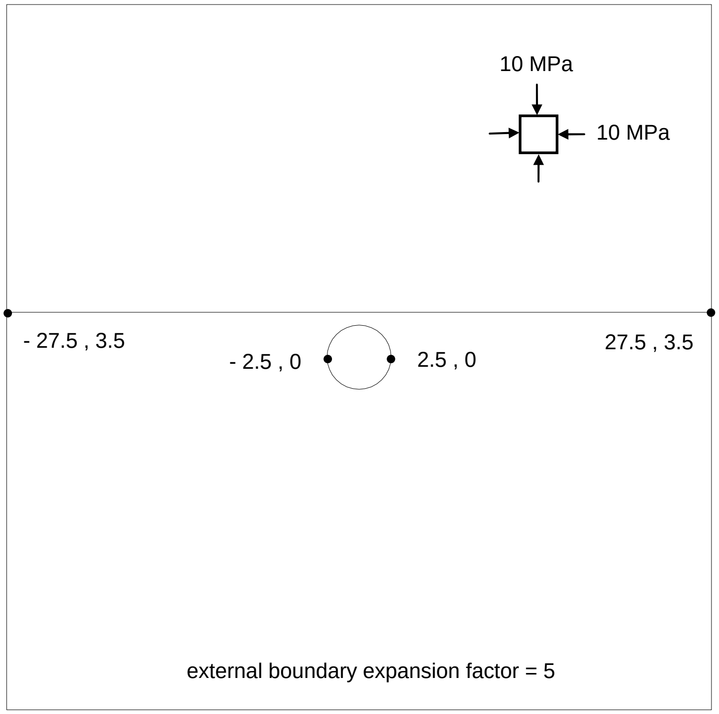
Joint Tutorial



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 **Tutorial 05 Joint.fez** file located in the Examples > Tutorial folder in your *Phase2* installation folder.

Model

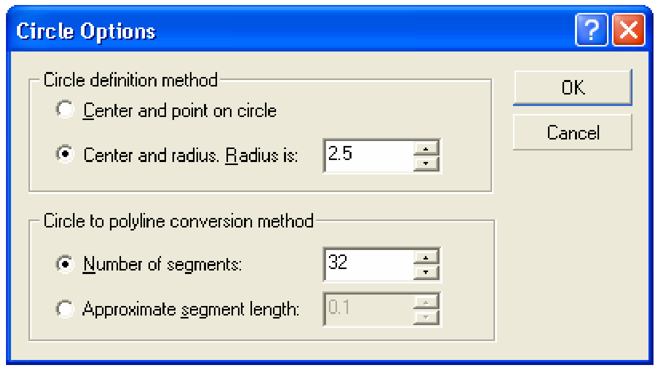
If you have not already done so, start the *Phase2* Model program by selecting Programs → Rocscience → Phase2 6.0 → Phase2 from the Start menu.

Entering Boundaries

First create the circular excavation as follows:

Select: Boundaries → Add Excavation

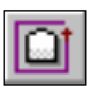
1. Right-click the mouse and select the Circle option from the popup menu. You will see the Circle Options dialog.



2. Select the Center and radius option, and enter a radius of 2.5. Enter Number of segments = 32 and select OK.

3. You will see a prompt to enter the circle center. Enter 0,0 in the prompt line, and the circular excavation will be created.

Now add the external boundary.

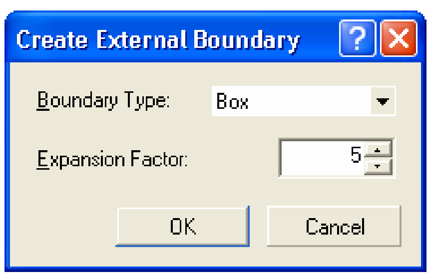


Select: Boundaries → Add External

✓ *Enter:*

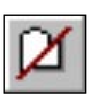
Boundary Type = Box

✓ Expansion Factor = 5



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

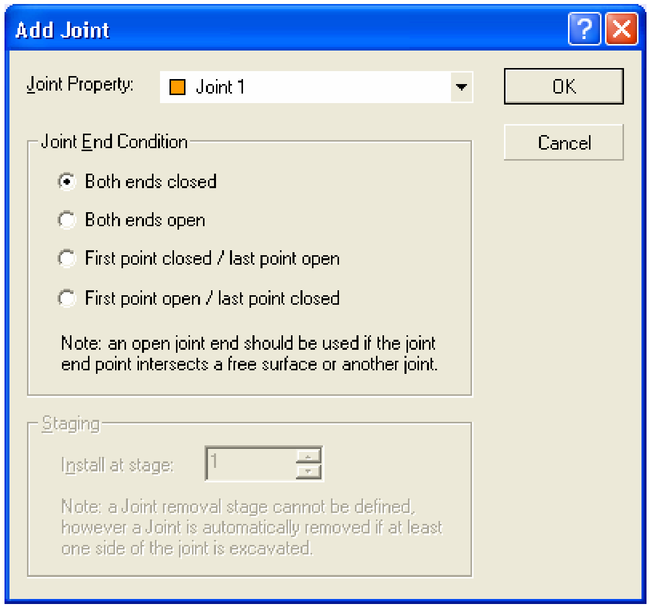
Now add the joint to the model.



Select: Boundaries → Add Joint

You will see the Add Joint dialog, which allows you to select a Joint property type, end condition and installation stage. We will use the default selections, so just select OK.

NOTE: see the *Phase2* Help system for a discussion of the Joint End Condition option.



Now enter the following coordinates defining the joint.

```
Enter vertex [t=table,i=circle,esc=cancel]: -30 3.5
Enter vertex [...]: 30 3.5
Enter vertex [...,enter=done,esc=cancel]: press Enter
```

The joint is now added to the model. Note that the “closed” Joint End Condition is indicated by an icon of a circle with a triangle inside, at both ends of the joint.

Phase2 automatically intersects boundaries and adds vertices when required.

Note 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:

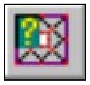
- the user does not know the exact intersection of two lines, the automatic intersection capability of *Phase2* saves the user the trouble of having to calculate such intersections, or when
- new vertices are required at known locations, they can be created automatically (rather than manually with the Add Vertices option).

Note: you could have entered (-27.5, 3.5) and (27.5,3.5) at the above prompts (i.e. 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.

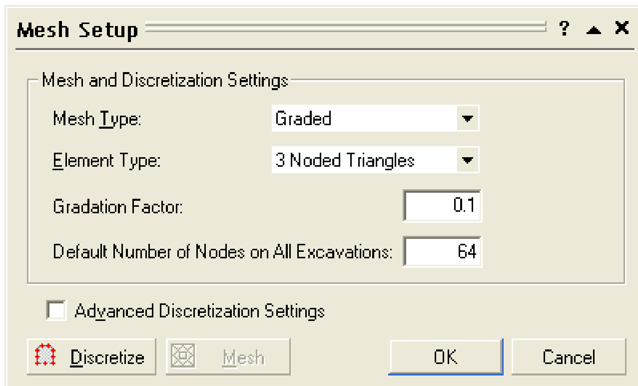
All boundaries have now been entered, so we can go ahead and mesh the model.

Meshing

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

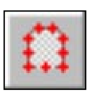


Select: Mesh → Mesh Setup



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

Now discretize the boundaries.



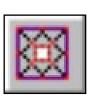
Select: Mesh → Discretize

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 from the toolbar or 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.

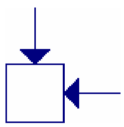
```
ELEMENTS = 3256 NODES = 1759
```

If you have followed the steps correctly so far, you should get 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 (i.e. zero displacement) condition for the external boundary.

Field Stress

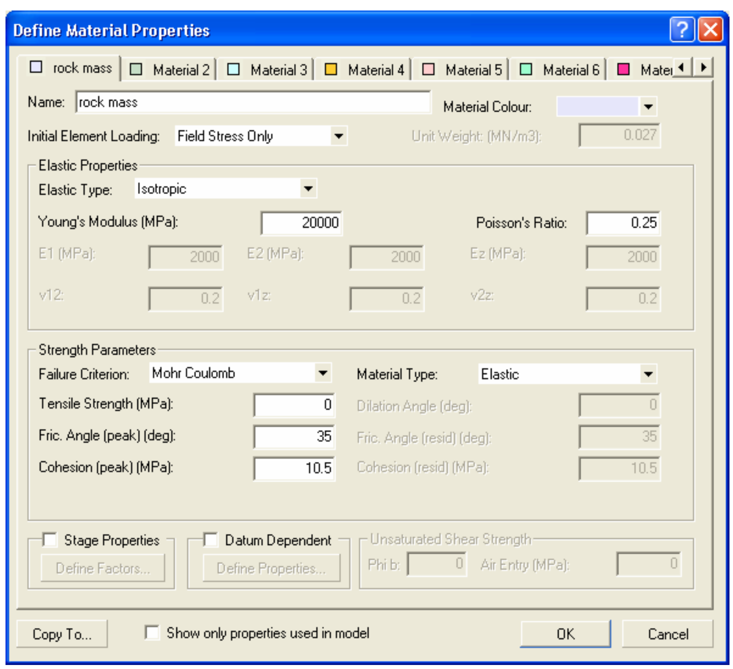


We will be using the default field stress for this model, which is a constant hydrostatic stress field with Sigma 1 = Sigma 3 = Sigma Z = 10 MPa. Therefore you do not have to enter any field stress parameters, the values we want are already in effect.

Properties

The properties of the rock mass and the joint must now be entered.

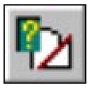
Select: Properties → Define Materials



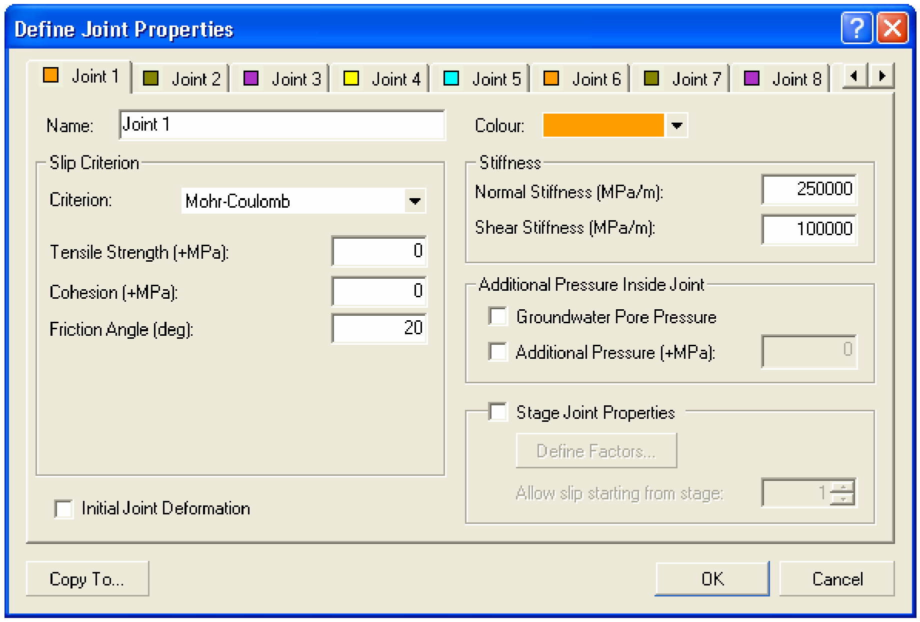
✓ *Enter:*
✓ Name = rock mass
Init.El.Ld.=Fld Stress Only
Material Type = Isotropic
Young's Modulus = 20000
✓ Poisson's Ratio = 0.25
Failure Crit. = Mohr Coul.
Material Type = Elastic
Tens. Strength = 0
Fric. Angle (peak) = 35
Cohesion (peak) = 10.5

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 rock mass properties, now do the same for the joint properties.



Select: Properties → Define Joints



✓ *Enter:*

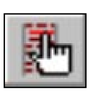
Name = Joint 1
Normal Stiffness = 250000
Shear Stiffness = 100000
✓ Slip Criterion = Mohr Coul.
Tensile Strength = 0
Cohesion = 0
✓ Friction Angle = 20
✓ Initial Joint Def. = ☐ (off)

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 rock mass 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 option to excavate the material within the circular excavation.

Right-click shortcut for Assigning



Assignment of properties and excavation can be easily done with a right-click shortcut, which we will now demonstrate.

1. Right-click the mouse within the circular excavation.
2. In the popup menu, go to the Assign Materials sub-menu, and select the Excavate option.

That's it, the excavation has been excavated in two quick steps, using the right-click shortcut.

TIP: when you have a lot of property assignments and excavating to do (e.g. for complex multi-stage models), it is easier to use the Assign Properties dialog to carry out assignments. However, when you only need to make one or two property or excavation assignments, or modifications to existing assignments, the right-click shortcut is very convenient and often faster to use.

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

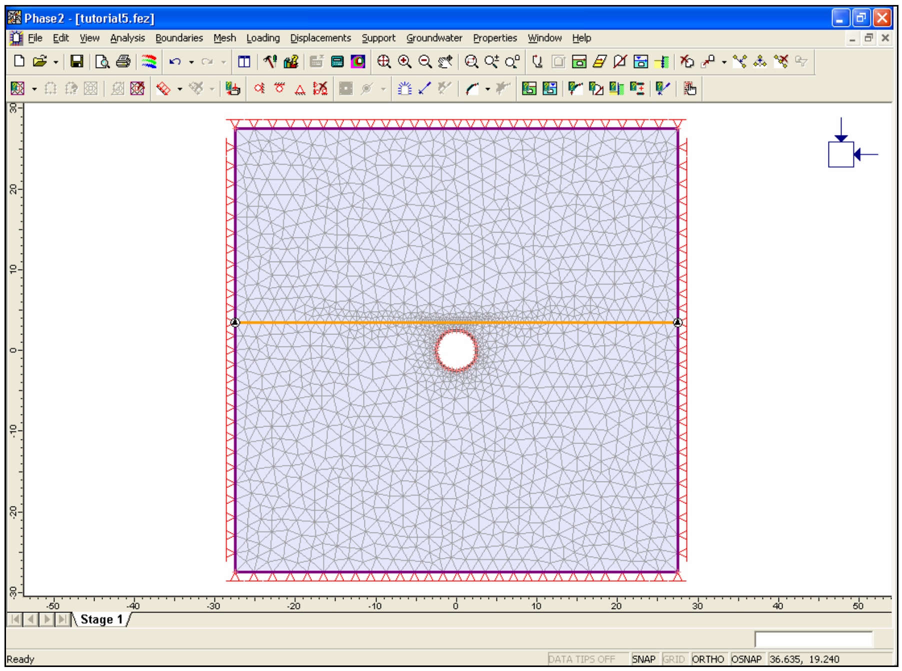
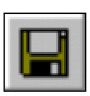


Figure 5-1: Finished model – *Phase2* Joint Tutorial

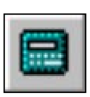
Compute

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



Select: File → Save

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

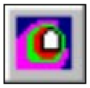


Select: Analysis → Compute

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: Analysis → Interpret

This will start the *Phase2* 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: 

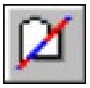
Notice the discontinuity of the strength factor contours above and below the joint. Now view the Total Displacement contours.

Select: 

The discontinuity of the displacement contours is not apparent. However, if you experiment with different contour options (e.g. try the Filled (with Lines) mode, the discontinuity of the displacement contours can be seen. This is left as an optional exercise.

TIP: 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



Now let's check for yielding of the joint. Select the Yielded Joints button in the 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. View the Strength Factor and Sigma 1 contours, and notice that the region of joint slip corresponds to the region of contour discontinuity, above and below the joint.

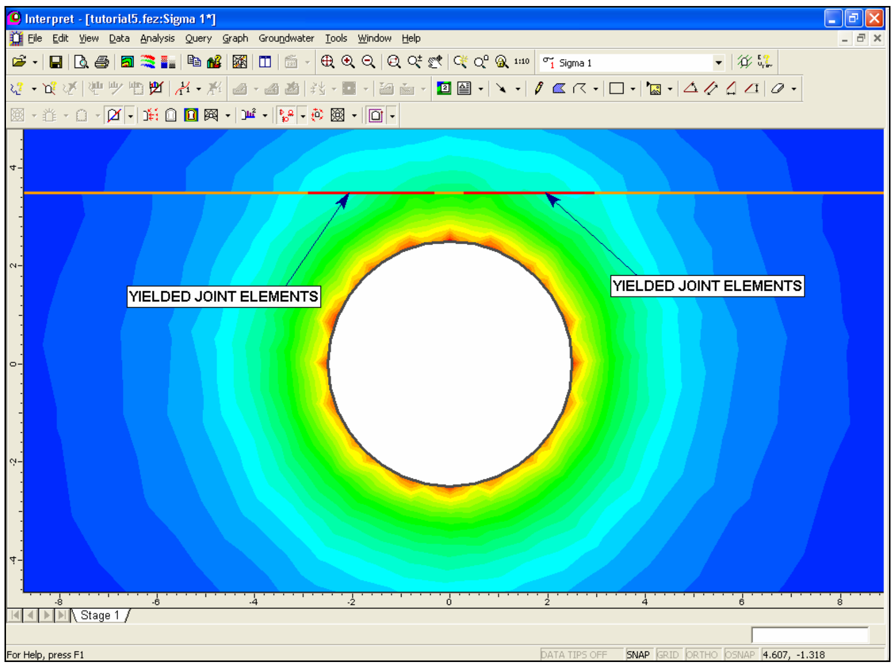


Figure 5-2: Yielded joint elements above excavation.

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.

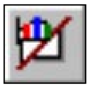
Let's quickly verify that there are 16 yielded joint elements. Right-click the mouse and select Display Options.

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

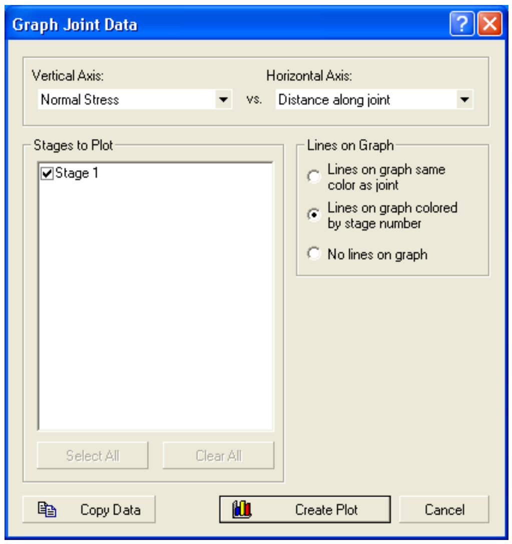
Graphs of normal stress, shear stress, normal displacement and shear displacement can be easily obtained for joints, using the Graph Joint Data option.



Select: Graph → Graph Joint Data

The Graph Joint Data option is also available if you right-click on a joint.

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



Just select Create Plot, to generate a plot of Normal Stress along the length of the joint.

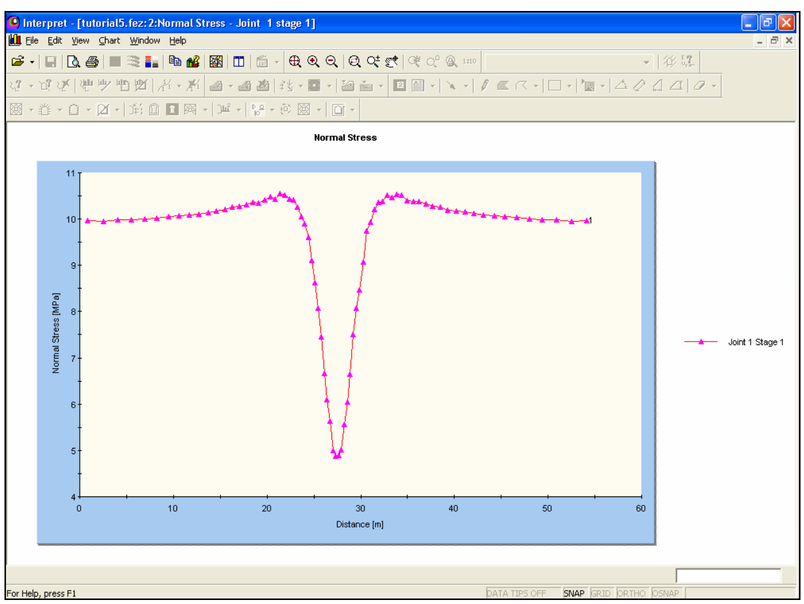


Figure 5-3: Normal stress along joint.

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.)

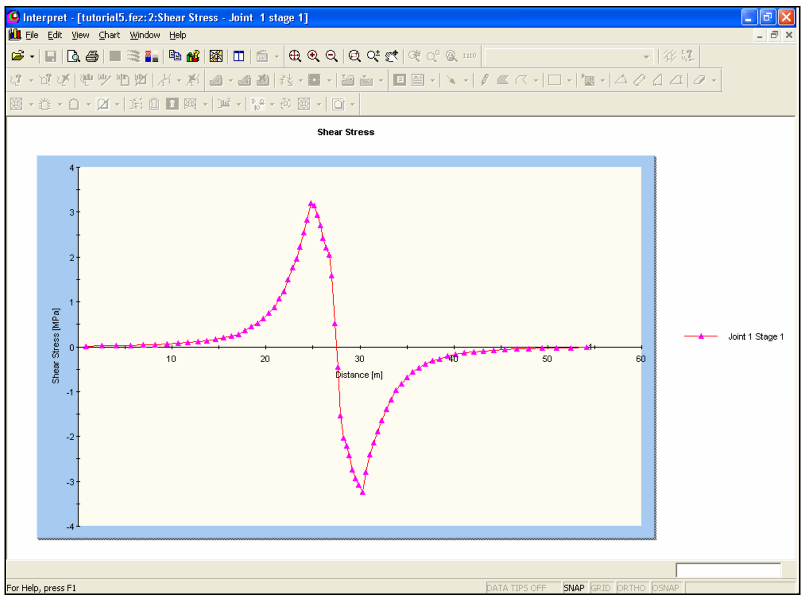


Figure 5-4: Shear stress along joint.

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°	8
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.