therefore the **end** of each curve.

The start of each curve therefore represents the end with the face plate, at the access drifts. *An important point to remember when you are installing bolts with face plates **between two excavations** – the **first** point of each bolt must be the end with the face plate. You must remember this when you create the bolt geometry using spot bolting or DXF import.*

Also notice on the plots that the peak capacity of the bolts (100 kN) is indicated by a horizontal line. The force in most of the bolts is well below this line, indicating that there is no yielding in the bolts.

Let's verify that there is no yielding in the bolts. Left-click in the model view, and select the Yielded Bolts button from the Display toolbar.

```
No yielded bolt elements
```

As we expected, no bolts have yielded (if there were yielded bolts, the yielded sections would be highlighted in red).

While we are looking at bolt data, let's illustrate one more feature, the ability to plot data from multiple stages on a single graph. First, maximize the model view (if you are still looking at the tiled view of all the graphs).

Select: Graph → Graph Bolt Data

```
Pick bolts to graph[enter=done,*=all, esc=quit]:
select the first (lowest) bolt on the model
```

Right-click the mouse and select Graph Selected as before, except this time select the first three stages to plot.
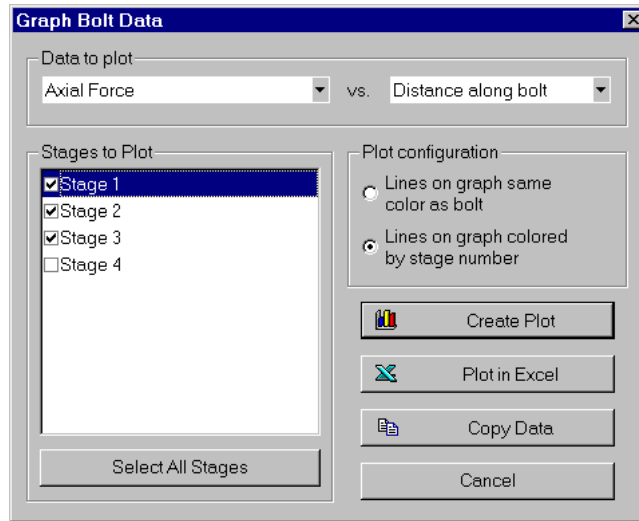
In the Graph Bolt Data dialog, select Stages 1, 2 and 3 to plot. Select Create Plot, and the Axial Force at stages 1, 2 and 3 for the selected bolt will be plotted.
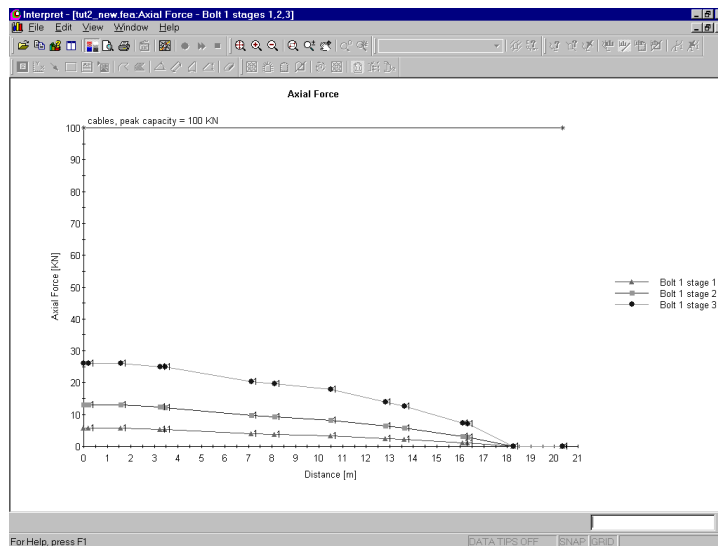
Figure 2-7: Axial force in Bolt 1 at Stages 1, 2 and 3.

To summarize the bolt data interpretation, it is always important to look at the effect of the *excavation* on the *bolts*, and not just the bolts on the excavation. In many cases, the bolts will have little effect on the contour plots (stress, strength, displacement), but will nonetheless be taking a substantial load. Unless the bolts are installed in a zone of yielding with large displacements (see the PHASE² Support Tutorial) this will often be the case.

Examining the load in the bolts allows you to design bolt support by varying bolt parameters (diameter, etc) to obtain optimal stress in the bolt system.

Close the Axial Force plot.

Differential Results

In the Interpret portion of this tutorial, we always used a Reference Stage = 0. Differential results *between any two stages* can be viewed by setting the Reference Stage > 0 in the Stage Settings dialog. For example:

Select: Data → Stage Settings
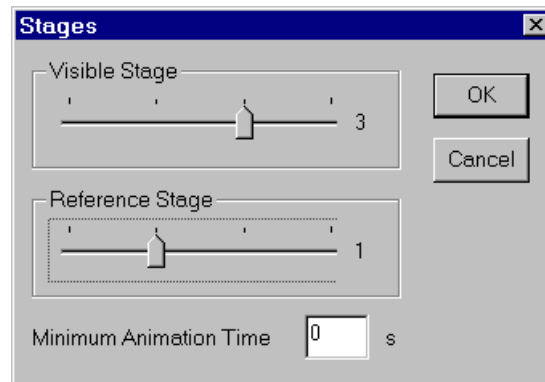
Set the Reference Stage to 1 and select OK.

Notice that the Stage Tabs now allow you to view results relative to the reference stage you just entered.

We will not explore differential results further in this tutorial, but the user is encouraged to explore this on their own. See the Help system in the PHASE[2] INTERPRET program for information about how to interpret differential results.

Log File

Before we conclude this tutorial, let's examine the Log File which is created after a PHASE[2] analysis.

Select: File → Log File

A summary of the number of load steps at each stage, and the number of iterations and tolerance at each load step, is displayed in its own view. Scroll down to view all of the information.

After a plastic analysis, it is a good idea to check the log file, to make sure that the solution converged within the specified tolerance. (The tolerance, number of load steps, and maximum number of iterations, can all be user specified in the Project Settings dialog when you create the model).
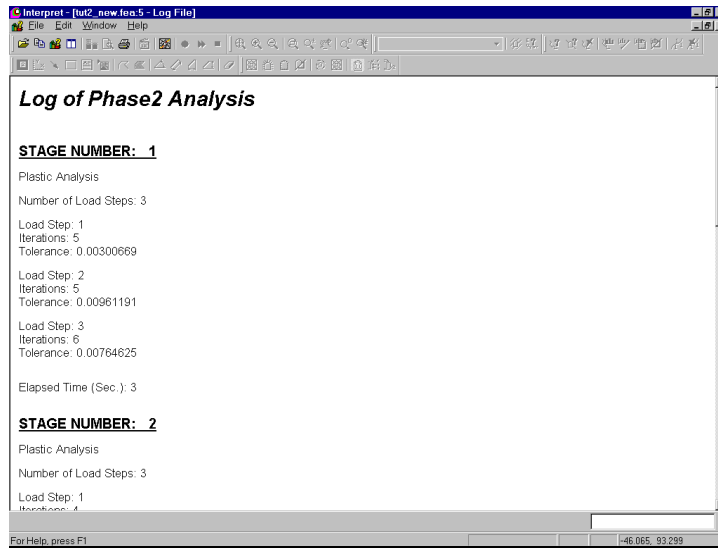
Figure 2-8: PHASE[2] log file.

That concludes the 'Materials and Staging' tutorial. To exit the INTERPRET program:

Select: File → Exit

Support Tutorial – Step 1

external boundary expansion factor = 3
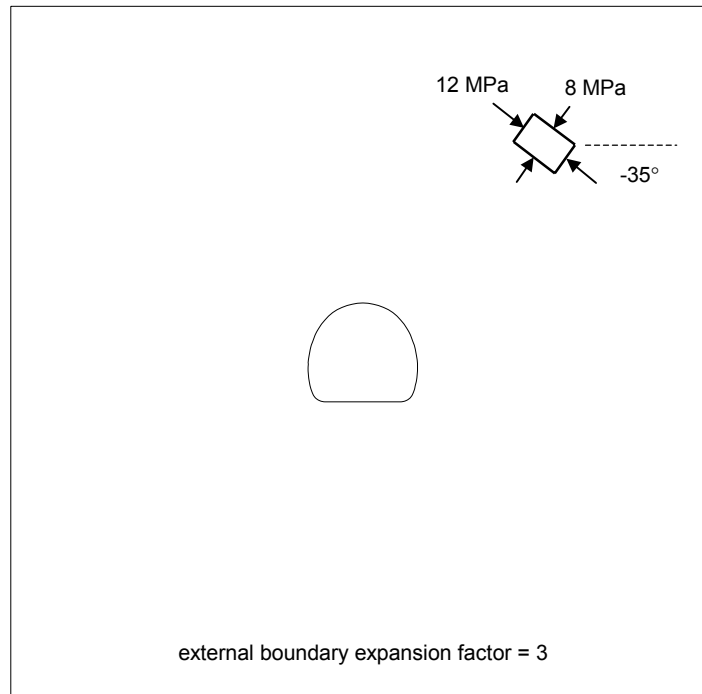
In this first step of the Support tutorial, the above model will be created and analyzed without support. Both an elastic and a plastic analysis will be carried out. Support (bolts and shotcrete) will be added in Step 2 of the Support Tutorial.

If you wish to skip the model process, the finished product of Step 1 of the Support Tutorial (with plastic material parameters) can be found in the **tut3a.fea** data file located in the EXAMPLES folder in your PHASE^2 installation folder.

Model

This model represents a horseshoe shaped tunnel of about 5 metre span, to be excavated in heavily jointed rock. The rock is described as blocky/seamy, of poor quality, and will require support to prevent collapse.

If you have not already done so, run the PHASE² MODEL program by double-clicking on the PHASE² icon in your installation folder. Or from the Start menu, select Programs → Rocscience → Phase2 → Phase2.

Entering Boundaries

For this example, the coordinates defining the tunnel cross-section have already been saved in DXF format, because of the large number of vertices which were used to accurately define the cross-section. Therefore you will not enter coordinates manually as in previous tutorials, but simply read in the DXF file containing the excavation geometry.

Read in the tunnel coordinates from a DXF file.

Select: File → Import → Import DXF
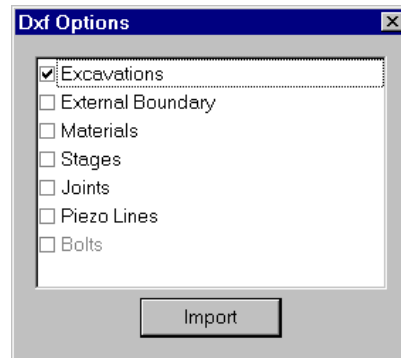
In the DXF Options dialog, select only the Excavations checkbox, and select Import.

You will then see an Open file dialog. Open the tunnel.dxf file which you should find in the EXAMPLES folder in your PHASE[2] installation folder.

You should see the excavation displayed on the screen. Now add the external boundary.

Select: Boundaries → Add External

Create External Boundary

Boundary Type: Box

Expansion Factor: 3

OK　　Cancel

✓ *Enter:*

Boundary Type = Box
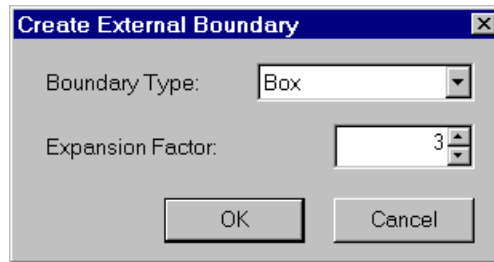
Expansion Factor = 3

We will use the default parameters, so just select OK to automatically create a BOX external boundary with an expansion factor of 3.

This completes the entry of boundaries for this example, so we will proceed to the meshing.

Meshing

We will now proceed to generate the finite element mesh. In PHASE[2], meshing is a simple two-step process. First you must DISCRETIZE the boundaries, and then the MESH can be generated. You can also configure various Mesh Setup parameters before generating the mesh. However, for this example, the default Mesh Setup parameters will be used, and we will proceed directly to the discretization.

Select: Mesh → Discretize

The boundaries will be discretized, and the status bar will display a summary of the number of discretizations for each boundary type.

```
Discretizations: Excavation=83, External=76
```

Now generate the finite element mesh, by selecting the Mesh option within the Mesh menu.

Select: Mesh → Mesh

The finite element mesh will be generated and the status bar will show the total number of elements and nodes in the mesh:

```
ELEMENTS = 1902 NODES = 990
```

Boundary Conditions

For this tutorial, no boundary conditions need to be specified by the user. The default boundary condition will therefore be in effect, which is a fixed (ie. zero displacement) condition for the external boundary.

Field Stress

For this tutorial we will use a constant field stress.

Select: Loading → Field Stress

Field Stress Properties

Field Stress Type:	Constant ▼	OK
Sigma 1 (MPa, Comp. +):	12	Cancel
Sigma 3 (MPa, Comp. +):	8	
Sigma Z (MPa, Comp. +):	10	
Angle (deg. from horizontal, CCW):	-35	
Locked-in horizontal stress (in plane) :	0	
Locked-in horizontal stress (out-of-plane) :	0	Advanced >>

✓ *Enter:*

  Fld. Str. Type = Constant

✓ Sigma 1 = 12

✓ Sigma 3 = 8

  Sigma Z = 10

✓ Angle = –35

In the Field Stress dialog enter Sigma 1 = 12 MPa, Sigma 3 = 8, Sigma Z = 10, and enter an Angle of –35 degrees (note the minus sign!)
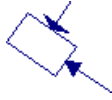
Notice that the stress block now indicates the relative magnitude and direction of the in-plane principal stresses you entered. *Note that the angle you entered is the counter-clockwise angle of Sigma 1 from the horizontal axis. A negative counter-clockwise angle is equivalent to a positive clockwise angle (+35 degrees in this case), as can be seen from the stress block.*

Properties

For the initial analysis, we will be defining the rock mass as elastic.

Select: Properties → Define Materials

> ✓ *Enter:*
>
> ✓ Name = phyllite
>   Init.El.Ld.=Fld Stress Only
>   Material Type = Isotropic
> ✓ Young's Modulus = 1120
> ✓ Poisson's Ratio = 0.3
> ✓ Failure Crit. = Hoek-Brown
>   Material Type = Elastic
> ✓ Comp. Strength = 50
> ✓ m (peak) = 0.43
> ✓ s (peak) = 0.00006

With the first tab selected in the Define Material Properties dialog, enter the above properties.

Since we defined the properties with the first material tab selected, we do not have to assign them to the model. PHASE[2] automatically assigns the properties of the first material, to all finite elements of the rockmass. (Property assignment is covered in detail in the Materials and Staging tutorial).

However, we still have to excavate the elements inside the excavation boundary, which is done with the Assign Properties dialog.

Select: Properties → Assign Properties

1. Select the Excavate button in the Assign dialog.

2. Click the mouse inside the excavation (tunnel) boundary. The elements will disappear, indicating that the tunnel is "excavated".

3. Now close the Assign dialog.

We are finished with the modeling and can run the analysis. Your model should appear as shown in the following figure.
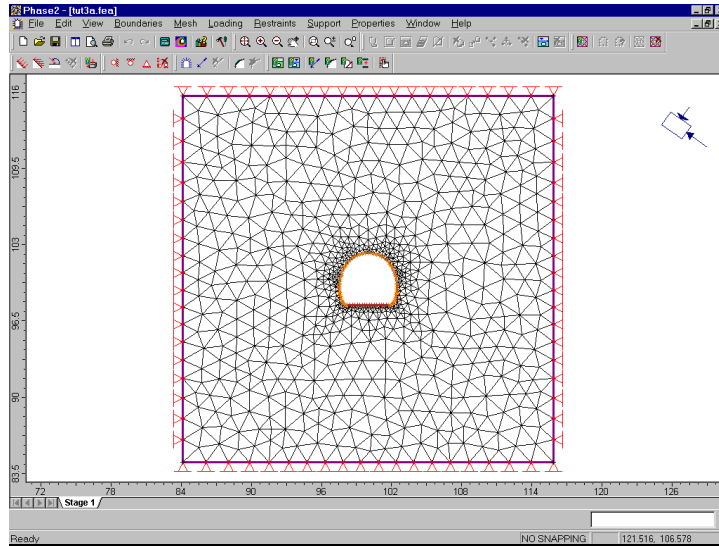
Figure 3-1: Finished model – PHASE^2 Support Tutorial (Step 1)

Compute

Before you analyze your model, save it as a file called support1.fea.

Select: File → Save

Use the Save As dialog to save the file. You are now ready to run the analysis.

Select: File → Compute

The PHASE^2 COMPUTE engine will proceed in running the analysis. When completed, you will be ready to view the results in INTERPRET.

Interpret

Start the PHASE^2 INTERPRET program:

Select: File → Interpret

Let's first view the strength factor contours. Select Strength Factor from the toolbar data list.

Support Tutorial
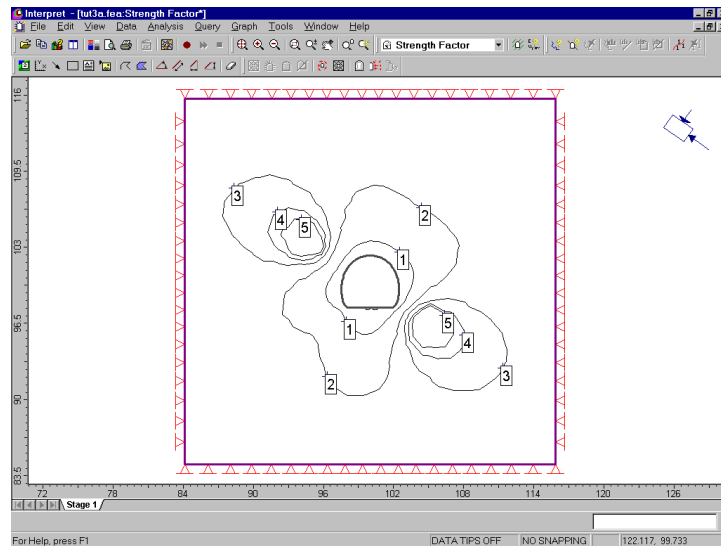
Elastic Analysis

Figure 3-2: Strength Factor Contours, after elastic analysis.

Strength factor represents the ratio of available rock mass strength to induced stress, at a given point. You will immediately notice the large zone of overstress surrounding the tunnel. All of the rock contained within the contour marked 1, has a strength factor (based on the elastic analysis results) less than 1, and will fail if left unsupported. (Note – in the Contour Options dialog, you will have to change the number of contour intervals to 7, and the contour Mode to Lines, to obtain the above figure. See the "Note about the Support tutorial figures" at the end of this section).

If you added contour labels, delete them now using Tools → Delete All Drawings.

Now let's view the displacements.

Select: 〔Ux+ Uy〕**Total Displacement** ▾

Notice the maximum displacement displayed in the status bar.

```
Maximum Total Displacement = 0.03723 m
```

Zoom in, and toggle the displacement vectors on.

Select: View → Zoom → Zoom Excavation

Select: View → Display Options

In the Display Options dialog, toggle Deformation Vectors on, enter a Scale Factor of 10, and select Done.
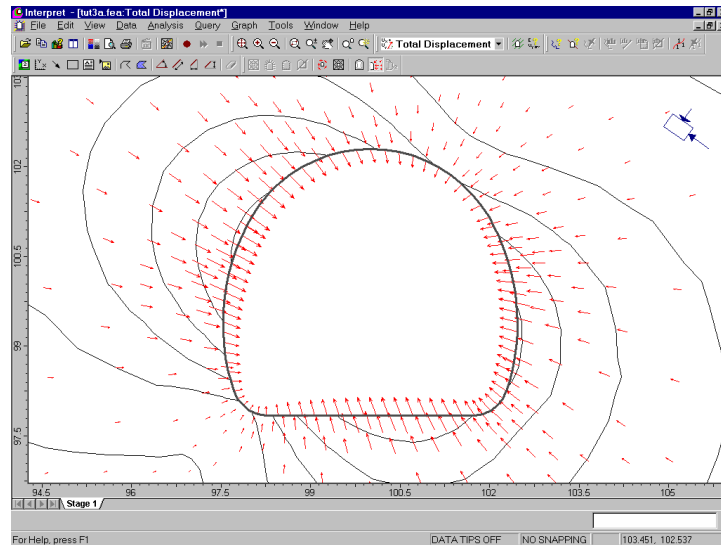
Figure 3-3: Displacement contours and vectors around excavation (elastic analysis).

The elastic displacements show an inward displacement of the tunnel walls, as well as a significant floor heave.

Now that we have determined, from the elastic analysis, that the region of overstress is significant, we will move on to the plastic analysis of this problem.

Note about the Support Tutorial figures

- *The contour plots in the Support tutorial were generated using the Black Lines Auto-Format option in the Contour Options dialog, and customizing the Number of Intervals to a suitable number (eg. 7 or 8). Contour Options is available in the View menu, and also the default right-click menu.*

- *Contour labels are added using the Label Contour option, and deleted using the Delete Drawings or Delete All Drawings options in the Tools menu. Contour labels should be deleted when changing data types, since labels added on one data type (eg. Strength Factor) will not fall on the contours of another data type (eg. Displacement).*

- *The display of the Legend was toggled off in the Support Tutorial figures. The Legend can be toggled on or off in the View menu.*

Model

From INTERPRET, switch back to the PHASE² MODEL program:

Select: File → Model

We will now define the rock mass to be plastic, and re-run the analysis.

Select: Properties → Define Materials

✓ *Enter:*

Name = phyllite
Init.El.Ld.=Fld Stress Only
Material Type = Isotropic
Young's Modulus = 1120
Poisson's Ratio = 0.3
Failure Crit. = Hoek-Brown
✓ Material Type = Plastic
Comp. Strength = 50
m (peak) = 0.43
s (peak) = 0.00006
Dilation = 0
✓ m (residual) = 0.43
✓ s (residual) = 0.00006

Toggle the Material Type to Plastic, and enter residual m and s parameters equal to the peak m and s parameters. This defines the material as ideally elastic-plastic (ie. no strength drop once yield is reached). Select OK.

We are now ready to re-run the analysis.

Compute

Before you analyze your model, let's save this as a new file called support2.fea. (Make sure you select Save As and not Save, or you will overwrite the support1.fea file).

Select: File → Save As

Save the file as support2.fea.

Select: File → Compute

The PHASE^2 COMPUTE engine will proceed in running the analysis. When completed, you will be ready to view the results in INTERPRET.

Interpret

From MODEL, switch back to INTERPRET:

Select: File → Interpret

Let's view the strength factor contours.

Select: Strength Factor

Notice the much larger extent of the region encompassed by the contour of strength factor = 2 (compared with Figure 3-2). Also notice that there is no region of strength factor < 1. Since your material is now plastic, the strength factor cannot go below one – when failure (yielding) occurs, the strength factor is by definition equal to one. It is only in an elastic analysis that the strength factor can go below one, as a hypothetical measure of overstress.

To view the 'failure zone' in a plastic material, toggle the display of yielded elements on, by selecting the Yielded Elements button in the Display toolbar. The number of yielded elements will be displayed in the status bar.

```
583 Yielded finite elements
```

Zoom in using the zoom method of your choice (suggestion – try the Zoom Mouse option). Observe the zone of plastic yielding (× = shear failure, O = tensile failure) around the excavation. Notice that the yielded zone corresponds roughly with the zone of strength factor < 1 from the elastic analysis (Figure 3-2), with additional propagation beyond this limit, as would be expected from a plastic analysis.

Support Tutorial
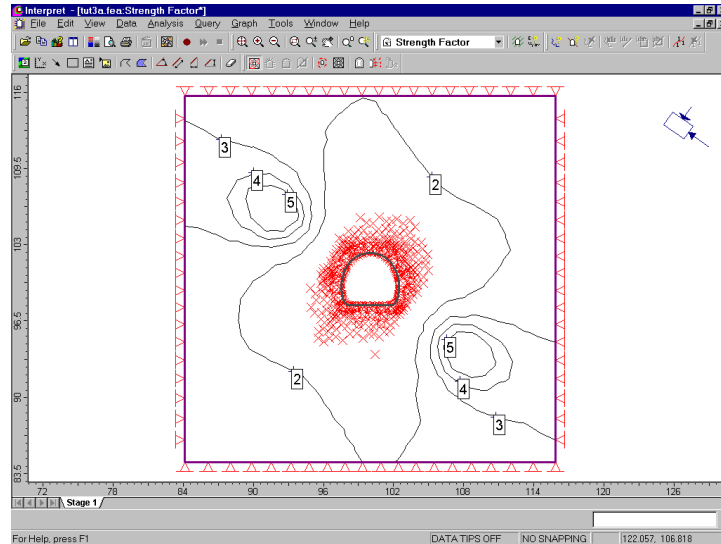
Plastic Analysis

Figure 3-4: Strength factor contours and yielded elements, after plastic analysis of unsupported excavation.

Toggle the display of yielded elements off, by re-selecting the Yielded Elements button in the Display toolbar

Let's look at the plastic displacements.

Select: **Total Displacement**

Note the maximum displacement indicated in the status bar.

```
Maximum Total Displacement = 0.0960 m
```

This is nearly three times the maximum displacement from the elastic analysis. Toggle the deformation vectors on.

Select: View → Display Options

In the Display Options dialog, toggle Deformation Vectors on, enter a Scale Factor of 10, and select Done.

Let's zoom in.

Select: View → Zoom → Zoom Excavation

You should see the following figure. Since we used the same scale factor for the deformation vectors, we can make a direct visual comparison between Figure 3-5 and Figure 3-3.
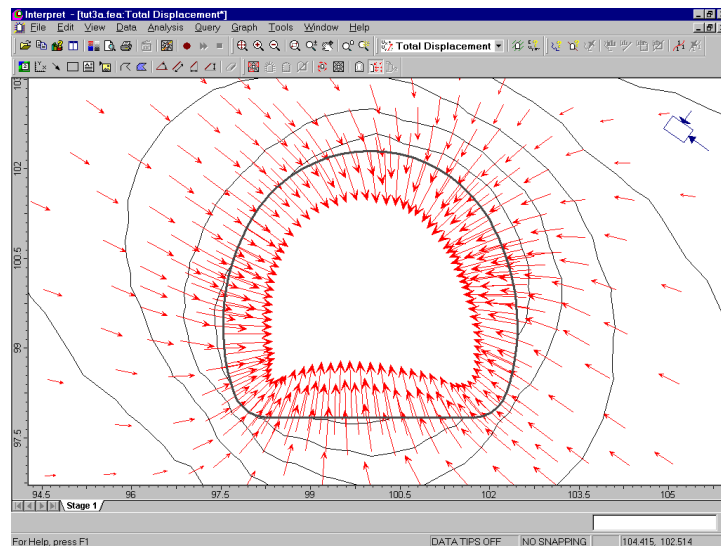
Figure 3-5: Displacement contours and vectors around excavation (plastic analysis).

From the deformation vectors, it can be seen that the overall maximum displacement (0.096m), is occurring in the floor of the tunnel.

You are now finished the first step in the "Support Tutorial". It's about time we added some support! Continue on to Step 2 of this tutorial.

Support Tutorial – Step 2

12 MPa   8 MPa

-35°

external boundary expansion factor = 3
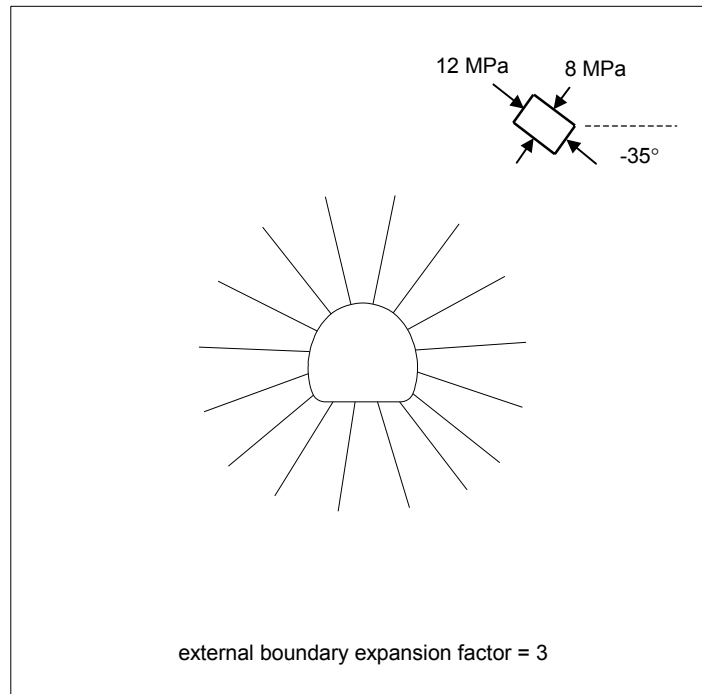
In the first step of the Support tutorial, we analyzed the above model without support, using both elastic and plastic material parameters. We will now proceed to analyze the same model after adding:

1. Bolts only

2. Bolts and shotcrete

3. Bolts and shotcrete in conjunction with load splitting.

The finished product of this tutorial (with bolts and shotcrete, but before the load splitting) can be found in the **tut3b.fea** data file in the EXAMPLES folder of your installation folder.

Model

If you have been following this tutorial from Step 1, then just continue on. If you quit the program after doing Step 1, and wish to continue now, start the PHASE² MODEL program, and open the support2.fea file you created in Step 1.

Select: File → Open

Open the support2.fea file.

Adding Pattern Bolts

> *Now we add the pattern bolt support.*

Let's now add some bolts to this model. We will be installing a radial array of 5 meter long pattern bolts, on a 1 x 1 meter grid.

Before adding our bolt pattern, let's zoom in on the excavation, so that we can accurately select the starting and ending points of the bolt pattern.

Select: View → Zoom → Zoom Excavation

Select: Support → Add Pattern Bolt

Pattern Bolts

Bolt Length (m):	5
In_Plane Spacing (m):	1
Apply Pattern To:	Excavation

Pattern Type:
○ Normal to Boundary   ● Radial

[OK]   [Cancel]

> ✓ *Enter:*
> ✓ Bolt Length = 5
> ✓ In-Plane Spacing = 1
>   Apply Ptn. To = Excavation
> ✓ Pattern Type = Radial

Since you are entering a Radial bolt pattern, you will then be prompted:

```
Select drilling point [esc=quit]: 100 100
```

This is the point from which the radial pattern will be generated. Use the keyboard to enter the exact point (100 , 100) as indicated in the prompt above. You can use the mouse to graphically enter the point, but it is best to use the keyboard when an exact point is desired.

Next you will be prompted to enter starting and ending vertices for the pattern. Because of the poor quality of the rock and extensive yielding all around the tunnel, we will bolt the entire circumference of the tunnel, including the floor. To bolt all the way around a tunnel, select *the same vertex* on the excavation boundary as both the starting and ending vertex. *For example, select the uppermost vertex at the midpoint of the roof.*

To generate bolt patterns around part of an excavation, the second vertex must be selected counter-clockwise relative to the first vertex.

```
Select boundary vertex to start bolt pattern
[esc=quit]: use the cursor to select the vertex
located at about 100.0 , 102.3
Select boundary vertex to end bolt pattern
[esc=quit]: click the mouse again on the same
vertex
```

The bolt pattern will be generated after the second point is entered, and the status bar will tell you how many bolts were installed.

```
Number of bolts installed = 16
```

Now use Zoom All to bring everything back into view. An alternative shortcut to zoom all, that you can use at any time, is to *press the F2 function key*.

Note – only excavation *vertices* can be selected as the start / end points of the bolt pattern. *If you try to select a discretization point on a boundary (ie. the red crosses generated when you Discretize), nothing will happen, unless the discretization point is coincident with a vertex.*

Bolt Properties

Now that the bolt geometry is defined, let's define the bolt properties. The bolt properties we enter will correspond to fully grouted, untensioned steel dowels, with a maximum load capacity of 20 tonnes (0.2 MN).

Select: Properties → Define Bolts

> ✓ *Enter:*
> ✓ Name = grouted
> ✓ Bolt Type = Fully Bonded
> ✓ Bolt Diameter = 25
>   Bolt Modulus = 200000
> ✓ Peak Capacity = 0.2
> ✓ Residual Capacity = 0.2
>   Pre-Tensioning = 0
>   Out-of-Plane Spacing = 1

Define Bolt Properties

| grouted | Bolt 2 | Bolt 3 | Bolt 4 | Bolt 5 | Bolt 6 | Bolt 7 | Bolt 8 | ◄ | ► |

Name: grouted   Bolt Colour:

Bolt Properties

Bolt Type: Fully Bonded
Bolt Diameter (mm):   25
Bolt Modulus,E (MPa):   200000
Peak Capacity (MN):   0.2
Residual Capacity (MN):   0.2
Pre-Tensioning (MN):   0
Out-of-plane Spacing (m):   1

Bolt Model
○ Linear   ○ Non-Linear

Face Plates
☐ Attached Face Plates

☐ Add Pull-Out Force
Force (MN):   0

☐ Constant Shear Stiffness
Stiffness (MN/m/m):   0

OK   Cancel

Enter the bolt properties with the first tab selected.

Since you entered the bolt properties with the first tab selected, you do not have to Assign these properties to the bolts – PHASE² will automatically assign the properties for you.

We are now ready to re-run the analysis.

Compute

Before you analyze your model, let's save this as a new file called support3.fea. (Make sure you select Save As and not Save, or you will overwrite the support2.fea file).

Select: File → Save As

Save the file as support3.fea.

Select: File → Compute

The PHASE[2] COMPUTE engine will proceed in running the analysis. When completed, you will be ready to view the results in INTERPRET.

Interpret

From MODEL, switch back to INTERPRET:

Select: File → Interpret

Let's see the effect of the bolts on the strength factor contours.

Select: Strength Factor

The extent of the region encompassed by the contour of strength factor = 2 is noticeably reduced, compared with Figure 3-4.

Toggle the display of yielded elements on by selecting the Yielded Elements button in the Display toolbar. The number of yielded elements will be displayed in the status bar.

```
546 Yielded finite elements
```

The yielded zone, based on the extent and location of the yielded elements, is not discernibly different from the unsupported yield zone shown in Figure 3-4. However, the number of yielded finite elements decreased from 583 (unsupported) to 546 (supported).

Support Tutorial

Plastic Analysis + Bolts
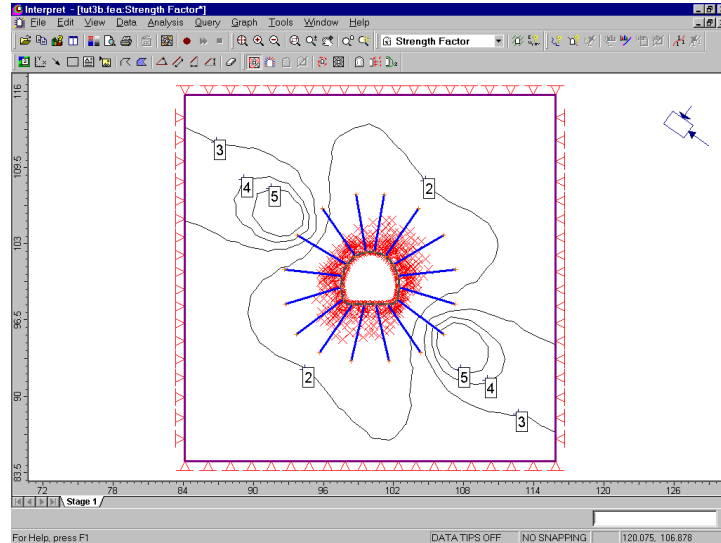
Figure 3-6: Strength factor contours and yielded points, for excavation with pattern bolt support only.

Let's check for yielding in the bolts. Select the Yielded Bolts button in the Display toolbar. The yielded bolt elements will be highlighted in red, and the number of yielded elements will be displayed in the status bar.

```
229 Yielded bolt elements
```

Almost all of the bolts have yielded, as shown by the bolt sections highlighted in red. Remember that "bolt elements", for fully bonded bolts, are defined by the intersections of bolts with the finite elements. (Bolt elements can be displayed with the Display Options dialog. This is left as an optional step.)

Recall that when we defined the bolt properties, we entered a residual bolt capacity equal to the peak bolt capacity. Therefore, even though the bolts have reached their yield capacity, they still provide support.

Finally, let's look at the effect of the bolts on the displacement.

Select: **Total Displacement** ▼

The maximum displacement indicated in the status bar is now:

```
Maximum Total Displacement = 0.0802 m
```

Compared to the unsupported excavation, the displacements have been slightly reduced, but not by much. (Maximum unsupported displacement = .0960 m). We'll have to add shotcrete to really support this tunnel.

Model

From INTERPRET, switch back to MODEL:

Select: File → Model

Now we add the liner support.

(Note: the finished product of the modeling section of this tutorial, with bolts and shotcrete, but before the load splitting, can be found in the "tut3b.fea" data file located in the EXAMPLES folder in your PHASE^2 installation folder, if you wish to skip the model process.)

Adding a Liner

We will now line the tunnel with shotcrete. First let's zoom in so that we can see what we are doing.

Select: View → Zoom → Zoom Excavation

Select: Support → Add Liner

```
Select segment(s) to add liner [enter=done,
esc=quit]:
```

To line the entire tunnel, use the following steps:

1. Click and hold the left mouse button, and drag a selection window which encloses the entire excavation. Release the left mouse button. Notice that all excavation line segments are selected.

2. Right-click the mouse and select Done Selection, or just press the Enter key. The entire tunnel will now be lined, as indicated by the thick blue line segments around the excavation boundary.

Anytime you want to line multiple adjacent boundary segments, it is always best to use a selection window, to ensure all desired segments are selected.

Liner Properties

Now define the liner properties. The properties we enter will correspond to a 200 mm thick layer of steel fibre reinforced shotcrete.

Select: Properties → Define Liners

> ✓ *Enter:*
> ✓ Name = shotcrete
> ✓ Thickness = 0.2
> ✓ Beam Form. = Bernoulli
> ✓ Young's Modulus = 3000
> ✓ Poisson's Ratio = 0.25
> ✓ Material Type = Plastic
>   Comp. Str. (peak) = 35
>   Comp. Str. (res) = 5
>   Tens. Str. (peak) = 5
>   Tens. Str. (res) = 0

Enter the liner properties with the first tab selected.

Since you entered the liner properties with the first tab selected, you do not have to Assign these properties to the liner – PHASE[2] will automatically assign the properties for you.

We are now ready to re-run the analysis.

Compute

Before you analyze your model, let's save this as a new file called **support4.fea**. (Make sure you select Save As and not Save, or you will overwrite the support3.fea file).

Select: File → Save As

Save the file as support4.fea.

Select: File → Compute

The PHASE^2 COMPUTE engine will proceed in running the analysis. When completed, you will be ready to view the results in INTERPRET.

Interpret

From MODEL, switch back to INTERPRET:

Select: File → Interpret

Now let's see how the addition of the shotcrete liner affected the strength factor and yielding.

Select: Strength Factor

The extent of the region encompassed by the contour of strength factor = 2 is now considerably reduced compared with Figures 3-4 and 3-6. Let's do a direct comparison of the three files on the same screen, as described below.

1. If you have been following the Support tutorial from the beginning, then you should have four files open in INTERPRET — support1, support2, support3 and support4. If this is the case, then CLOSE the support1 file, and leave the other three open.

2. If you have been closing files as you went along, then re-open the support2 and support3 files.

3. Now tile the three views (use the Tile Vertically button in the toolbar).

4. Display the Strength Factor in each view.

5. Select Zoom Excavation in each view (use the F8 key).

6. Select Zoom Out approximately 5 or 6 times, in each view (use the F4 key).

7. Display the yielded elements in each view.

8. If the legends are displayed, toggle them off in the View menu for each view.

9. Your screen should appear as below.

<table>
<tr><td><u>Strength Factor</u></td></tr>
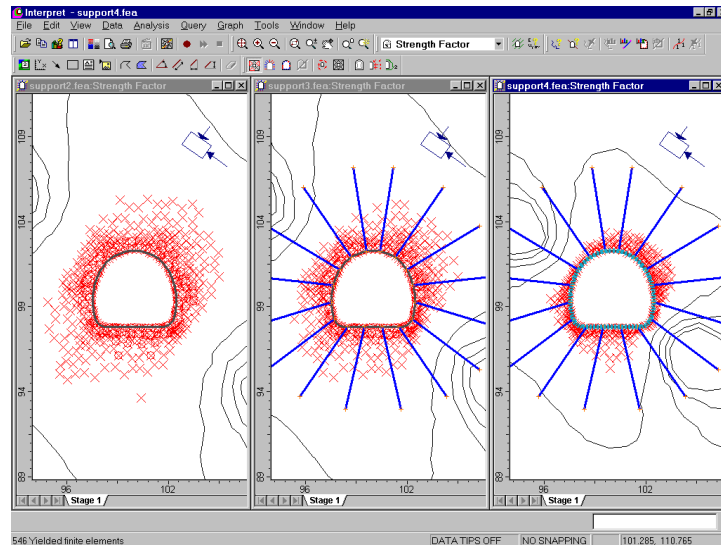<tr><td>Unsupported (left)
Bolts only (middle)
Bolts + Shotcrete (right)</td></tr>
</table>

Figure 3-7: Strength factor contours and yielded elements, for support2, support3 and support4 files.

Note – If the order of the three views is not as shown in the above figure, then click consecutively in the support4, support3 and support2 views, and re-tile the view.

Observe the effect of support on the strength factor contours, and the yielded element zone. It can be seen that the pattern bolting alone did not have much effect, but the application of a shotcrete liner, in conjunction with the pattern bolting, has been effective in reducing failure around the tunnel.

The number of yielded finite elements for each model is summarized below (the number of yielded elements is displayed in the status bar whenever the yielded elements are displayed):

FILE	# of yielded finite elements
Support1 (elastic analyis)	0
Support2 (plastic, no support)	583
Support3 (bolts only)	546
Support4 (bolts + shotcrete)	344

Now maximize the view of the support4 file, so that we can view it full screen again.

Toggle the display of yielded elements off, by re-selecting the Yielded Elements button in the Display toolbar.

Let's check for yielding in the bolts. Select the Yielded Bolts button in the Display toolbar.

Again, most of the bolts have yielded, as shown by the bolt sections highlighted in red. The status bar indicates the total number of yielded bolt elements.

```
189 Yielded bolt elements
```

The number of yielded bolt elements was decreased from 229 to 190 by the presence of the liner. Toggle off the display of yielded bolt elements by re-selecting the Yielded Bolts button in the Display toolbar.

We can look at yielding in the liner in the same manner as yielding in the bolts is displayed. Select the Yielded Liners button in the Display toolbar. The status bar will indicate:

```
28 Yielded liner elements
```

Zoom in so that you can see the yielded liner elements.

Select: View → Zoom → Zoom Excavation

The yielded liner elements, highlighted in red, are concentrated at the upper right, lower left, and floor of the tunnel. Toggle the yielded liner elements off by re-selecting the Yielded Liners button in the Display toolbar.

Finally, let's look at the displacements after adding the liner.

Select: Total Displacement ▼

The maximum total displacement indicated in the status bar is now:

```
Maximum Total Displacement = 0.0508 m
```

The combination of bolts and shotcrete has reduced the maximum displacement to about half of the unsupported value (0.096 m). Toggle the deformation vectors on.

Select: View → Display Options

In the Display Options dialog, toggle Deformation Vectors on, enter a Scale Factor of 10, and select Done

As can be seen from the contours and the displacement vectors, the maximum displacement is still occurring in the floor of the tunnel. This suggests the casting of a thicker concrete slab on the tunnel floor, however, we will not be exploring this further in this tutorial. You may want to experiment with changing the thickness of the liner on the floor of the tunnel (to say, 300 mm), after completing the rest of this tutorial.

It is left as an optional exercise to display the Total Displacement contours and Deformation Vectors for the support2, support3 and support4 files, to obtain the figure below.
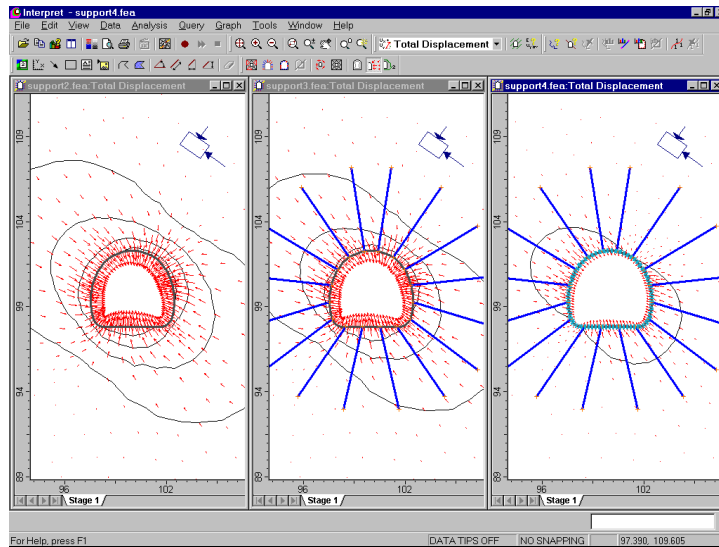

Figure 3-8: Total displacement contours and vectors for support2, support3 and support4 files.

Hints:

1. Tile and zoom the views as described earlier for the strength factor contours.

2. When displaying the deformation vectors for each view, use a Scale Factor of 10 (in the Display Options dialog).

3. In Contour Options, select Custom Range, enter a Range of 0 to 0.12, and Number of Intervals equal to 6, for each view. (The outermost contour shown in Figure 3-8 should then be the 0.02 meter total displacement contour).

Show Values

We will now demonstrate another very useful feature of PHASE², the Show Values option, which allows the user to display analysis results for bolts, liners and joints, graphically or numerically, directly on the model.

If you are viewing the results for multiple files, first maximize the view of the **support4** file. Press F8 to Zoom Excavation. Then select Show Values from the toolbar or the Analysis menu.

Select: Analysis → Show Values
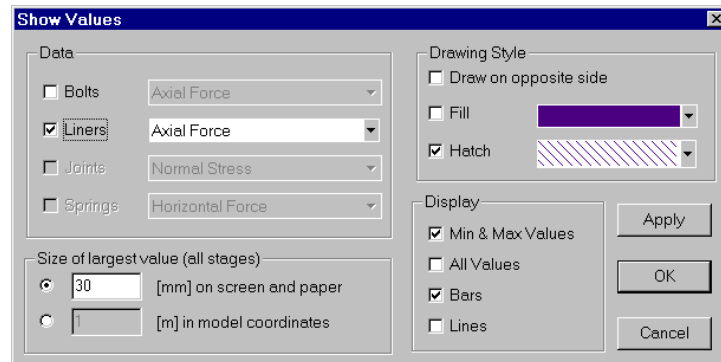
In the Show Values dialog, select the Liners checkbox, and Axial Force. Select other options in the dialog, as shown above. Select OK.

You should now see graphical "bars" representing the axial force in each liner element, displayed directly on the model, as shown in Figure 3-9. Notice that the Minimum and Maximum values of Axial Force are also displayed.
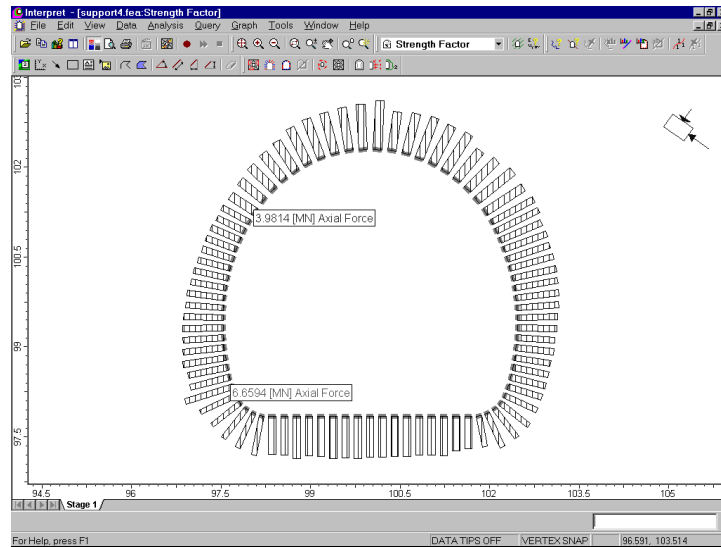
Figure 3-9: Axial force in liner displayed with Show Values option.

Note – the following additional steps were used to obtain the figure above:

1. In Contour Options, set the contour Mode to Off.

2. In Legend Options, turn the Legend Off.

3. In Display Options, turn Off Excavation boundaries and Bolts.

It is left as an optional exercise for the user to experiment with the many different display possibilities which are possible with the Show Values option.

As the final step in the Support tutorial, we will examine one more feature of PHASE2, called "load splitting", using this same example.

Model

If you are viewing multiple files in INTERPRET, make sure the **support4** view is selected, before you switch back to MODEL.

Select: File → Model

Load Splitting

Now we introduce the load splitting option.

The previous analyses in Step 2 of this tutorial (ie. pattern bolt support only, and combined pattern bolt/shotcrete support), implicitly assume that the support is installed *immediately* after excavation, and that no displacement takes place prior to the installation of support.

Of course this is not realistic, and a certain amount of deformation will always occur before the support can be installed. 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. Load splitting can therefore be used to simulate the delayed installation of support. In this simple example, it will be done as follows:

1. A staged model is required in order to enable Load Splitting, therefore we will first set the Number of Stages = 2 in Project Settings.

2. Using the Load Splitting option, the Load Split will be defined as 30% in Stage 1 and 70% in Stage 2.

3. The support (bolts and liner) will then be installed in Stage 2, rather than Stage 1.

Effectively, this allows some deformation to take place in Stage 1 (before support is installed), and then the support installed in Stage 2 can respond to the remainder of the field stress induced load.

The first step towards including load splitting in a model, is to set the Number of Stages in Project Settings.

Select: File → Project Settings

Set the Number of Stages = 2 and select OK.

Now we can enter the Load Split information.

Select: Loading → Load Split

In the Load Split dialog, select the Enable Load Split checkbox, and then enter Split Factor = 0.3 for Stage 1 and Split Factor = 0.7 for Stage 2. Select OK.

✓ *Enter:*

Project Name = (optional)

✓ Number of Stages = 2

Analysis = Plane strain

Max. # of iterations = 500

Tolerance = 0.001

Load Steps = Auto

Solver Type = Gauss. Elim.

The 0.3 / 0.7 load split assumes that 30 % of the field stress induced load has been relieved by displacement of the excavation boundaries before the support is installed. These load split factors, can be estimated from a plot such as Figure 6-5 in the PHASE² Axisymmetry Tutorial, based on how close the support can be installed to the advancing face of the tunnel.

Installing the Support

Now we have to install the support (bolts and liner) in Stage 2, and we will be finished with the modeling.

Select: Properties → Assign Properties

To install the support at Stage 2:

1. Make sure the Stage 2 tab is selected (at the bottom left of the view).

2. In the Assign dialog, select Bolts from the list at the bottom of the dialog, and then select the Install button.

3. Press F2 to Zoom All (if the model is not fully zoomed).

4. Use the mouse to click and drag a selection window which encloses all of the bolts in the model.

5. The bolts should now be selected. Right-click the mouse and select Done Selection, or press Enter. The bolts are now installed in Stage 2.

6. Now select Liners from the list in the Assign dialog, and select the Install button.

7. Press F8 to Zoom Excavation.

8. Use the mouse to click and drag a window which encloses the entire excavation. All of the liner elements on the excavation boundary should now be selected.

9. Right-click the mouse and select Done Selection, or press Enter. The entire liner is now installed in Stage 2.

10. Close the Assign dialog by selecting the X in the dialog, or press Escape twice (once to exit the Install mode, and once to close the dialog).

Now verify the staging of the support. Select the Stage 1 tab. The bolts and liner should appear in a light blue colour, indicating that they are NOT INSTALLED in Stage 1. Select the Stage 2 tab. The bolts and liner should appear in the dark blue colour, indicating that they are installed in Stage 2.

Now we will re-run the analysis.

Compute

Before you analyze your model, let's save this as a new file called **support5.fea**. (Make sure you select Save As and not Save, or you will overwrite the support4.fea file).

Select: File → Save As

Save the file as support5.fea.

Select: File → Compute

The PHASE² COMPUTE engine will proceed in running the analysis. When completed, you will be ready to view the results in INTERPRET.

Interpret

From MODEL, switch back to INTERPRET:

Select: File → Interpret

We will now see what effect the load splitting had on the results of the analysis. Select the Stage 2 tab, since we want to view the final stage results, after the installation of the support.

Let's first look at the strength factor.

Select: 🎨 Strength Factor ▼

Toggle the display of yielded elements on by selecting the Yielded Elements button in the Display toolbar. The number of yielded elements is now:

```
414 Yielded finite elements
```

This compares with 344 yielded finite elements before the load split.

It is left to the user to verify that the strength factor contours and yielded zone are essentially the same as prior to the load split. This is not surprising, as we did not expect the load split to have much effect on the strength factor. Toggle the display of yielded elements off, by re-selecting the Yielded Elements button in the Display toolbar.

Of more interest is the state of yielding in the bolts. Select the Yielded Bolts button in the Display toolbar. The status bar now indicates:

```
156 Yielded bolt elements
```

The number of yielded bolt elements is substantially reduced as a result of the load split, from 189 before the load split, to 156. *This is the primary result of interest from the load splitting analysis.*

Similarly, check the yielding in the liner, by selecting the Yielded Liner option in the Display toolbar. You should see that the number of yielded liner elements is only 4, compared to 28 before the load split. The load split has almost completely eliminated yielding in the liner.

Toggle the display of yielded bolts and liners off, by re-selecting the Yielded Bolts and Yielded Liners options.

Let's check the displacements.

Select: **Total Displacement** ▼

The maximum displacement shown in the status bar is now:

```
Maximum Total Displacement = 0.0514 m
```

This is almost identical to the maximum displacement before the load split, of .0508 m.

It is left to the user to verify that the displacement contours and deformation vectors are essentially the same as prior to the load split.

In summary, it should be emphasized that the primary effect of the load splitting, for this example, was to decrease the yielding in the bolts and liner, and thus improve the modeling of support. By allowing some unsupported deformation to take place in the first stage, we have traded off some increased yield of the rock mass, for decreased yield in the support.

Let's look at one more thing. Select the Stage 1 tab, and check the first stage displacements.

Select: **Total Displacement** ▾

The maximum displacement indicated in the status bar is:

```
Maximum Total Displacement = 0.0148 m
```

Compare this with the maximum displacement after the final (second) stage, which was 0.0514. Based on these numbers only, the proportion of the displacement taking place in the first stage is .0148 / .0514 = about 29%. This is roughly in agreement with our load split of 30 / 70. Better agreement than this should not be expected, since our analysis is plastic, and we are only comparing a single number. It is only mentioned to further illustrate the significance of using the load split option in PHASE[2].

Additional Exercise

One final exercise in the Support Tutorial will be left to the user to complete, if so desired.

Thus far we did not discuss the nature of the bolts we installed, however, based on the modulus of 200,000 MPa we can infer that we have been using solid steel dowels. Let's change the bolt modulus to 75,000 MPa , as an estimated stiffness of a seven strand steel cable. Re-run the analysis, with this one change, keeping all other model parameters the same (as in the load split example).

You should find that the strength factor and displacement results are nearly identical as compared to the results using the 200,000 MPa bolts. The big difference is that bolt yielding has been greatly reduced, as summarized in the following table.

Bolt Modulus	Maximum Displacement	Number of Yielded Bolt Elements
200,000 MPa	0.0514 m	156
75,000 MPa	0.0520 m	80

The conclusion that could be suggested, is that solid steel dowels may be too stiff for this very weak and highly stressed rock mass. The high stiffness of the reinforcement is not compatible with the large plastic strains which occur near the excavation boundary and which result in overstressing of the dowel/grout bond. The less stiff cables provide an almost identical support load (in terms of the extent to which the plastic zone is restricted and the deformations are limited) to the grouted dowels, but the cable/grout bond is not overstressed to nearly the same extent as for the dowels.

Surface Excavation Tutorial
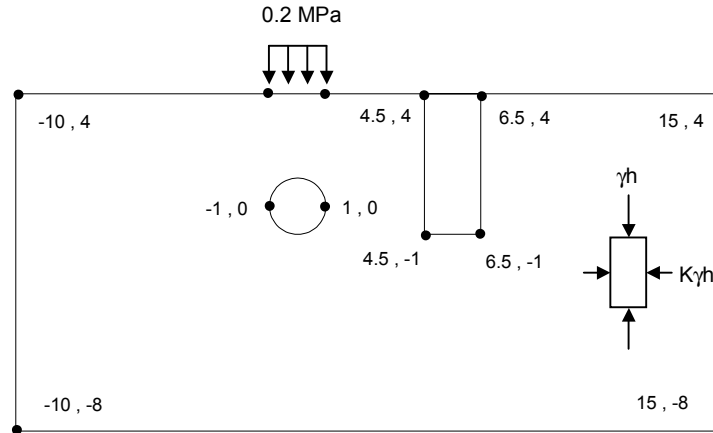
This tutorial illustrates how to model a simple surface excavation, consisting of a trench located near a circular tunnel, and a traction directly above the tunnel. The gravity field stress option will be used, and the analysis will be staged, by excavating the tunnel in the first stage, the trench in the second, and adding the traction in the third stage.

If you wish to skip the model process, the finished product of this tutorial can be found in the **tut4.fea** data file located in the EXAMPLES folder in your PHASE² installation folder.
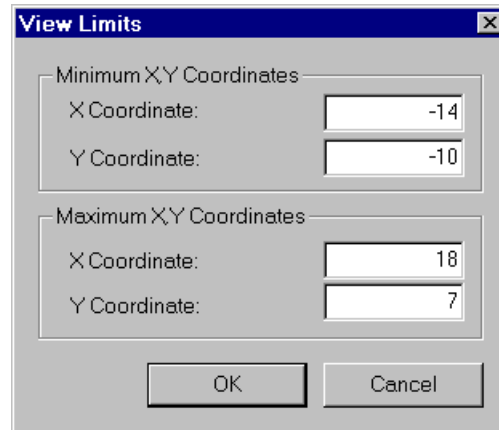
Model

If you have not already done so, run the PHASE² MODEL program by double-clicking on the PHASE² icon in your installation folder. Or from the Start menu, select Programs → Rocscience → Phase2 → Phase2.

Limits

Let's first set the drawing limits of the region so that the limits encompass the model geometry.

Select: View → Limits

Enter the following coordinates in the View Limits dialog, and select OK.

View Limits

Minimum X,Y Coordinates

X Coordinate: -14

Y Coordinate: -10

Maximum X,Y Coordinates

X Coordinate: 18

Y Coordinate: 7

OK   Cancel

These limits will approximately center the model in the drawing region, when you enter it as described below.
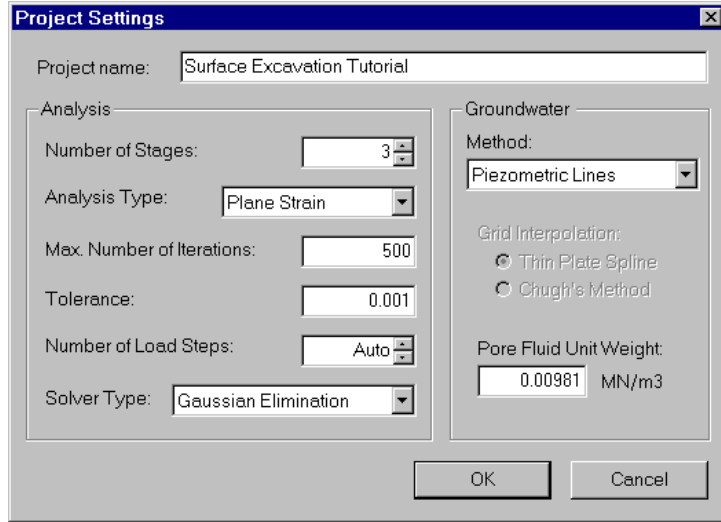
Project Settings

Whenever we are creating a staged model, the first thing we should always do is set the Number of Stages in Project Settings.

Select: File → Project Settings

In the Project Settings dialog, enter Number of Stages = 3, and select OK.

Entering Boundaries

First enter the circular tunnel as follows:

Select: Boundaries → Add Excavation

```
Enter vertex [a=arc,esc=quit]: a
```

By entering "a" at this first prompt, you can enter a circle at the following prompts:

```
Number of segments in arc <20>: enter 60
Enter first arc point[esc=quit]: -1 0
Enter second arc point[u=undo, esc=quit]: 1 0
Enter third arc point[u=undo, esc=quit]: c
```

By entering "c" at the last prompt, the arc closes on itself and forms a circle, with a diameter determined by the distance between the first and second points, in this case 2 meters.

Sidebar note:

✓ *Enter:*

  Project Name = (optional)

✓ Number of Stages = 3

  Analysis = Plane strain

  Max. # of iterations = 500

  Tolerance = 0.001

  # Load Steps = Auto

  Solver Type = Gauss. Elim.

Now enter the rectangular trench.

Select: Boundaries → Add Excavation

```
Enter vertex[a=arc,esc=quit]: 4.5 4
Enter vertex[a=arc,u=undo,esc=quit]: 4.5 -1
Enter vertex[a=arc,u=undo,esc=quit]: 6.5 -1
Enter vertex[a=arc,c=close,u=undo,esc=quit]: 6.5 4
Enter vertex[a=arc,c=close,u=undo,esc=quit]: c
```

Now enter the external boundary.

Select: Boundaries → Add External

✓ *Enter:*

✓ Bndry Type = User Defined

This model will require a user-defined external boundary, so in the above dialog, set the Boundary Type to User-Defined, and select OK. You will then be able to enter the external boundary coordinates in the same manner as entering the excavations.

```
Enter vertex[esc=quit]: 15 4
Enter vertex[u=undo,esc=quit]: 6.5 4
Enter vertex[u=undo,esc=quit]: 4.5 4
Enter vertex[c=close,u=undo,esc=quit]: 1 4
Enter vertex[c=close,u=undo,esc=quit]: -1 4
Enter vertex[c=close,u=undo,esc=quit]: -10 4
Enter vertex[c=close,u=undo,esc=quit]: -10 -8
Enter vertex[c=close,u=undo,esc=quit]: 15 -8
Enter vertex[c=close,u=undo,esc=quit]: c
```

Notice that on the upper surface of the external boundary (ie. the ground surface), we defined a line segment identical to the upper segment of the trench, and also a segment directly above the circular tunnel, which will be used when we define the traction.

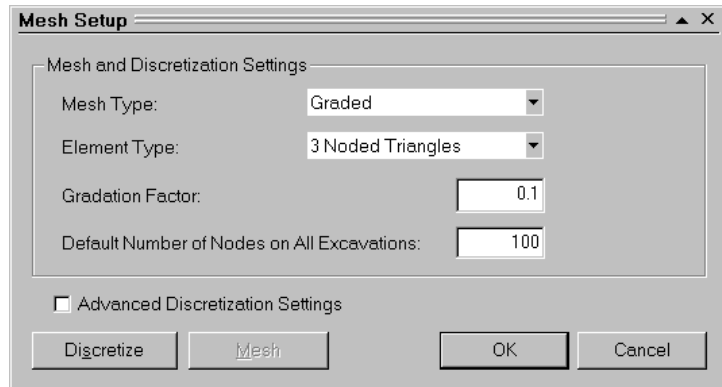We are done entering boundaries, so let's proceed to the meshing.

Meshing

As usual, we will discretize and mesh the model. However, we will first configure the # of Excavation Nodes in Mesh Setup, so that we get a finer mesh around the excavations.

Select: Mesh → Setup

✓ Enter:

  Mesh Type = Graded

  Elem. Type = 3 Noded Tri.

  Gradation Factor = 0.1

✓ # Excavation Nodes = 100

Mesh Setup

Mesh and Discretization Settings

Mesh Type:	Graded
Element Type:	3 Noded Triangles
Gradation Factor:	0.1
Default Number of Nodes on All Excavations:	100

☐ Advanced Discretization Settings

Discretize   Mesh   OK   Cancel

Enter Number of Excavation Nodes = 100 and select OK.

Now select discretize.

Select: Mesh → Discretize

The model is discretized, and the status bar will indicate the actual number of discretizations created.

```
Discretizations: Excavation=130, External=75
```

Note that:

- The number of excavation discretizations is 130, but we entered 100 in the Mesh Setup dialog. Depending on your excavation geometry, the discretization algorithm will not always give you exactly the #Excavation Nodes you entered in Mesh Setup.

- The top segment of the trench and the corresponding segment of the external boundary were automatically given the same discretization.

- Notice that PHASE^2 automatically grades the discretization on the external boundary, according to the distance from excavation boundaries. The discretization on the ground surface is finer near the top of the trench, and is gradually graded more coarsely towards the left and right edges of the model. The discretization along the left, right and bottom edges of the external boundary, is much coarser than along the top edge near the excavations.

Now let's generate the mesh.

Mesh

Select the Mesh option within the Mesh menu, and the mesh will be generated, based on the discretization you just created.

Select: Mesh → Mesh

The status bar will indicate the total number of elements and nodes in the mesh.

```
ELEMENTS = 1747 NODES = 912
```

Note that the automatically graded discretization along the ground surface helps to create a smooth transition between the fine mesh at the top of the trench, and the rest of the ground surface.

Boundary Conditions

By default, when the mesh is generated, all nodes on the external boundary are given a fixed, zero displacement boundary condition. This is indicated by the triangular "pin" symbols which you can see at each node of the external boundary.

Since this is a surface excavation model, we must specify that the ground surface is a free surface. This is done using the Free option in the Restraints menu.

Select: Restraints → Free

```
Select segments to free [enter=done, esc=quit]:
Use the mouse to select the five segments
representing the ground surface. When finished,
right-click and select Done Selection, or press
Enter.
```

The triangular pin symbols should now be gone from the ground surface indicating that it is free to move without restraint.
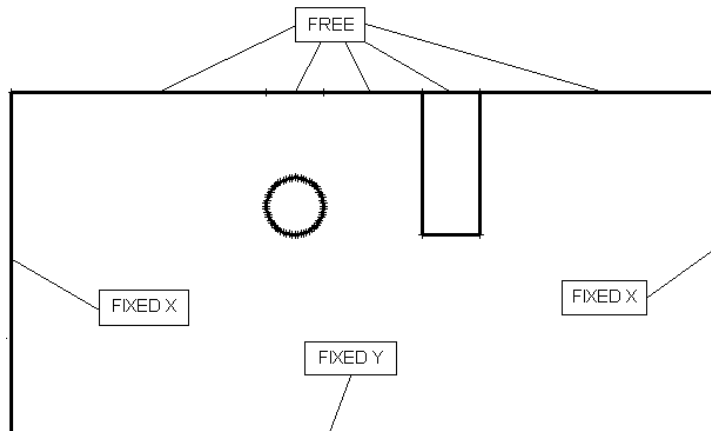

Figure 4-2: Displacement boundary conditions on external boundary.

Let's now specify the left and right edges of the external boundary as fixed in the X direction only (ie. free to move in the Y direction) and the lower edge as fixed in the Y direction only (ie. free to move in the X direction).

Select: Restraints → Restrain X

```
Select segments to restrain in the X direction
[enter=done,esc=quit]: Use the mouse to select the
left and right edges of the external boundary.
Right-click and select Done Selection, or press
Enter.
```

Select: Restraints → Restrain Y

```
Select segments to restrain in the Y direction
[enter=done,esc=quit]: Use the mouse to select the
bottom edge of the external boundary. Right-click
and select Done Selection, or press Enter.
```

Now we have some tidying up to do – the nodes at the bottom corners have rollers, and they should be pinned.

Select: Restraints → Restrain X,Y

1. Right-click the mouse and select Boundary Node from the popup menu. This will change the mode of restraint application from boundary segments to boundary nodes.

2. Select the lower left (-10 , -8) and lower right (15, -8) vertices of the external boundary. Triangular pin symbols should replace the roller symbols at these vertices.

3. Select Escape or right-click and select Cancel.

This brings us to an important point – after applying restraints to boundary segments, you should always check that nodes at the ends of segments have the correct conditions applied.

Adding a Traction

Now let's add the traction to the ground surface segment above the tunnel.

Select: Loading → Tractions → Horizontal / Vertical

Horizontal/Vertical Traction

Horizontal (MN/m, +ve right):   0

Vertical (MN/m, +ve down):   0.2

☑ Stage traction

Stage	1	2	3	
Factor	0	0	1	

OK   Cancel

✓ *Enter:*

Horizontal = 0

✓ Vertical = 0.2

✓ Stage Traction

In the Horizontal/Vertical Traction dialog, enter a Vertical Traction = 0.2 MN/m. Select the Stage Traction checkbox, and enter Factor = 0 for Stage 1 and Stage 2, and Factor = 1 for Stage 3. Select OK.

Because of the Factors we have defined, the traction will only be applied in the third stage of the analysis, and will not exist in the first or second stages. Factor = 1 means the magnitude will be the same as entered in the Traction dialog. Factor = 0 means no traction will be applied at that stage. Other values of Factor can be used to increase or decrease the load magnitude of the traction at any stage of the model.

Now select the external boundary line segments to be loaded:

```
Select segments to add traction [enter=done,
esc=quit] : use the mouse to select the external
boundary line segment directly above the circular
tunnel. Right-click and select Done Selection, or
press Enter.
```
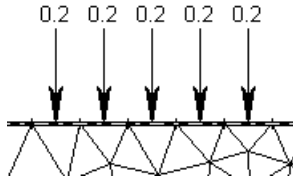
Viewing the Traction

To view the traction, select the Stage 3 tab. Since we only applied the traction in Stage 3, it is only visible in Stage 3. For display purposes, the size of the traction arrows can be scaled by the user in the Display Options dialog. This is left as an optional exercise. Since we are not finished modeling, select the Stage 1 tab again.

Field Stress

For most problems involving a ground surface, we will want to use a gravity stress field.

Select: Loading → Field Stress

Enter the above parameters and select OK. Note:

- The Ground Surface Elevation is indicated by a horizontal dotted gray line, and corresponds to the y-coordinate of the top surface of the external boundary (4 meters). The display of this line can be toggled on or off at any time in the View menu.

- For a gravity field stress, the stress block reflects the in-plane horizontal/vertical stress ratio, which in this case is 0.5.

✓ *Enter:*

✓ Fld. Str. Type = Gravity

✓ Ground Surf. Elev. = 4

✓ Unit Wt. Overburden = .02

✓ Str.Ratio (in-plane) = 0.5

✓ Str.Ratio(out-of-plane) =0.5

  Locked-in (in-plane) = 0

  Locked-in (out-of-plane) = 0

- The Unit Weight of Overburden indicates that our material is a soil, rather than rock.

Properties

Select: Properties → Define Materials

<div>

✓ *Enter:*

✓ Name = till
   In.El.Ld.=Fld Str & Bdy For.

✓ Unit Weight = 0.02
   Material Type = Isotropic

✓ Young's Modulus = 50

✓ Poisson's Ratio = 0.25
   Failure Crit. = Mohr Coul.
   Material Type = Elastic
   Tens. Strength = 0

✓ Fric. Angle (peak) = 38

✓ Cohesion (peak) = 0.01

</div>

With the first tab selected in the Define Material Properties dialog, enter the above properties.

Note the following:

- the Unit Weight we entered was the same as the Unit Weight of Overburden entered in the Field Stress dialog.

- The modulus and strength values we entered are those of a till with high frictional strength.

- Note that the default setting for 'Initial Element Loading' is now 'Field Stress and Body Force' – because we are dealing with a surface excavation and a gravitational stress field, the body force component of loading on each element is significant. (For a Constant stress field, the body force component is usually not considered, and the default 'Initial Element Loading' is 'Field Stress Only').

Since we defined the properties with the first material tab selected, we do not have to assign them to the model. By default, PHASE^2 automatically assigns the properties of the first material to all finite elements. We do however, have to assign the staging of the excavations.

Select: Properties → Assign Properties

We will excavate the tunnel in Stage 1, and the trench in Stage 2, as follows:
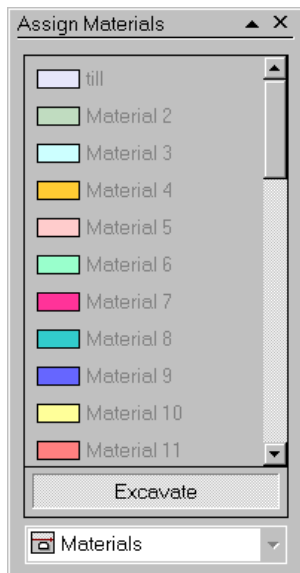
1. Make sure the Stage 1 tab is selected (at the bottom left of the view).

2. Select the "Excavate" button in the Assign dialog.

3. Click the left mouse button inside the circular tunnel. The elements in the tunnel will disappear, indicating that the tunnel is 'excavated'.

4. Select the Stage 2 tab.

5. Click the left mouse button inside the rectangular trench. The elements in the trench will disappear, indicating that it is 'excavated'.

That's all that is required. Close the Assign dialog by selecting the X in the upper right corner of the dialog.

As an optional step, it's always a good idea to verify the assignments by selecting each Stage tab in turn and inspecting the model.

Assign Materials

- till
- Material 2
- Material 3
- Material 4
- Material 5
- Material 6
- Material 7
- Material 8
- Material 9
- Material 10
- Material 11

Excavate

Materials

- Select Stage 1 – only the tunnel should be excavated.

- Select Stage 2 – both the trench and tunnel should be excavated.

- Select Stage 3 – the traction should now appear above the circular tunnel.

You can use the Stage tabs at any time to view the stages of your model, and verify material assignments, and excavation and support sequencing.

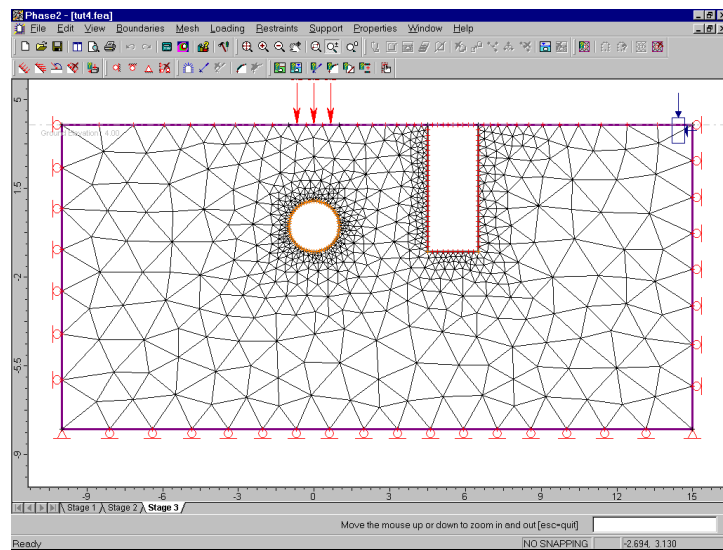We have now completed the modeling, your finished model should appear as shown below.

Figure 4-3: Finished model – PHASE^2 Surface Excavation Tutorial

Compute

Before you analyze your model, save it as a file called **surf1.fea**.

Select: File → Save

Use the Save As dialog to save the file. You are now ready to run the analysis.

Select: File → Compute

The PHASE^2 COMPUTE engine will proceed in running the analysis. When completed, you will be ready to view the results in INTERPRET.

Interpret

To view the results of the analysis:

Select: File → Interpret

This will start the PHASE^2 INTERPRET program.

Sigma 1

You are now viewing the Sigma 1 contours for Stage 1. Toggle the principal stress trajectories on by selecting the Stress Trajectories button in the Display toolbar.

As you can see, the gravitational stress field results in horizontal Sigma 1 contours, except where the contours are perturbed by the excavation. Overall, the major principal stress is vertical as can be seen by the 'long' axis of the stress trajectories – remember our horizontal / vertical stress ratio was 0.5 (in-plane and out-of-plane).

Now view the stress contours for Stage 2 and then Stage 3, by selecting the stage tabs at the lower left of the view.
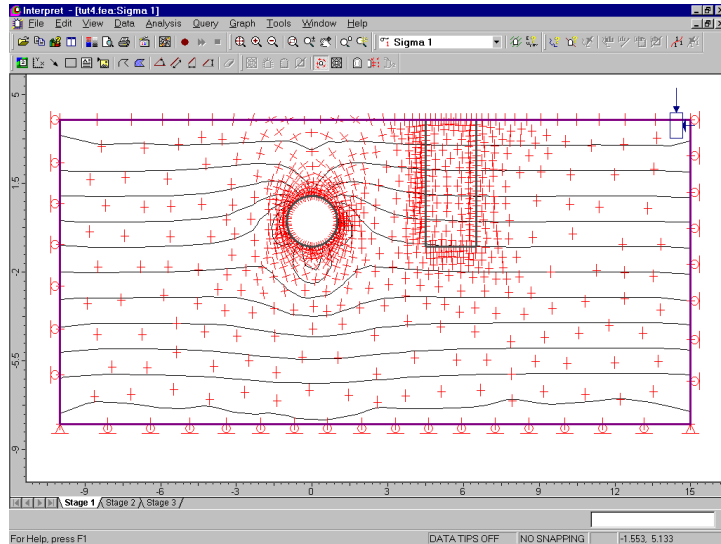
Figure 4-4: Sigma 1 contours and principal stress trajectories. Gravitational field stress is in effect.

Toggle the display of stress trajectories off by re-selecting the Stress Trajectories button in the Display toolbar.

Let's explore how we might view results for multiple stages on the same screen. The following procedure is just an example of how this could be done.

1. Select Window → New Window.

2. Select Window → Tile Horizontally.

3. Select the Stage 1 tab on the upper view and the Stage 2 tab on the lower view, and compare the contours.

4. Select the Stage 3 tab on the lower view, and compare the Stage 1 vs. Stage 3 contours.

5. Select the Stage 2 tab in the upper view, and compare the Stage 2 vs. Stage 3 contours.

When comparing contours of the same data (eg. Sigma 1) at different stages, always check that the ranges are the same. If not, use Contour Options to set the desired range.

Strength Factor

⚁ **Strength Factor** ▾

Now change the data type for both views to Strength Factor, and repeat steps 3 to 5 above.

Based on the Strength Factor contours at Stage 3, it is evident that this excavation would collapse without support. Keeping in mind that our analysis was elastic, notice the regions of failure around the tunnel, and between the trench and the tunnel (ie. contours with strength factor < 1).

Also notice the tensile failure along the ground surface, and above the tunnel. Remember we entered a tensile strength of zero for this material. The tensile failure along the ground surface could be thought of as a 'tension crack' layer. Tensile failure is indicated in PHASE^2 when Sigma 3 is less than the calculated negative stress (tension) cutoff for the failure criterion in use. Let's verify this.

σ₃ **Sigma 3** ▾

1. In the upper view, select the Stage 3 tab, and change the data type to Sigma 3.

2. Right-click the mouse in the upper view, and select Contour Options. In the Contour Options dialog, select Custom Range, and enter a Maximum Range of zero. Enter the Number (of Intervals) = 7, and select Apply.

3. Now click the mouse in the lower view (displaying Strength Factor contours). In the Contour Options dialog, enter the Number (of Intervals) = 7, and select Done.

4. Now observe that the contoured region of Sigma 3 < 0 corresponds to the "tension" zone of the Strength Factor contours, as shown in the following figure.
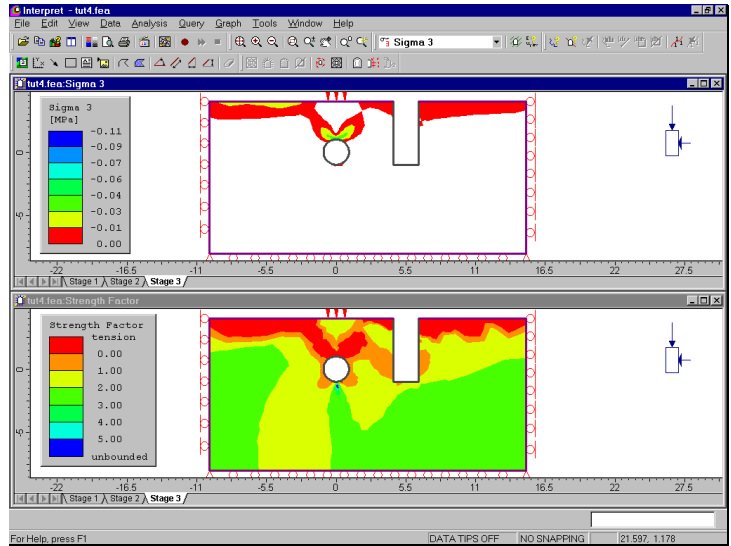
Figure 4-5: Comparison of Strength Factor and Sigma 3 contours at Stage 3.

This is simply an example of the data interpretation possibilities using multiple views. Many more possibilities will become apparent as you become familiar with the PHASE[2] INTERPRET program.

Displacement

Now let's look at displacements. First maximize one of the two views, it doesn't matter which one.

Select: **Total Displacement**

Press F2 to Zoom All. View the displacement contours at each stage, by selecting the stage tabs. Observe the maximum displacement, displayed in the status bar, and where it is occurring on the model.

The Stage 1 maximum displacement, about 3 mm, is occurring at the bottom of the tunnel. The Stage 2 maximum displacement about 4.5 mm, is occurring at the left side of the trench. The Stage 3 maximum displacement, about 17 mm, is underneath the traction.

Now let's turn off the display of the contours, and view the deformed shape of the boundaries and mesh, magnified by a factor of 100.

Toggling contours Off is useful when you wish to hide the contours and view other information, for example stress trajectories, deformation vectors or deformed boundaries.

1. Right-click the mouse and select Contour Options.

2. In the Contour Options dialog, set the Mode to Off, and select Done.

3. Right-click the mouse and select Display Options.

4. In the Display Options dialog, select Deform Mesh and Deform Boundaries, and enter a Scale Factor of 100. Select Done.

5. Select Zoom Excavation (in the toolbar, or press F8).
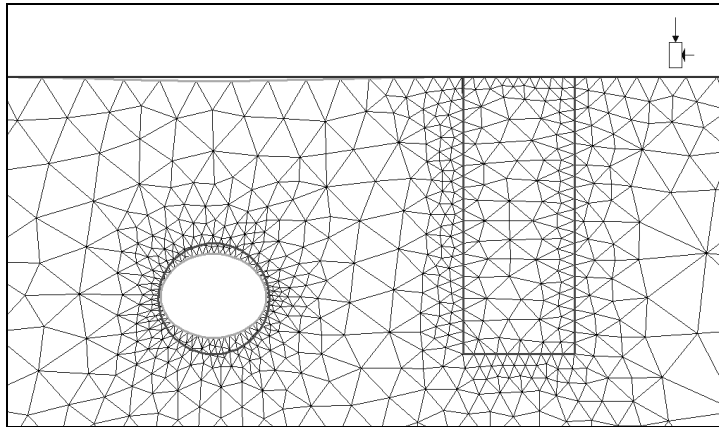
6. Select the Stage 1 tab.

Figure 4-6a: Deformed mesh and boundaries, stage 1. Displacements magnified by 100.

Notice the flattened shape of the circular tunnel, and the subsidence of the ground surface above the tunnel. If you look carefully, you will notice that the bottom of the tunnel has displaced slightly more than the top. This is due to the gravity stress field, which of course increases with depth.
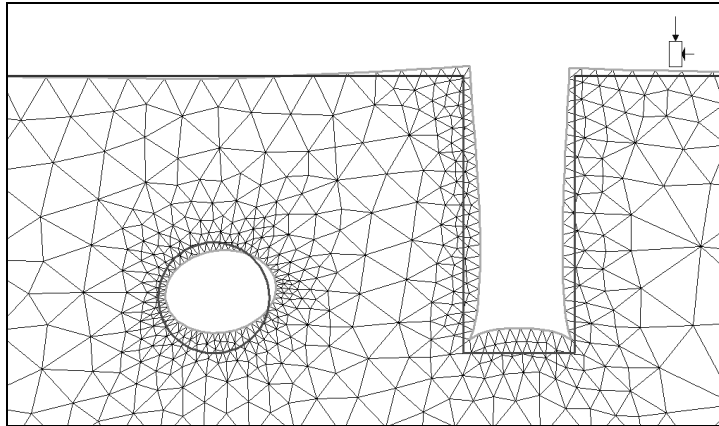
7. Select the Stage 2 tab.

Figure 4-6b: Deformed mesh and boundaries, stage 2. Displacements magnified by 100.

The deformation of the trench boundaries is clearly visible. Notice that the excavation of the trench has shifted the displacement of the tunnel towards the right.

8. Select the Stage 3 tab.

The displacements are now dominated by the effect of the traction. The maximum displacement is directly beneath the traction. The overall displacement of the tunnel has been shifted downward, and the bottom of the tunnel is now almost in its original position.
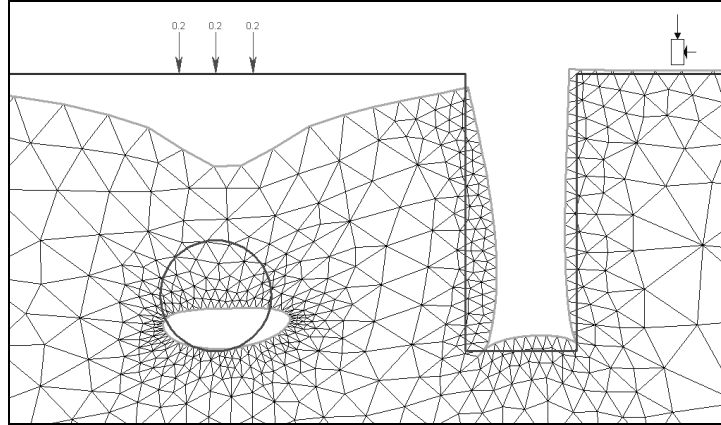
Figure 4-6c: Deformed mesh and boundaries, stage 3.
Displacements magnified by 100.

Let's "animate" the results.

Select: Data → Animate Stages

The stage tabs are now automatically selected for you,
giving you an animated display of results at each stage.
The timing of the animation can be set in the Data→Stage
Settings option.

To exit the animation mode, press Escape.

Before we move on:

• Display the contours again. Right-click the mouse and
  select Contour Options. Set the Mode to Filled and
  select Done.

• Also turn off the display of the Mesh and Deformed
  Boundaries, by selecting the corresponding buttons in
  the Display toolbar.

• If you are not already zoomed in, press F8 to Zoom
  Excavation

Query Data

PHASE² allows the user to query data anywhere in the rock mass, to obtain values interpolated from the contour plots. These values can be displayed directly on the model, or graphed. A query can be a single point, a line segment, or any arbitrary polyline.

The following steps will illustrate how to:

- Create a query, and show the values on the model

- Graph the query with data from multiple stages on the same plot

- Edit the query

Creating a Query

To create a query:

Select: Query → Add Material Query

1. It will be handy to use the Snap option in this case, so right-click the mouse and select Snap.

2. When you are in Snap mode, if the cursor is near a model vertex, a circle will appear around the vertex, indicating that if you click the mouse, you will "snap" exactly to the location of the vertex.

3. Use the mouse to select the vertex at (4.5 , 4) i.e the upper left corner of the trench.

4. Use the mouse to select the vertex at (4.5 , -1) i.e the lower left corner of the trench.

5. Right-click the mouse and select Done. You will see the following dialog:
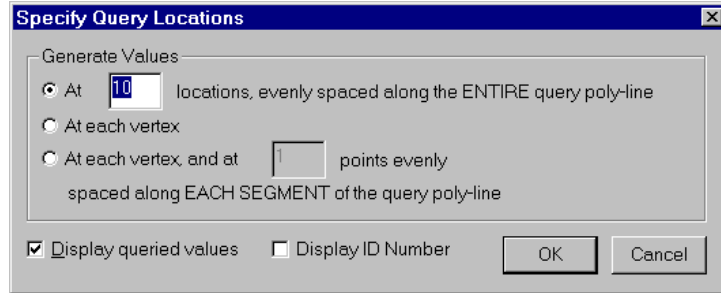
We will use the default values, so just select OK.

6. You should now see 10 values along the left edge of the trench, since we used the default value of 10 locations in the Specify Query locations dialog. (If you are not zoomed in, select Zoom Excavation to get a better view).

7. The values correspond to the stage and the data type you are viewing. Select the stage tabs, and observe the change in the values.

8. Select different data types (eg. Sigma 1, Strength Factor), and observe the change in values.

Number of Decimal Places Displayed

The number of decimal places used to display the query values, can be customized by the user in the Legend Options dialog.

1. If you changed data types as suggested above, switch back to viewing Total Displacement, at Stage 2.

2. If the Legend is currently displayed, right-click on the Legend and select Legend Options.

3. (If the Legend is NOT currently displayed, then select Legend Options from the View menu, and select the Show Legend checkbox in the Legend Options dialog.)

4. In the Legend Options dialog, select Number Format
    = Decimal, and use the mouse to change the number
    of decimal places (click on the up or down arrows).
    Notice that as the number of decimal places is
    changed, the display of values on the query, and also
    the interval values in the Legend, is immediately
    updated.

5. Set the number of decimal places to 4, and select OK
    in the Legend Options dialog.

Graphing a Query

A short cut for graphing data for a single query, is to
right-click on the query and select Graph Data.

1. Right click on the query (ie. anywhere along the left
    edge of the trench), and select Graph Data from the
    popup menu.

2. You will see the Graph Query Data dialog.
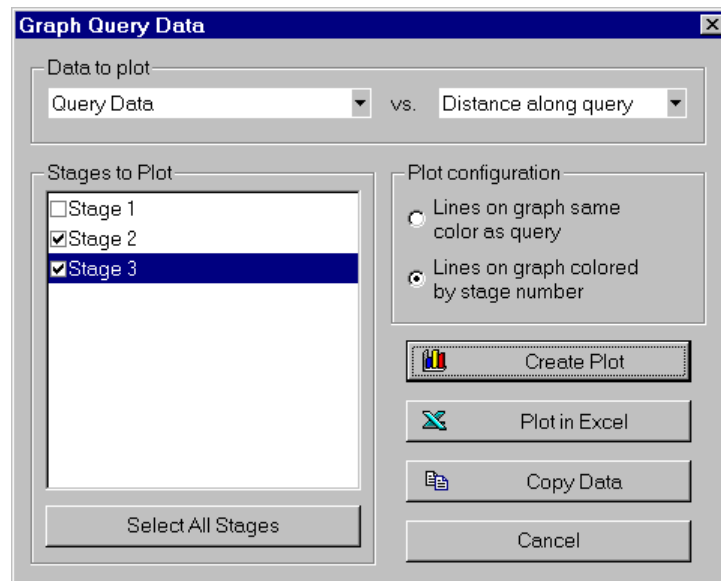
3. Select the Stages to Plot checkboxes for Stage 2 and
    Stage 3. Select the Create Plot button, and a graph of
    Total Displacement along the query, for both Stage 2
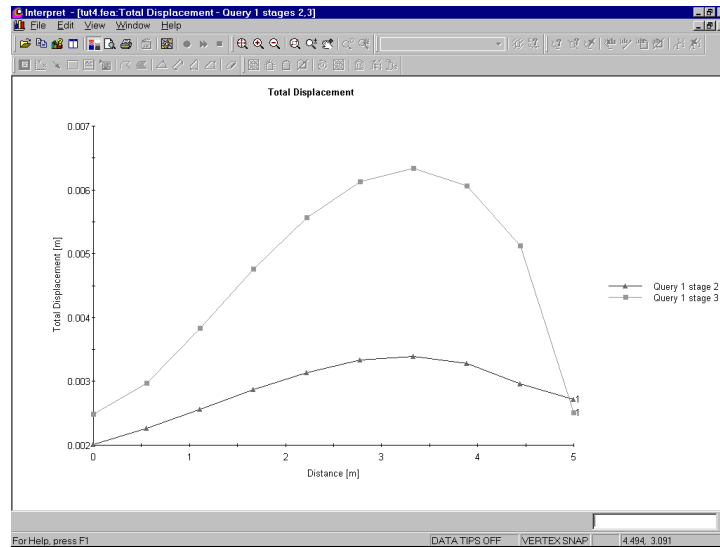   and Stage 3, will be generated.

Figure 4-7: Trench wall displacement, before and after adding
surface load.

This graph shows the "before" and "after" effect of the
surface load (traction) on the displacement of the trench
wall. The upper curve represents the Stage 3 results, and
the lower curve represents the Stage 2 results. The
maximum difference is about 3 mm, at about 3.3 meters
below the ground surface.

Note that each curve on the graph has 10 points. This is
because when we created the query, we only specified 10
locations at which to generate values, in the Specify
Query Locations dialog. We can change the number of
points to obtain a smoother graph. Close the graph, and
we will edit the query and generate a new graph.

Editing a Query

To edit the query:

1. Right click on the query, and select Edit from the popup menu.

2. You will see the Specify Query Locations dialog again. This time enter 50 as the number of locations. Also, toggle off the "Display queried values" checkbox. Select OK.

3. Notice the values are no longer displayed on the model. Since we are now querying at 50 locations, the numbers would not be readable without zooming in, so we decided to toggle them off.

4. Now repeat the steps outlined in the previous section (Graphing a Query) to obtain a new, smoother graph with 50 points on each displacement curve.

Axis ranges and titles can be modified by right-clicking on the graph and selecting Chart Properties.

Finally, note that the axis ranges and titles can be modified by right-clicking on the graph and selecting Chart Properties. That is left as an optional exercise for the user to complete.

Close the view of the graph by selecting the X in the upper right corner of the view.

That concludes this tutorial, further exercises based on this model are suggested below.

Additional Exercises

Single Stage Model

Although this tutorial model was set up as a three stage analysis, it could also have been set up as a single stage model. The staging was done for purposes of illustration, and to allow us to see the intermediate stage results.

As an exercise, re-do this problem as a single stage model. Hints:

• you do not have to explicitly define the trench as an excavation, it can be defined implicitly by the external boundary, as illustrated below.

• in the Mesh Setup option, use #Excavation Nodes = 60, since the only 'excavation' in this model (according to the PHASE^2 boundary definitions) is the circular tunnel, which has 60 segments. After you Discretize, you will have to do some Custom discretizing of the trench boundaries and adjacent segments, to obtain a mesh similar to the one shown below.
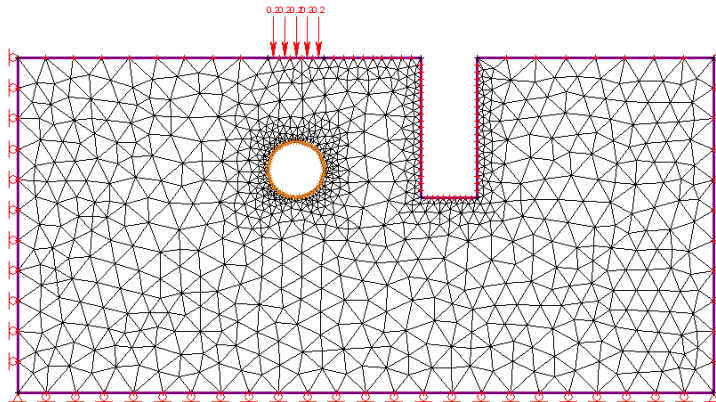

Figure 4-8: Surface excavation model, single stage version.

When you run the analysis, the results should be virtually identical to the third stage results presented in this tutorial, since the analysis in both cases is elastic. If we were doing a plastic analysis, this would not necessarily be the case.

Adding a Liner

As another exercise, add a liner to the entire circular tunnel boundary, and re-run the analysis. Use the default liner properties (ie. an elastic liner, 0.1 m thickness, modulus = 30000 Mpa). For details about adding liners to excavations, see the Support Tutorial, Step 2, in this manual.

When you look at the analysis results, notice that the zone of tension around the tunnel is reduced in the Strength Factor contour plot, and also the displacements are reduced.

(You can add the liner to the single stage model described above, or you can add it to the staged model, in which case you can experiment with installing the liner at different stages).

External Boundary Distance from Excavations

If you go back and examine the contour plots (Sigma 1, Strength Factor, Displacement) in this tutorial, you will see that the external boundary is actually influencing the contours, especially at the left edge and bottom edge of the boundary. For example, the Sigma 1 contours at the lower edge of the external boundary should be horizontal for a gravity stress field, but they are not. This tells us that the external boundary is too close to the excavations, and is actually restricting movement.

Re-do the staged analysis with a larger external boundary. For example, in the following figure, the left, right and bottom edges are located at x = –20, x = 25, and y = –18, respectively.
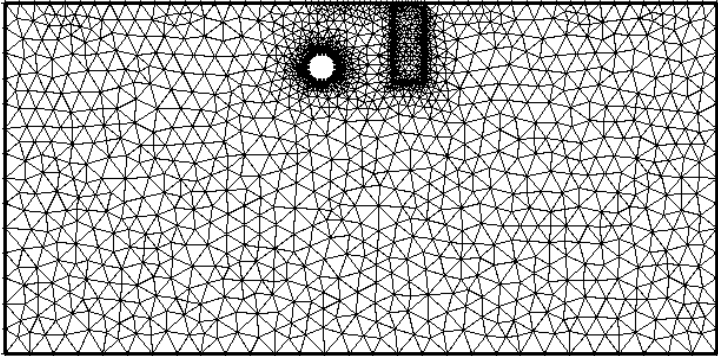

Figure 4-9: Modified external boundary for surface excavation tutorial.

You will find that:

- The Sigma 1 contours become horizontal at a certain distance below the excavations, which is appropriate for a gravity stress field.

- The maximum displacements are greater at each stage, as indicated in the table below.

	ORIGINAL BOUNDARY	EXTENDED BOUNDARY
Stage 1	.0030	.0035
Stage 2	.0045	.0062
Stage 3	.0169	.0185

Table 4-1: Maximum displacement (m) at each stage, for original and extended external boundaries.

In general, the extended boundary is far enough away to better simulate 'infinite' conditions, and should no longer influence the results near the excavations.

Joint Tutorial
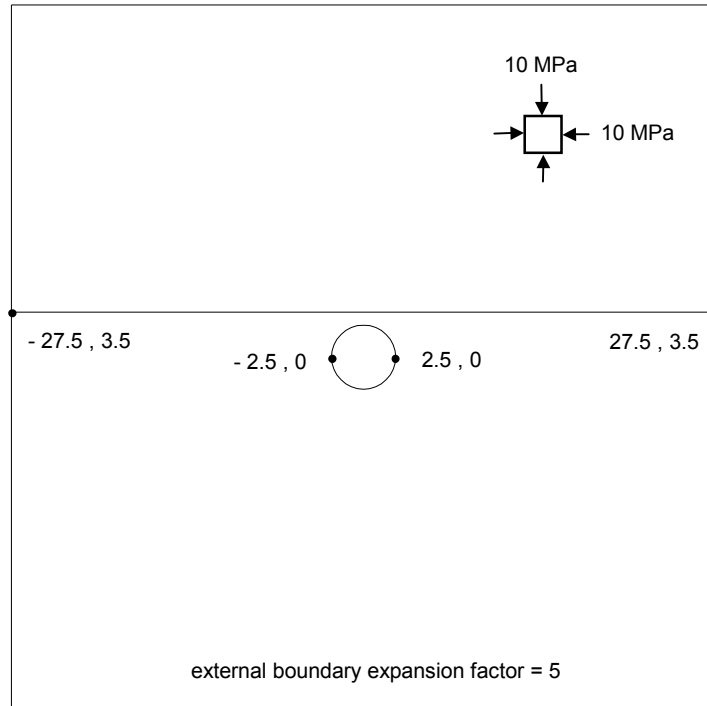
This tutorial involves a circular opening of 2.5 meter radius, to be excavated close to a horizontal plane of weakness (joint), located 3.5 meters above the center of the circular opening.

For this analysis, the rock mass is assumed to be elastic, but the joint will be allowed to slip, illustrating the effect of a plane of weakness on the elastic stress distribution near an opening. (This example is based on the one presented on pg. 193 of Brady and Brown, *Rock Mechanics for Underground Mining*, 1985 – consult this reference for further information.)

The finished product of this tutorial can be found in the **tut5.fea** data file located in the EXAMPLES folder in your PHASE² installation folder.

Model

If you have not already done so, start the PHASE² MODEL program by selecting Programs → Rocscience → Phase2 → Phase2 from the Start menu.

Limits

Let's first set the drawing limits of the region so that the limits encompass the model geometry.

Select: View → Limits

Enter the following coordinates in the View Limits dialog, and select OK.
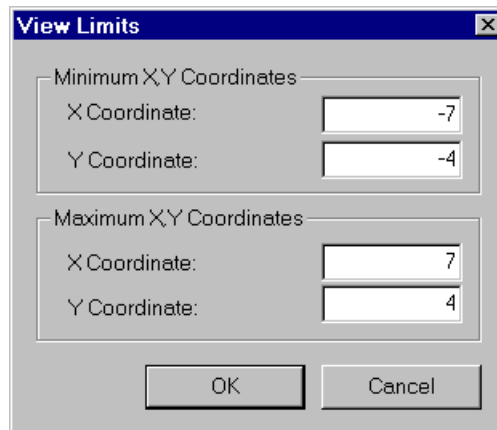
These limits will approximately center the circular opening in the drawing region, when you enter it as described below.

Entering Boundaries

First create the circular excavation as follows:

Select: Boundaries → Add Excavation

```
Enter vertex [a=arc,esc=quit]: a
```

By entering "a" at this first prompt, you can easily enter a circle (or arc) at the following prompts:

```
Number of segments in arc <20>: enter 32
Enter first arc point[esc=quit]: 2.5 0
Enter second arc point[u=undo, esc=quit]: -2.5 0
Enter third arc point[u=undo, esc=quit]: c
```

By entering "c" at the last prompt, the arc closes on itself and forms a circle, with a diameter determined by the distance between the first and second points, in this case 5 (meters).

The circular excavation has been entered, so you can now add the external boundary.

Select: Boundaries → Add External

✓ *Enter:*

Boundary Type = Box

✓ Expansion Factor = 5

Create External Boundary

Boundary Type: Box

Expansion Factor: 5

OK   Cancel
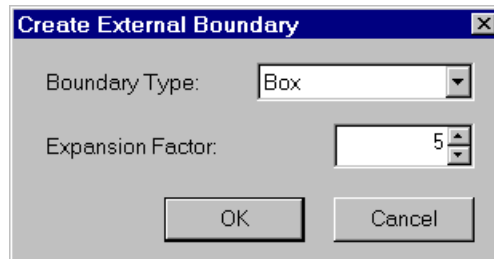
Enter an Expansion Factor of 5, and select OK, and the external boundary will be automatically created.

You can now add the joint to the model:

Select: Boundaries → Add Joint

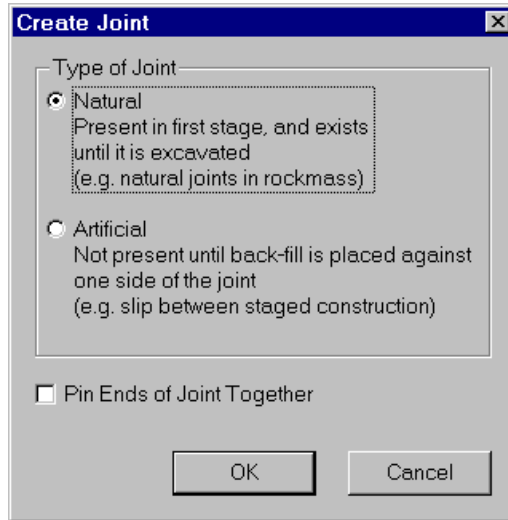Whenever you select Add Joint, you will first see the Create Joint dialog. This allows you to define a joint as either Natural or Artificial. We are modeling a Natural joint in the rockmass, so just select OK in this dialog. NOTE: see the PHASE² Help system for a discussion of Natural and Artificial joints, as they are defined in PHASE².

Now enter the following coordinates defining the joint.

```
Enter vertex [esc=quit]: -30 3.5
Enter vertex [u=undo,esc=quit]: 30 3.5
Enter vertex [enter=done,u=undo,esc=quit]: press
Enter
```

PHASE² automatically intersects boundaries and adds vertices when required.

Notice that the two points defining the joint were actually entered just outside of the external boundary, and PHASE² automatically intersected the boundaries and added new vertices. This capability of PHASE² is very useful, for example when:

• the user does not know the exact intersection of two lines, the automatic intersection capability of PHASE² saves the user the trouble of having to calculate such intersections, or when

- new vertices are required at known locations, they can be created automatically (rather than manually with the Add Vertices option).

Note: you could have entered (-27.5, 3.5) and (27.5,3.5) at the above prompts (ie. points "exactly" on the external boundary) and achieved the same result. However, to be on the safe side, we entered points slightly beyond the boundary, to ensure intersection between the newly entered joint boundary, and the existing external boundary.

All boundaries have now been entered, so we can go ahead and mesh the model.

Meshing

We will now proceed to generate the finite element mesh. First let's customize the Number of Excavation nodes in Mesh Setup.

Select: Mesh → Setup

Mesh Setup

Mesh and Discretization Settings

Mesh Type:	Graded
Element Type:	3 Noded Triangles
Gradation Factor:	0.1
Default Number of Nodes on All Excavations:	64

☐ Advanced Discretization Settings

Discretize   Mesh   OK   Cancel
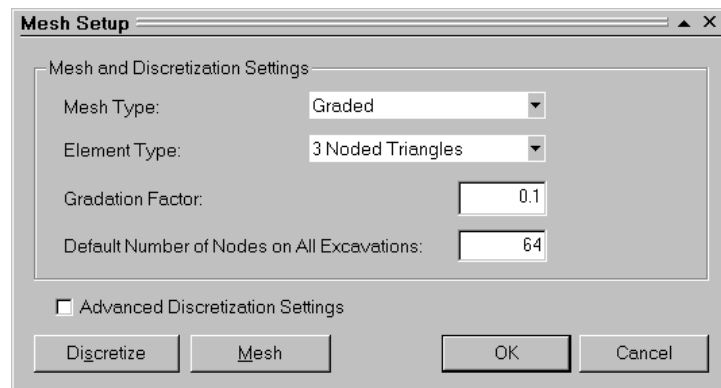
In the Mesh Setup dialog, enter Number of Excavation Nodes = 64. Select OK.

Now discretize the boundaries.

Select: Mesh → Discretize

This will automatically discretize all of the model boundaries. The discretization forms the framework for the finite element mesh. Notice the summary of discretization shown in the status bar, indicating the number of discretizations for each boundary type.

```
Discretizations: Excavation=64, External=112,
Joint=75
```

Note that the # of excavation discretizations = 64, which is exactly what we entered in the Mesh Setup dialog, and is twice the number of line segments entered for the circular excavation. Therefore each line segment of the excavation will have two finite elements on it when the mesh is generated.

Now select the Mesh option within the Mesh menu, to generate the finite element mesh.

Select: Mesh → Mesh

The finite element mesh is generated, with no further intervention by the user. When finished, the status bar will indicate the number of elements and nodes in the mesh:

```
ELEMENTS = 3256 NODES = 1761
```
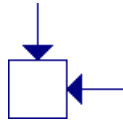
If you have followed the steps correctly so far, you should get exactly the same number of nodes and elements as indicated above.

Boundary Conditions

For this tutorial, no boundary conditions need to be specified by the user, therefore the default boundary condition will be in effect, which is a fixed (ie. zero displacement) condition for the external boundary.

Field Stress

We will be using the default field stress for this model, which is a constant hydrostatic stress field with Sigma 1 = Sigma 3 = Sigma Z = 10 Mpa. Therefore you do not have to enter any field stress parameters, the values we want are already in effect.

Properties

The properties of the rockmass and the joint must now be entered.

Select: Properties → Define Materials

✓ *Enter:*

  Name = Material 1
  Init.El.Ld.=Fld Stress Only
  Material Type = Isotropic
  Young's Modulus = 20000
✓ Poisson's Ratio = 0.25
  Failure Crit. = Mohr Coul.
  Material Type = Elastic
  Tens. Strength = 0
  Fric. Angle (peak) = 35
  Cohesion (peak) = 10.5

With the first tab selected in the Define Material Properties dialog, enter the above properties (only a Poisson's ratio of 0.25 needs to be entered, all other properties should be at the correct values). Select OK.

You have just defined the rockmass properties, now do the same for the joint properties.

Select: Properties → Define Joints

✓ Enter:

Name = Joint 1
Normal Stiffness = 250000
Shear Stiffness = 100000
✓ Slip Criterion = Mohr Coul.
Tensile Strength = 0
Cohesion = 0
✓ Friction Angle = 20
✓ Initial Joint Def. = ☐ (off)

With the first tab selected in the Define Joint Properties dialog, enter the above properties. Note – turn OFF Initial Joint Deformation, by clearing the checkbox.

You have now defined all the required properties for the model. Since you entered both the rockmass and the joint properties with the first tab selected in the Define Properties dialogs, you do not have to Assign these properties to your model. PHASE² automatically assigns the Material 1 and Joint 1 properties for you.

However, we still have to use the Assign Properties dialog to excavate the material within the circular excavation.

Select: Properties → Assign Properties
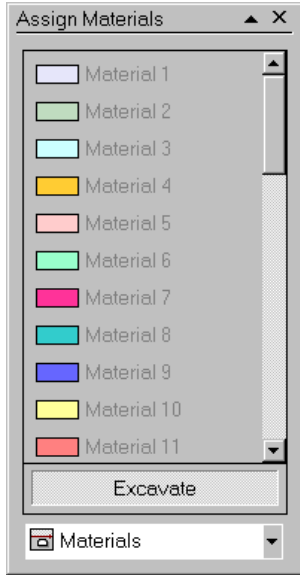
1. Select the Excavate button in the Assign Properties dialog.

2. Click the mouse within the circular excavation, to excavate the elements within the excavation.

3. Close the Assign Properties dialog.

You have now completed the modeling for this tutorial, your model should appear as shown below.
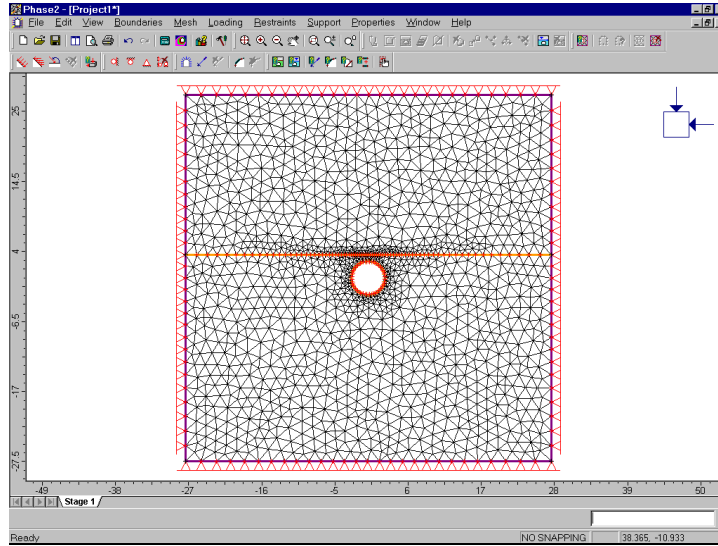

Figure 5-1: Finished model – PHASE² Joint Tutorial

Compute

Before you analyze your model, save it as a file called **joint.fea**.

Select: File → Save

Use the Save As dialog to save the file. You are now ready to run the analysis.

Select: File → Compute

The PHASE^2 COMPUTE engine will proceed in running the analysis. When completed, you will be ready to view the results in INTERPRET.

Interpret

To view the results of the analysis:

Select: File → Interpret

This will start the PHASE^2 INTERPRET program.

First let's zoom in so that we can get a better look at what's going on near the excavation.

Select: View → Zoom → Zoom Excavation

That zooms us in a bit too close, so select the Zoom Out button on the Zoom toolbar 3 times, to zoom back out a bit (or press the F4 key three times).

Select: View → Zoom → Zoom Out

(Note: we could have used Zoom Window to achieve the same result. The advantage of the above procedure, is that it gives us an exactly reproducible view of the model each time we use it.)

Observe the effect of the joint on the Sigma 1 contours. Notice the discontinuity of the contours above and below the joint. The effect of the joint is to deflect and concentrate stress in the region between the excavation and the joint. Now view the strength factor contours.

Select: **Strength Factor**

Notice the discontinuity of the strength factor contours above and below the joint. Now view the Total Displacement contours.

Select: **Total Displacement**

The discontinuity of the displacement contours is not apparent. However, let's change the contour style and see what happens. Right-click the mouse and select Contour Options.

In the Contour Options dialog, change the Auto-Format to DOS style, and select Done.

Now the discontinuity of the displacement contours can be seen. This is an important point – the appearance of contour plots, and your interpretation of them, can change significantly if you use different Contour Options. The contour style, range, and number of intervals, can all affect your interpretation of the data.

Joint Yielding

So now we ask, what is causing this perturbation of the contours in the vicinity of the joint? We will find out if we view the yielded joint elements. Select the Yielded Joints button in the Display toolbar.

The yielded joint elements are highlighted in red on the model, and the number of yielded elements is displayed in the status bar:

```
16 Yielded joint elements
```

Two separate zones of yielding in the joint can be seen, to the right and left of the excavation. It is now clear that inelastic slip (yielding) along the joint is responsible for the discontinuity of contours above and below the joint.

Also view the Strength Factor and Sigma 1 contours, and notice that the region of joint slip corresponds closely to the region of contour discontinuity.

Remember that the joint is allowed to slip because when we defined the joint properties, we used the Mohr-Coulomb slip criterion, with a friction angle of 20 degrees.

Let's just quickly verify that there are 16 yielded joint elements. Right-click the mouse and select Display Options.

In the Display Options dialog, select Discretizations and select Done.

You can now count the yielded joint elements, and there are in fact 16 (8 in the left yielded region, and 8 in the right). Toggle off the display of Discretizations in the Display options dialog.

Graphing Joint Data

Graphs of normal stress, shear stress, normal displacement and shear displacement can be easily obtained for joints, using the Graph Joint Data option.

Select: Graph → Graph Joint Data

Since there is only one joint in the model, it is automatically selected, and you will see the Graph Joint Data dialog:

The Graph Joint Data option is also available if you right-click on a joint.
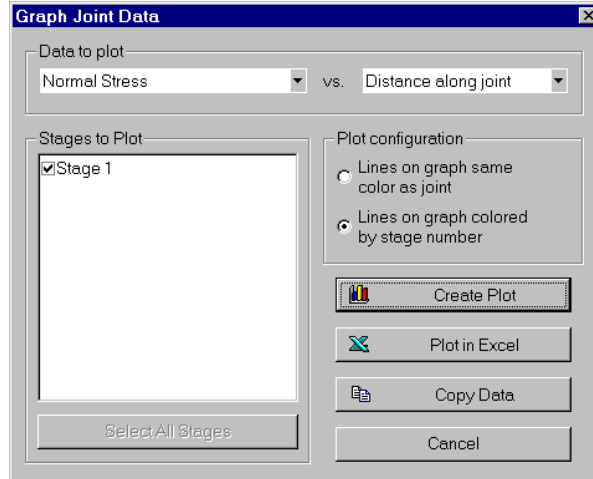
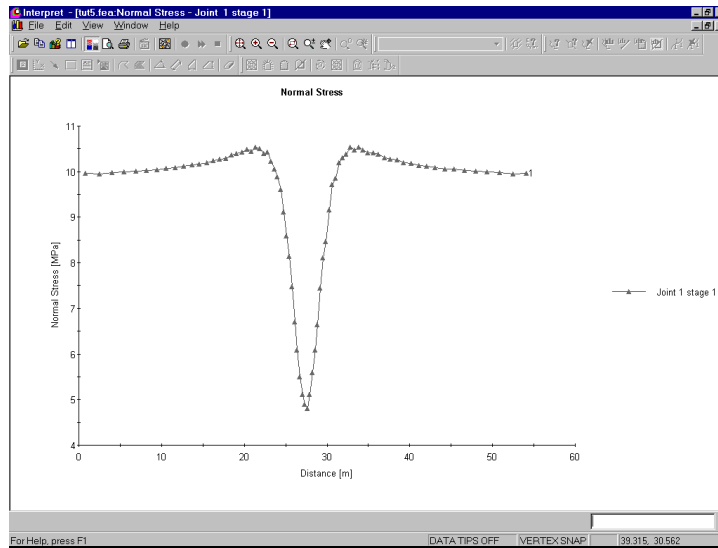*Just select Create Plot, to generate a plot of Normal Stress along the length of the joint.*

Figure 5-2: Normal stress along joint.

As expected, there is a sharp drop in normal stress where the joint passes over the excavation.

Notice the number 1 at the end of the curve. If you switch
back to the model view, you will also see a number 1 on
the joint. This serves two purposes: 1) if there are
multiple joints in your model, this number serves as an ID
number and 2) it identifies which end of the joint
corresponds to the end of the curve. For this example, it
does not matter, since the joint and model are symmetric.
If the model were not symmetric, then the location of the
ID number would be important.

Now repeat the above procedure, to create a graph of
shear stress along the joint (in the Graph Joint Data
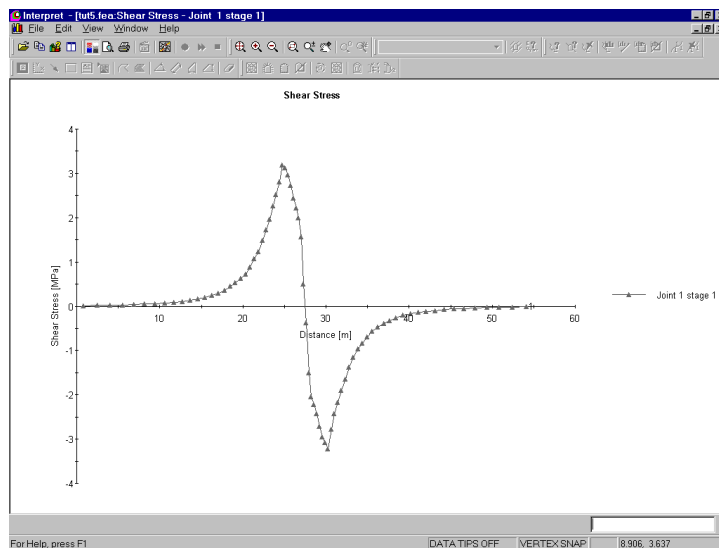dialog, select the Data to Plot as Shear Stress, and select
Create Plot.)

Figure 5-3: Shear stress along joint.

Notice the reversal of the shear stress direction over the
excavation. It is this sense of slip which produces the
inward displacement of rock on the underside of the plane
of weakness.

It is left as an optional exercise, to create graphs of normal displacement and shear displacement for the joint, and verify that the shape of the graphs correspond to the normal and shear stress plots. (Normal and shear displacement for joints refers to the relative movement of nodes on opposite sides of the joint).

Additional Exercise

Critical Friction Angle for Slip

Calculations in Brady & Brown indicate that if the angle of friction for the plane of weakness exceeds about 24°, no slip is predicted on the plane, and the elastic stress distribution can be maintained. As an exercise, run the analysis using angles of friction for the joint of 20 to 24 degrees, and then use the Yielded Joints option (as described above), to check the slip on the joint. You should find the results below:

| Angle of friction for joint	Number of yielded joint elements
20°	16
21°	12
22°	9
23°	4
24°	0

Table 5-1: Effect of joint friction angle on joint slip.

The results above confirm that the critical angle for joint slip in this example, is around 24 degrees.

Reference for Tutorial 5

Brady, B.H.G. and Brown, E.T., *Rock Mechanics for Underground Mining*, George Allen & Unwin, London, 1985, pp193-194.

Axisymmetry Tutorial
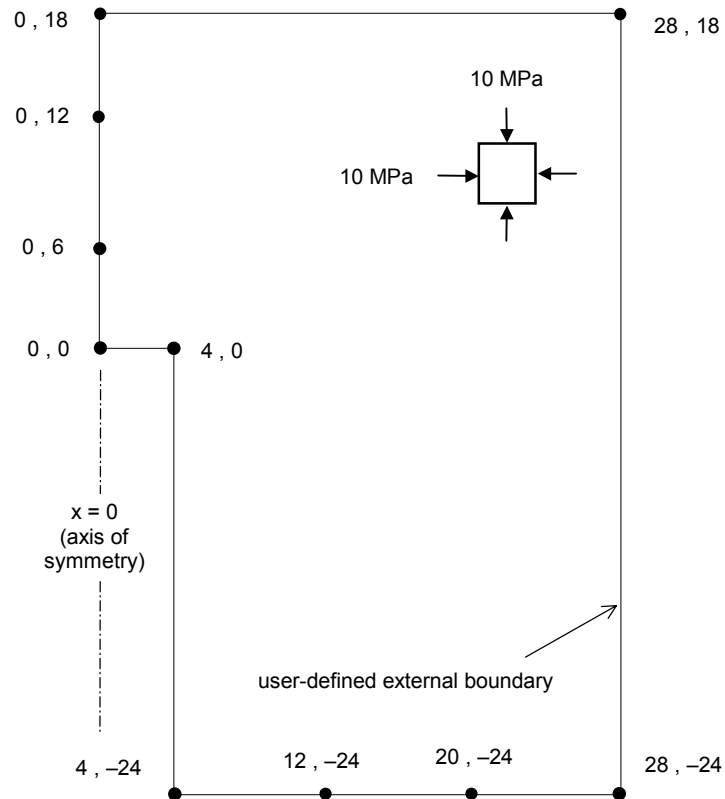
This tutorial will illustrate the axisymmetric modeling option of PHASE². Axisymmetric modeling allows you to analyze a 3-D excavation which is rotationally symmetric about an axis. The input is 2-dimensional, but the analysis results apply to the 3-dimensional problem.

An Axisymmetric model in PHASE² is typically used to analyze the end of a circular tunnel. The model we will be analyzing, shown above, represents the end of a cylindrical tunnel of 4 meter radius.

NOTE: the finished product of this tutorial can be found
in the **tut6.fea** data file located in the EXAMPLES folder
in your PHASE^2 installation folder.

A few representations of simple axisymmetric models are
shown below.

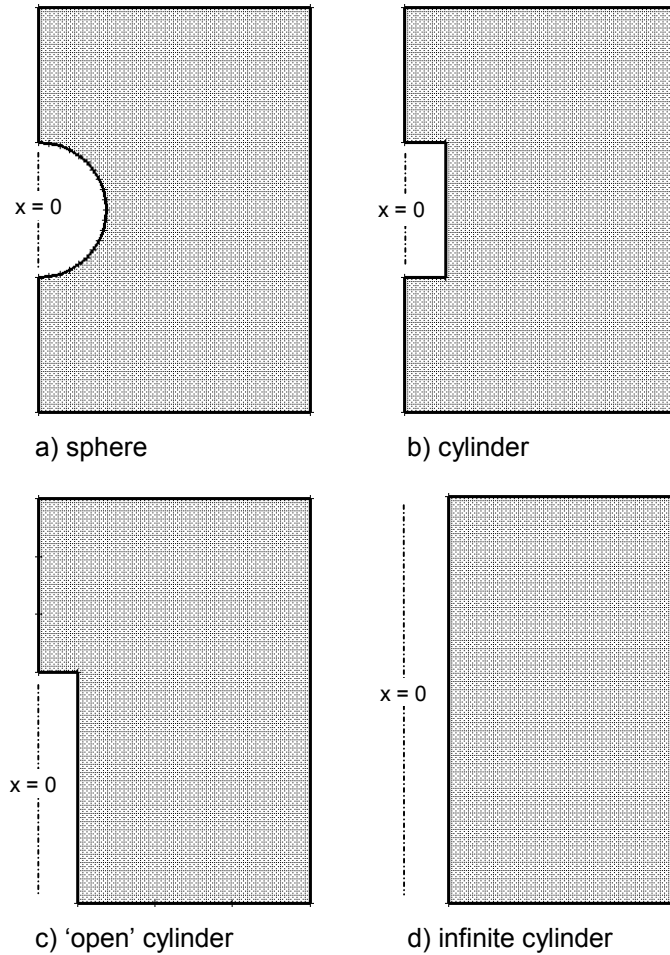a) sphere

b) cylinder

c) 'open' cylinder

d) infinite cylinder

Figure 6-1: Simple axisymmetric models. For a), b) and c), the left
edge of each boundary is coincident with the X = 0 (vertical) axis.
For d), the boundary is displaced from the X = 0 axis, therefore
modeling an infinite circular tunnel.

Note that:

- Only an external boundary is necessary to define an axisymmetric model – the excavation is implicitly defined by the shape and location (relative to the x=0 axis) of the external boundary. Appropriate boundary conditions must also be applied to complete the modeling.

- The axis of rotation is always the X = 0 (vertical) axis. Your model must always be mapped to fit this convention, regardless of the actual orientation of the excavation. Because of the symmetry, only "half" of the problem needs to be defined.

So, to visualize an axisymmetric excavation, just imagine the shape formed by rotating the external boundary about the x=0 axis. Note that Figure 6-1 is for illustration, and that actually the boundaries should be extended relative to the excavations (in Figures 6-1a and 6-1b) to ensure that the fixed boundary conditions do not affect the results around the excavation. Figure 6-1d can actually be defined by a narrow horizontal strip, since the problem is effectively one-dimensional (ie. results will only vary along a line perpendicular to the tunnel), and is in fact equivalent to a circular excavation in a plane strain analysis.

There are various restrictions on the use of axisymmetric modeling in PHASE2, for example –

- the field stress must be axisymmetric ie. aligned in the axial and radial directions.

- cannot be used with BOLTS (however LINERS are permitted)

- cannot be used with JOINTS

- all materials must have ISOTROPIC elastic properties

In this tutorial, we will look at results not only around the end of the tunnel, but also along its length, where the conditions are effectively plane strain. We will later verify these results by comparing with a plane strain analysis.

Model

If you have not already done so, start the PHASE^2 MODEL program by selecting Programs → Rocscience → Phase2 → Phase2 from the Start menu.

Limits

Let's first set the drawing limits so that they encompass the coordinates you will be entering for the external boundary.

Select: View → Limits

Enter the following coordinates in the View Limits dialog, and select OK.
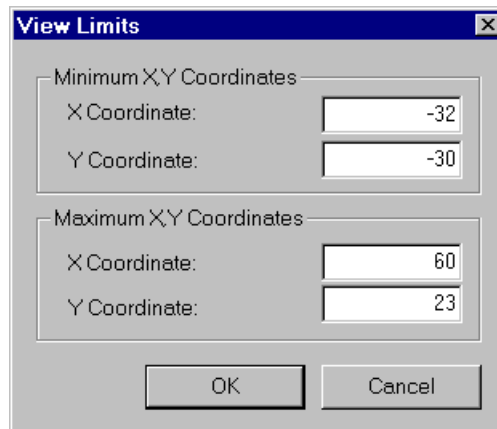
These limits will allow you to see the entire external boundary as you are entering it.

Project Settings

The next step is to select Project Settings and toggle the Analysis Type to Axisymmetric.

Select: File → Project Settings

In the Project Settings dialog, toggle the Analysis Type to Axisymmetric, and select OK.

Entering Boundaries

Since only an external boundary is required to define an axisymmetric problem in PHASE², proceed directly to the Add External option (rather than the usual procedure of first adding excavations).

Select: Boundaries → Add External

Enter the following coordinates at the prompts:

```
Enter vertex [esc=quit]: 28 18
Enter vertex [u=undo,esc=quit]: 0 18
Enter vertex [u=undo,esc=quit]: 0 12
Enter vertex [c=close,u=undo,esc=quit]: 0 6
Enter vertex [c=close,u=undo,esc=quit]: 0 0
Enter vertex [c=close,u=undo,esc=quit]: 4 0
Enter vertex [c=close,u=undo,esc=quit]: 4 -24
Enter vertex [c=close,u=undo,esc=quit]: 12 -24
Enter vertex [c=close,u=undo,esc=quit]: 20 -24
Enter vertex [c=close,u=undo,esc=quit]: 28 -24
Enter vertex [c=close,u=undo,esc=quit]: c
```

This is the only boundary required for the problem, so we can proceed to the meshing.

Meshing

As usual, we will discretize and mesh the model. However, let's first take a look at the Mesh Setup option.

Select: Mesh → Setup

✓ *Enter:*

Mesh Type = Graded

Elem. Type = 3 Noded Tri.

External Nodes = 60
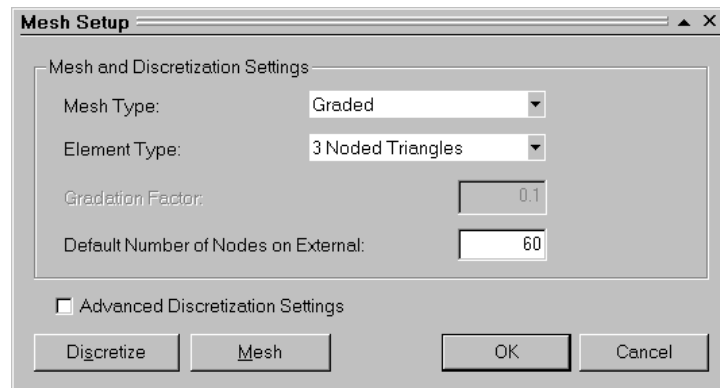
Notice that the Mesh Setup dialog normally asks you for the # Excavation Nodes. However, for models which have no explicitly defined excavations (such as this one), the # External Boundary Nodes is entered instead. Also, the Gradation Factor is not applicable when there are no excavation boundaries defined. Select OK or Cancel, since we are using the default parameters.

Now let's discretize the external boundary.

Select: Mesh → Discretize

The status bar will indicate the actual number of discretizations created on the external boundary.

```
Discretizations: External=64
```

Note that this is a fairly coarse discretization. The boundary segments which are part of, or adjacent to, the excavation, will require a finer discretization. We can do this with the Custom Discretize option.

Custom Discretization

Select: Mesh → Custom Discretize

```
Select Segments to Discretize [enter=done,
esc=quit]: use the mouse to select the long edge
of the tunnel ie. the long vertical segment at the
lower left of the external boundary. Right-click
and select Done Selection, or just press Enter.
```

In the Custom Discretize dialog, enter 60 as the number of discretizations, and select OK.
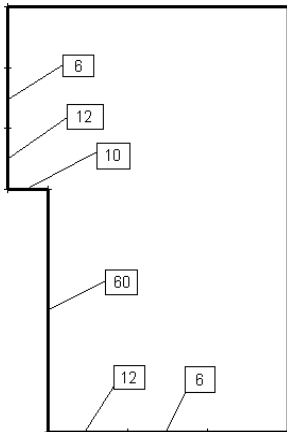
The length of the tunnel is now discretized into 60 elements. Now follow this same procedure to apply custom discretizations as indicated in the margin figure.

Note:

• you can select more than one line segment at a time, if they require the same number of discretizations (for example, the segments with 6 and 12 discretizations, in this case).

• the segments which are *not* marked in the margin figure, are to be left at their original discretizations.

CUSTOM DISCRETIZATION

Mesh

Now select the Mesh option within the Mesh menu, and the mesh will be generated, based on the discretization you just created.

Select: Mesh → Mesh

The status bar will indicate the total number of elements and nodes in the mesh.

```
ELEMENTS = 1225 NODES = 685
```

At this point, we will make the following observation – you may have wondered, when we created the external boundary, why we added the extra vertices on the upper left vertical segment and lower right horizontal segment of the boundary, since these boundaries could have been defined by single segments. As you can now see, the extra segments allowed us to custom discretize the boundaries, in order to get a smooth transition between the fine mesh around the tunnel, and the coarser mesh of the rest of the boundary. (If we did not do this, a poor mesh would be generated where the fine to coarse transition is too abrupt.)

Boundary Conditions

In most of the tutorials so far, we have not been specifying any boundary conditions. We were using the default boundary condition, which is a fixed (zero displacement) external boundary.

For an axisymmetric model, the external boundary conditions are very important, and must be user specified. We cannot simply leave the boundary fixed, or else nothing would happen (ie. no displacements could take place).

First, let's 'free' the tunnel boundaries.

DISPLACEMENT BOUNDARY
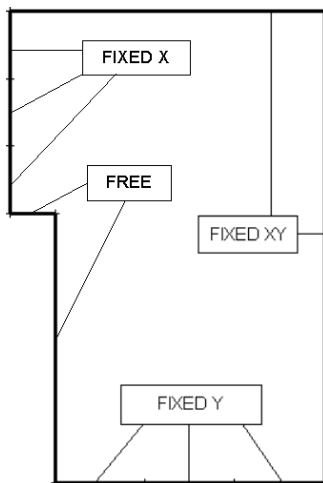CONDITIONS

Select: Restraints → Free

```
Select segments to free [enter=done, esc=quit]:
```
Use the mouse to select the 2 segments marked FREE in the margin figure on the previous page. When finished, right-click and select Done Selection, or press Enter.

The triangular pin symbols are now gone from the two boundary segments (representing the end of the tunnel and the length of the tunnel), indicating that they are free to move with no restriction in any direction.

Now let's specify the boundary segments at the upper left edge as restrained in the X direction, but free to move in the Y direction. (Note that these segments are located on the axis of symmetry, and therefore must have zero X displacement).

Select: Restraints → Restrain X

```
Select segments to restrain in the Y direction
[enter=done,esc=quit]:
```
Use the mouse to select the three segments marked FIXED X in the margin figure on the previous page. Right-click and select Done Selection, or press Enter.

Observe that the triangular pins on these segments have been replaced by vertical rollers.

Now let's specify the boundary segments along the bottom as restrained in the Y direction, but free to move in the X direction.

Select: Restraints → Restrain Y

```
Select segments to restrain in the Y direction
[enter=done,esc=quit]:
```
Use the mouse to select the three segments marked FIXED Y in the margin figure on the previous page. Right-click and select Done Selection, or press Enter.

Observe that the triangular pins on the bottom segments have been replaced by horizontal rollers.

Now we have a bit of tidying up to do.

Select: Restraints → Restrain X,Y

1. Right-click the mouse and select Boundary Node from the popup menu. This will change the mode of restraint application from boundary segments to boundary nodes. (The mode can also be changed in the Restraints→ Options dialog).

2. Select the upper left corner of the model, ie. the vertex at (0 , 18). A triangular pin symbol should appear at the vertex, replacing the roller symbol.

3. Select the lower right corner of the model, ie. the vertex at (28 , -24). A triangular pin symbol should appear at the vertex.

4. Press Escape or right-click and select Cancel.

The above steps were necessary, since the upper left and lower right vertices required a Restrained XY condition. This leads us to an important point – after applying restraints to boundary segments, you should always check that nodes at the ends of segments have the correct conditions applied.

Field Stress

For this example, we will be using the default hydrostatic stress field of 10 MPa. However, let's look at the field stress option, to see how an axisymmetric field stress is specified.

Select: Loading → Field Stress

Only two independent principal stresses (Horizontal and Vertical) are specified for an axisymmetric problem, and no angle is allowed. Select OK or Cancel.

Note the following correspondences between Plane Strain and Axisymmetric field stress, as defined in PHASE2:

PLANE STRAIN	AXISYMMETRIC
Sigma 1 (in-plane)	'Horizontal' stress
Sigma 3 (in-plane)	'Horizontal' stress
Sigma Z (out-of-plane)	'Vertical' stress
Angle	not applicable

Table 6-1: Equivalent plane strain and axisymmetric field stress components.

NOTE:

- The Horizontal (axisymmetric) field stress can also be thought of as a uniform radial stress around the excavation.

- An angle cannot be specified for the axisymmetric field stress, because this would make the analysis effectively 3-dimensional, which is beyond the scope of the PHASE[2] axisymmetric analysis.

- It should be emphasized that the terms Horizontal and Vertical refer strictly to the setup of your model in PHASE[2], 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 this axis.

Properties

In this tutorial, we will not be defining or assigning any properties, therefore all default properties will be in effect. We have dealt with defining and assigning properties in previous tutorials.

For reference purposes, the default rock properties which will be in effect are shown in the margin. (If you want, you can verify this by selecting Properties→ Define Materials).

Note that our analysis will therefore be elastic, and also note the values of Young's Modulus and Poisson's ratio.

We have now completed the modeling, your finished model should appear as shown below.

✓ *Enter:*

Name = Material 1
Init.El.Ld.=Fld Stress Only
Material Type = Isotropic
Young's Modulus = 20000
Poisson's Ratio = 0.2
Failure Crit. = Mohr Coul.
Material Type = Elastic
Tens. Strength = 0
Fric. Angle (peak) = 35
Cohesion (peak) = 10.5

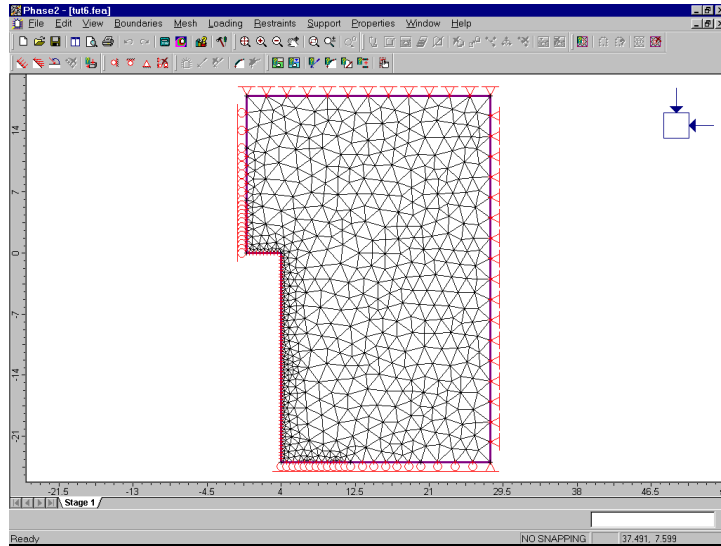Figure 6-2: Finished model – PHASE² Axisymmetric Tutorial

Compute

Before you analyze your model, save it as a file called **axi1.fea**.

Select: File → Save

Use the Save As dialog to save the file. You are now ready to run the analysis.

Select: File → Compute

The PHASE² COMPUTE engine will proceed in running the analysis. When completed, you will be ready to view the results in INTERPRET.

Interpret

To view the results of the analysis:

Select: File → Interpret

This will start the PHASE² INTERPRET program.

Sigma 1

On the Sigma 1 contours, notice the stress concentration at the 'corner' of the tunnel (remember the tunnel is circular).

Toggle the stress trajectories on by selecting the Stress Trajectories button in the Display toolbar. The principal stress trajectories illustrate the "stress flow" around the end of the tunnel.
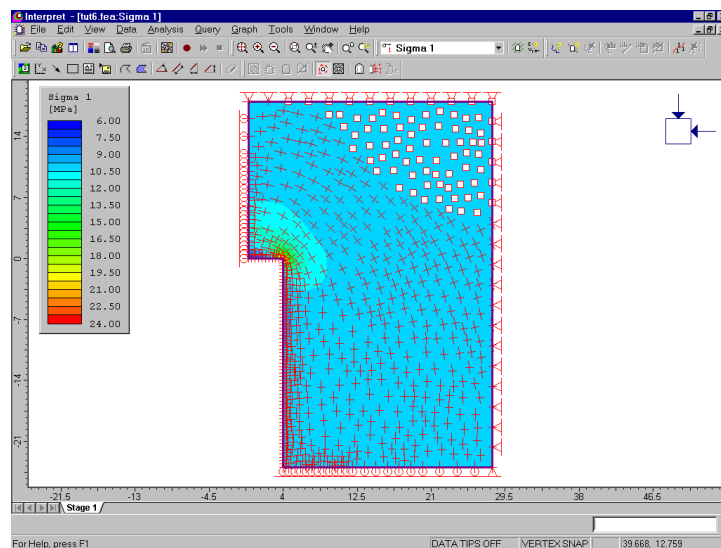

Figure 6-3: Principal stress trajectories around axisymmetric excavation.

The square dot markers in the upper right corner of the model indicate nodes where the difference between Sigma 1 and Sigma 3 is less than a certain tolerance, so that the conditions are effectively hydrostatic, and a distinction between 'major' and 'minor' principal stress is not warranted.

Toggle the stress trajectories off by re-selecting the Stress Trajectories button in the Display toolbar.

As an optional step, look at Sigma 3 and Sigma Z, and consider the significance of the principal stress results from an axisymmetric analysis. As with plane strain, Sigma 1 and Sigma 3 are the major and minor 'in-plane' principal stresses. Sigma Z is therefore the 'out-of-plane' stress, however, since the problem is axisymmetric, Sigma Z is really the induced circumferential or hoop stress around the excavation.

Displacement

Now let's view the displacements.

Select: **Total Displacement** ▼

Note the maximum total displacement displayed in the status bar is 0.00228 m, or just over 2 mm. This is quite small, but remember our analysis was elastic and we used a relatively high Young's modulus.

Now let's view the deformation vectors. Right-click the mouse and select Display Options.

In the Display Options dialog, toggle Deformation Vectors on, enter a Scale Factor of 600, and select Done.

The deformation vectors show the inward displacement along the length and face of the tunnel. Notice how the "corner" of the tunnel effectively restrains the displacements in both x and y directions.
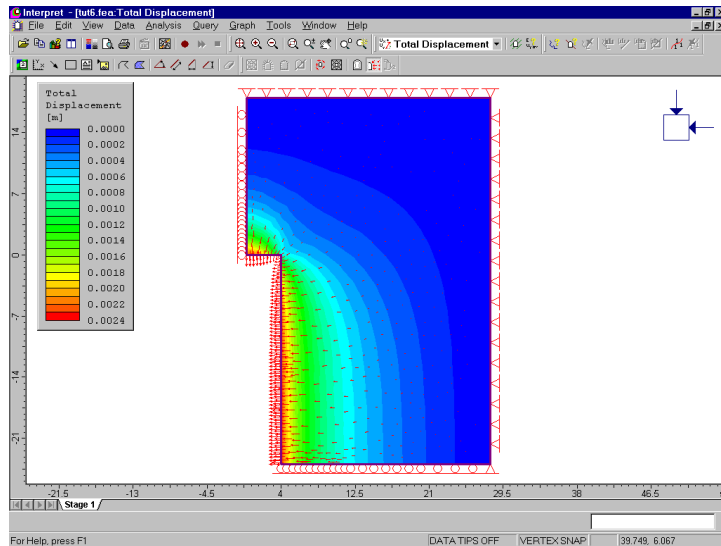
Figure 6-4: Displacement contours and vectors around axisymmetric excavation.

Toggle the deformation vectors off by selecting the Deformation Vectors button in the Display toolbar.

Query Data

PHASE[2] allows the user to query data anywhere in the rockmass, to obtain values interpolated from the contour plots. A query can be a single point, a line segment, or any arbitrary polyline.

Let's first create a query along the length of the tunnel.

Select: Query → Add Material Query

1. It will be handy to use the Snap option, so right-click the mouse and select Snap. While in Snap mode, if you place the cursor near a vertex, you can snap exactly to the location of a vertex.

2. Use the mouse to select the vertex at (4 , 0).

3.   Use the mouse to select the vertex at (4 , -24).

4.   Right-click the mouse and select Done. You will see
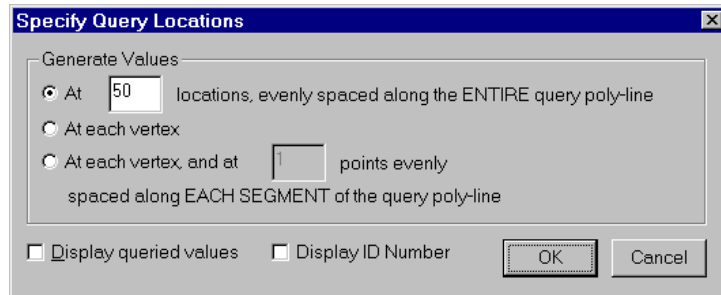   the following dialog:

Specify Query Locations

Generate Values

⊙ At [50]   locations, evenly spaced along the ENTIRE query poly-line

○ At each vertex

○ At each vertex, and at [1]   points evenly
     spaced along EACH SEGMENT of the query poly-line

☐ Display queried values   ☐ Display ID Number   [OK]   [Cancel]

*Enter 50 locations, toggle off Display Queried Values, and
select OK.*

A query has now been created along the length of the
tunnel. The 50 locations at which data will be generated
are indicated by black cross markers.

Graphing a Query

Graphs are created from queries with the Graph Material
Queries option. However, a convenient shortcut to graph
data for a single query, is to simply right-click on a query
and select Graph Data.

1.   Right-click on the query you just created (ie.
     anywhere along the length of the tunnel), and select
     Graph Data from the popup menu.

2.   You will see the Graph Query Data dialog. Select the
     Create Plot button in this dialog.

3.   You should see the graph in Figure 6-5. Since we were
     viewing the Total Displacement contours, we obtained
     a graph of total displacement vs. distance along the
     query. The data graphed always corresponds to the
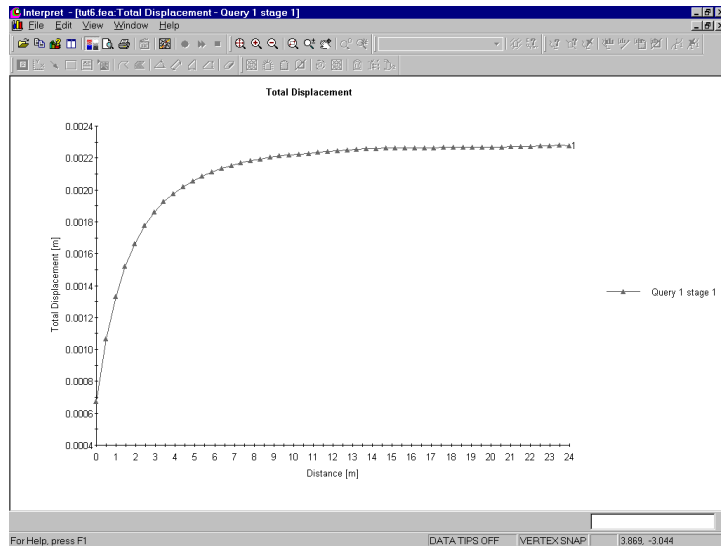   contoured data being viewed.

Figure 6-5: Total displacement along length of tunnel.

As you can see, the displacement levels off and becomes constant at a certain distance away from the tunnel face. This curve is useful in that it allows us to see the distance at which end effects can be ignored, and plane strain conditions can be assumed. Also, this curve can be used to estimate the "load split" factors, as described in the Support Tutorial, Step 2.

Deleting a Query

Queries are deleted with the Delete Material Query option. However, a convenient shortcut to delete a single query, is to simply right-click on the query and select Delete Query.

1. First close the graph if you are still viewing it.

2. Right-click on the query and select Delete Query, and the query will be removed from the model.

Graphing Multiple Queries

Now let's create two more queries, this time perpendicular to the tunnel, and plot them on the same graph.

Select: Query → Add Material Query

1. The Snap option should still be in effect, so use the mouse to select the vertex at (4 , -24), and then select the vertex at (28 , -24). Right-click the mouse and select Done.

2. In the Specify Query Locations dialog, again enter 50 locations, toggle off Display queried values, and select OK.

Add another query.

Select: Query → Add Material Query

3. Use the mouse to select the vertex at (4 , 0). Now, since there is no vertex to snap to, enter the coordinates (28 , 0) in the prompt line. Right-click the mouse and select Done.

4. In the Specify Query Locations dialog, again enter 50 locations, toggle off Display queried values, and select OK.

You have created two new queries, one along the lower edge of the model, and a parallel one at the face of the tunnel.

This time, to graph the queries, we will use the Graph Material Queries option, since we want both queries on the same graph. (The right-click shortcut can only be used to graph a single query).

Select: Graph → Graph Material Queries

5. Select the two queries by clicking on them with the left mouse button. (Alternatively, you could right-click the mouse and select Select All from the popup menu.)

6. Right-click and select Graph Selected (or just press Enter), and you will see the Graph Query Data dialog.

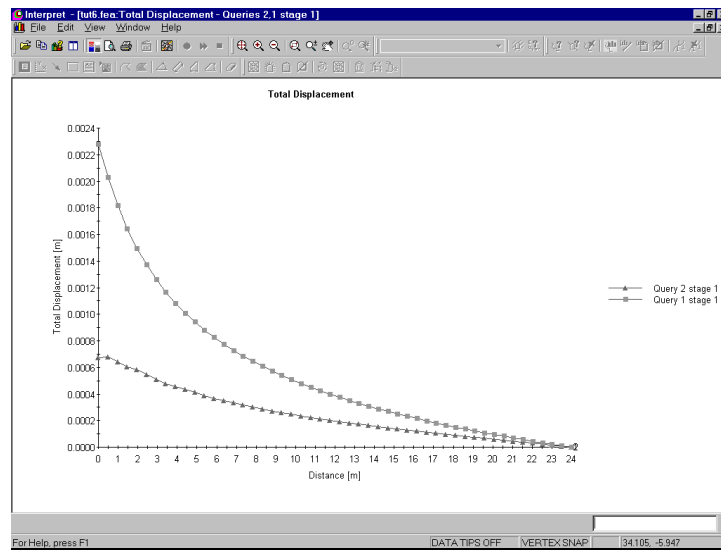7. Select the Create Plot button in the dialog, and you should see the following graph.

Figure 6-6: Total displacement perpendicular to tunnel, at face (lower curve), and at 24 meters from face (upper curve).

The total displacement decreases as we move away from the tunnel. Note:

•   the Total Displacement along the lower boundary is exactly equivalent to the Horizontal (X) Displacement, since the Vertical (Y) Displacement along this boundary is zero. (If you graphed this query while viewing Horizontal Displacement instead of Total Displacement, you could verify this for yourself.)

- The Total Displacement curve at the face of the tunnel includes both X and Y displacement components.

Now close the graph view by selecting the X in the upper right corner of the view.

Writing a Query to a File

Finally let's save one of the queries to a file using the Write Query File option, so that we can use it in the last part of this tutorial.

Select: Query → Write Query File

1. Select the query at the bottom edge of the model with the mouse.

2. Right-click and select Write Selected.

3. In the Save As dialog, save the query in a file called "axidisp".

4. You will then see a dialog allowing you to add a comment which identifies the file. This is optional, you can add a comment if you wish.

You have just saved the data in a PHASE^2 "points value" file. A .pvf file is a simple text file format, which can be read back into PHASE^2. We will be demonstrating that in the final part of this tutorial.

Plane Strain Comparison with Axisymmetric Results

To further illustrate the significance and meaning of an axisymmetric model, we will create a plane strain model which is equivalent in all respects to our axisymmetric model (except of course that the tunnel will now be infinite, with no 'end effects'), and compare the analysis results.

From INTERPRET, switch back to MODEL.

Select: File → Model

(Or if you quit the program and are just re-starting the tutorial at this point, then start the PHASE^2 MODEL program.)

Model

If you have been following this tutorial from the beginning, and the axisymmetric model is still open,

Select: File → New

to open a new document window, so that you can begin creating a new model.

Plane strain is the default analysis type, so you do not have to set this in Project Settings, it will already be in effect. Let's first create a circular tunnel of radius 4 meters (ie. the same radius as the axisymmetric tunnel).

Select: Boundaries → Add Excavation

```
Enter vertex [a=arc,esc=quit]: a
```

By entering "a" at the first prompt, you can enter a circle (or arc) at the following prompts:

```
Number of segments in arc <20>: enter 60
Enter first arc point[esc=quit]: 4 0
Enter second arc point[u=undo, esc=quit]: -4 0
Enter third arc point[u=undo, esc=quit]: c
```

By entering "c" at the last prompt, the arc closes on itself and forms a circle, with a diameter determined by the distance between the first and second points, in this case 8 meters.

(Since we did not set any limits using View→Limits before we started modeling, the excavation is only partly visible. Press the F2 function key, or use Zoom All, to center the excavation in the drawing region. This is optional, since the entire model will center itself when we add the external boundary).

Now add the external boundary.

Select: Boundaries → Add External

✓ *Enter:*

  Boundary Type = Box
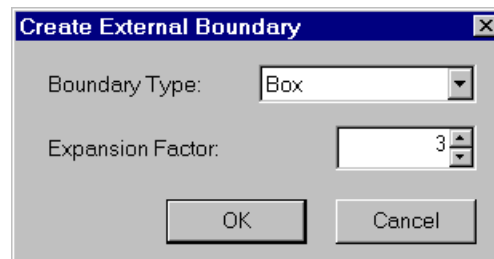
  Expansion Factor = 3

We will use the default parameters, so just select OK to automatically create a BOX external boundary with an expansion factor of 3.

Notice that this external boundary is the same distance away from the excavation, as the right edge of the external boundary for the axisymmetric problem (ie. 28 meters from the center of the tunnel).

Mesh

Now discretize and mesh the model. First select Mesh Setup.

Select: Mesh → Setup

In the Mesh Setup dialog, set the Number of Excavation Nodes to 60. Select the Discretize button in the Mesh Setup dialog (this is equivalent to using the Discretize option in the Mesh menu). The status bar will indicate:

```
Discretizations: Excavation=60, External=68
```

Select the Mesh button in the Mesh Setup dialog (this is equivalent to using the Mesh option in the Mesh menu). The status bar will indicate:

```
ELEMENTS = 1492 NODES = 781
```

Select OK in the Mesh Setup dialog.

Boundary Conditions

We will use the default boundary condition, which is a fixed (ie. zero displacement) condition on the external boundary. This corresponds to the Fixed XY condition of the *right edge* of the external boundary in the axisymmetric model.

Field Stress

We will use the default Field Stress (ie. hydrostatic conditions σ1=σ3=σZ = 10MPa, which is the same field stress we used for the axisymmetric problem), so you do not need to use the Field Stress option.

Properties

We will use the default rock material properties, so you do not have to enter or assign any properties. The default properties are Material Type = Elastic, Young's Modulus = 20,000 MPa, Poisson's ratio = 0.2.

However, we do have to excavate the tunnel.

Select: Properties → Assign Properties

1. In the Assign dialog, select the Excavate button at the bottom of the dialog.

2. Click the mouse inside the circular tunnel, and the elements will be excavated.

3. Close the Assign dialog by selecting the X button.

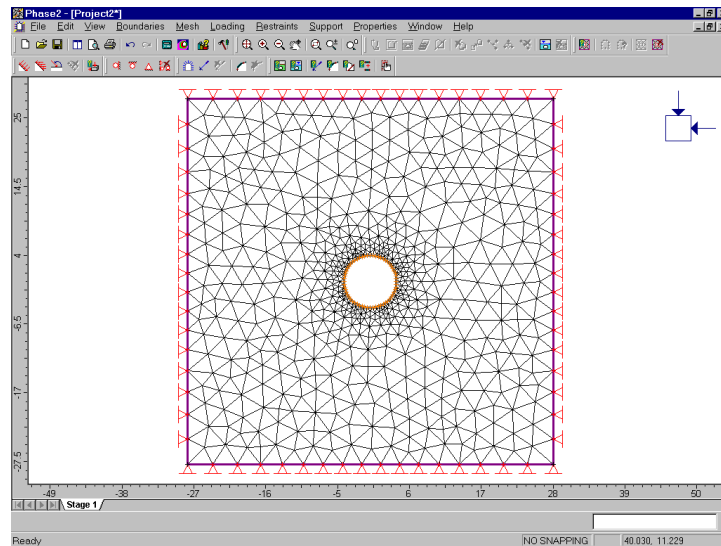We are finished with the modeling, the model should appear as below.

Figure 6-7: Infinite tunnel, 4 meter radius, plane strain problem.

Compute

Before you analyze your model, save it as a file called
axi2.fea.

Select: File → Save

Use the Save As dialog to save the file. You are now ready
to run the analysis. (Alternatively, if you select Compute
before saving a new file, PHASE^2 will recognize this, and
display the Save As dialog).

Select: File → Compute

The PHASE^2 COMPUTE engine will proceed in running
the analysis. When completed, you will be ready to view
the results in INTERPRET.

Interpret

We will now wrap up this tutorial with a few comparisons
between the axisymmetric and plane strain models we
have created. Switch back to INTERPRET.

Select: File → Interpret

View the total displacement contours.

Select: Total Displacement

Notice the maximum displacement displayed in the status
bar.

Maximum Total Displacement = 0.0022677 m

This is almost identical to the maximum displacement
from the axisymmetric problem (0.00228).

Now let's use the Query and Graph options again to plot the displacement vs. distance from the tunnel boundary.

Select: Query → Add Material Query

1. Enter the point (4 , 0) at the first prompt and (28 , 0) at the second prompt. Right-click and select Done, or just press Enter.

2. In the Specify Query Locations dialog, enter 50 locations, toggle off Display queried values, and select OK.

3. Notice the query created from the right edge of the tunnel to the right edge of the external boundary. Right-click on the query and select Graph Data.

4. You will see the Graph Query Data dialog. Select the Create Plot button, and a graph of total displacement will immediately be generated.

You can see the displacement decreasing from the maximum at the tunnel boundary, to zero at the fixed, external boundary.

Finally, let's compare this curve with the one we saved using Write Material Query at the end of the axisymmetric analysis. First close the graph you just created, if you are still viewing it.

Select: Graph → Graph Material Queries

1. Select the query with the left mouse button. (Notice that it is highlighted with a dotted line when selected).

2. Right-click the mouse and select Add from File. In the Open file dialog, open the "axidisp" file. (This is the query you saved earlier on the axisymmetric model).

3. You should see an information dialog, informing you that the query was successfully read. Select OK.

4. Now right-click and select Graph Selected, or press Enter. In the Graph Query dialog, select the Create Plot button, and the graph will be generated.

The graph should show two Total Displacement curves which overlap almost exactly.

- One curve represents the query along the lower edge of the axisymmetric model, which we read in using Step 2 above.

- The other is the query added on the plane strain model.

This illustrates the relationship between the axisymmetric and plane strain models – although the two models look very different, we can extract the same results from either one.

Additional Exercises

When you plotted the total displacement contours for the plane strain tunnel model, you may not have noticed, but the contours begin to get "square" as you get further from the tunnel (immediately around the tunnel they are circular). The displacements are conforming to the shape of the external boundary and the fixed boundary condition we imposed on it.
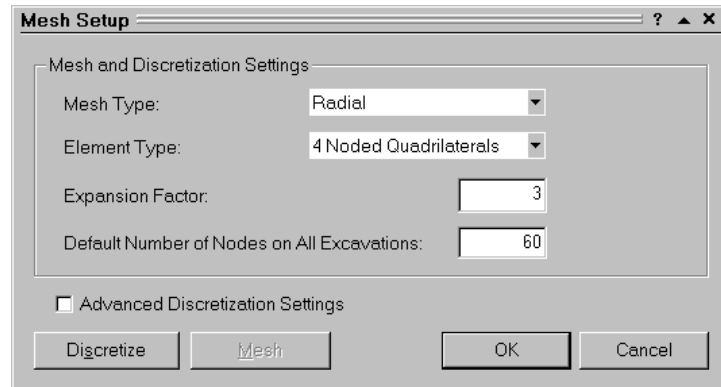
Radial Mesh

For circular problems such as this one, there is a more appropriate meshing option we could have used, called RADIAL meshing. Radial meshing produces a symmetric, reproducible radial mesh for symmetric problems such as this.

As an additional exercise, re-do the plane strain circular tunnel analysis, with the following changes:

- after you add the excavation, DO NOT add the external boundary, but select Mesh→Setup instead.

- in the Mesh Setup option, toggle the Mesh Type to RADIAL, the Element Type to 4 Noded Quadrilaterals, and enter the #Excavation Nodes = 60. Note that for a RADIAL mesh, an Expansion Factor for the external boundary is entered, rather than a Gradation Factor.

✓ *Enter:*

✓ Mesh Type = Radial

✓ Elem. Type = 4 Node Quad

   Expansion Factor = 3

✓ # Excavation Nodes = 60

- Discretize and Mesh. The external boundary will appear when the radial mesh is generated.

- Carry out the analysis and data interpretation as before.

When you plot the displacement versus distance from the tunnel, you should get nearly identical results as when you used the BOX external boundary. However, the displacement contours are no longer distorted, and are circular at any distance from the excavation.

Also note that quadrilateral elements, in conjunction with RADIAL meshing, are very efficient, and give very good results.

Distance of External Boundary from Excavation

As pointed out in other tutorials, the distance of the external boundary from the excavation(s), and the boundary conditions we impose on it, are very important.

Create another RADIAL mesh, as described above, except this time use an Expansion Factor of 5 (in the Mesh Setup dialog). Re-run the analysis and plot the displacement versus distance from the tunnel. You will see that actually our previous Expansion Factor of 3 was too low, because displacements increase significantly when we move the fixed external boundary farther from the tunnel.
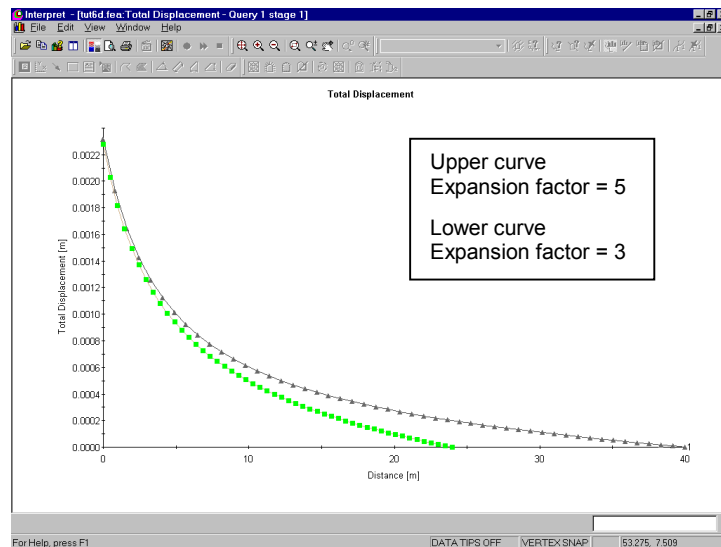

Figure 6-8: Moving the fixed external boundary farther from the excavation results in increased displacements.

The displacements near the excavation are comparable, but diverge towards the external boundary. For example, at about 18 meters from the excavation, the displacement for the Expansion Factor = 5 curve is about double the Expansion Factor = 3 curve (see Figure 6-8).

The restraining effect of a fixed external boundary should therefore always be considered. When comparing with analytical solutions, as in the PHASE[2] verification manual, it is very important to take this into account.

One final suggested exercise:

- Re-do the axisymmetric problem and move the right edge of the external boundary over to 44 meters. This gives an equivalent distance from the excavation as the plain strain model with an expansion factor of 5. Compare results with the equivalent plain strain model.