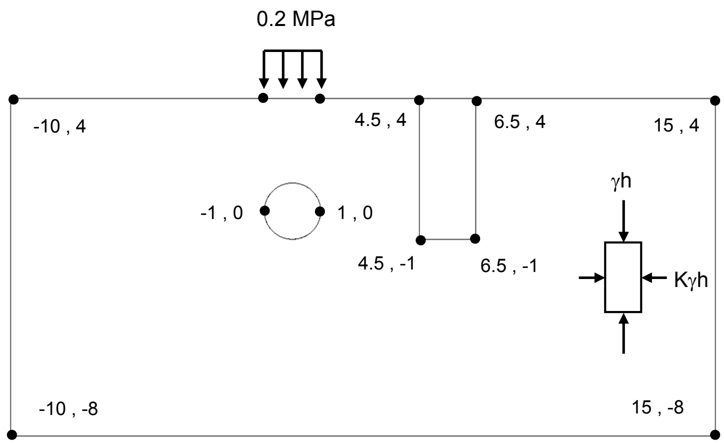
Surface Excavation Tutorial



This tutorial illustrates how to model a simple surface excavation, consisting of a trench located near a circular tunnel, and a distributed load 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 load in the third stage.

If you wish to skip the model process, the finished product of this tutorial can be found in the **Tutorial 04 Surface Excavation.fez** data file. All tutorial files installed with *RS2* 9.0 can be accessed by selecting File > Recent Folders > Tutorials Folder from the *RS2* main menu.

Model

If you have not already done so, run the *RS2* Model program by double-clicking on the *RS2* icon in your installation folder. Or from the Start menu, select Programs → Rocscience → RS2 9.0 → RS2.

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.

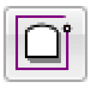
Select: Analysis → Project Settings

In the Project Settings dialog, select the Stages tab, enter Number of Stages = 3. For the stage names, enter Excavate Tunnel, Excavate Trench, Surface Load for stages 1, 2 and 3. Select OK.

Entering Boundaries

First enter the external boundary. A surface excavation will always require a user-defined external boundary. NOTE:

- If you select the Add External option *before* any excavation boundaries have been entered, then you can immediately begin entering coordinates.
- If you select the Add External option *after* defining excavation boundaries, then you will have to select the User Defined option in the Add External boundary dialog, in order to enter coordinates.

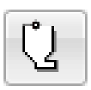


Select: Boundaries → Add External

```
Enter vertex[t=table,i=circle,esc=cancel]: 15 4
Enter vertex[...]: 1 4
Enter vertex[...]: -1 4
Enter vertex[...]: -10 4
Enter vertex[...]: -10 -8
Enter vertex[...]: 15 -8
Enter vertex[...,c=close,esc=cancel]: c
```

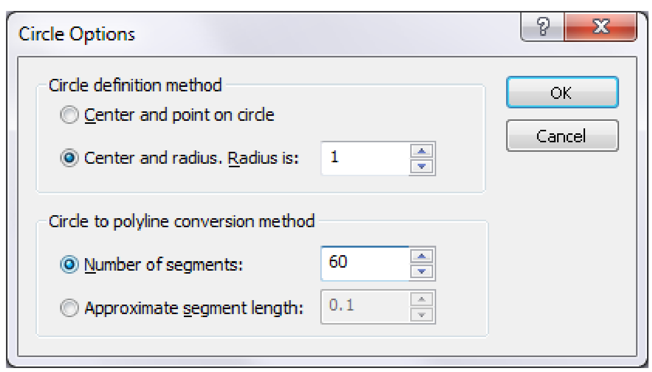
Press F2 to Zoom All.

Now enter the circular tunnel.



Select: Boundaries → Add Excavation

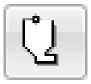
1. Right-click the mouse and select the Circle option from the popup menu. You will see the following dialog.



2. Select the Center and radius option, enter Number of Segments = 60 and select OK.

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

Now enter the rectangular trench.

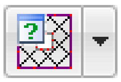


Select: Boundaries → Add Excavation

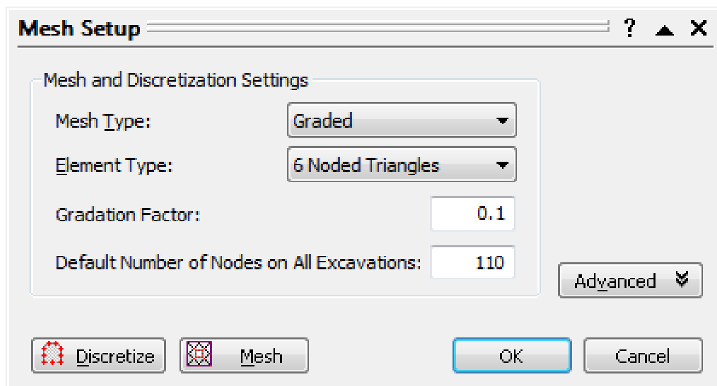
```
Enter vertex[t=table,i=circle,esc=cancel]: 4.5 4
Enter vertex[...]: 4.5 -1
Enter vertex[...]: 6.5 -1
Enter vertex[...]: 6.5 4
Enter vertex[...,c=close,esc=cancel]: c
```

Meshing

Let's look at the Mesh Setup dialog.



Select: Mesh → Mesh Setup



Notice the default number of excavation nodes = 110. Select the **Discretize** button in the dialog. The model is discretized, and the status bar will indicate the actual number of discretizations created.

Discretizations: Excavation: 136 External: 101

NOTE:

- The number of excavation discretizations is 136, but 110 was entered in the Mesh Setup dialog. Depending on your excavation geometry, the discretization algorithm will not always give you exactly the Number of Excavation Nodes you entered in Mesh Setup.

- Notice that *RS2* 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. Select the **Mesh** button in the dialog, and select OK. The mesh is generated based on the discretization you just created. The status bar will indicate the total number of nodes and elements in the mesh.

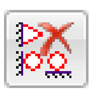
Note that the automatically graded discretization along the ground surface helps to create a smooth transition between the fine mesh at the top of the trench, and the rest of the ground surface.

Boundary Conditions

Select the **Loads & Restraints** workflow tab.

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.

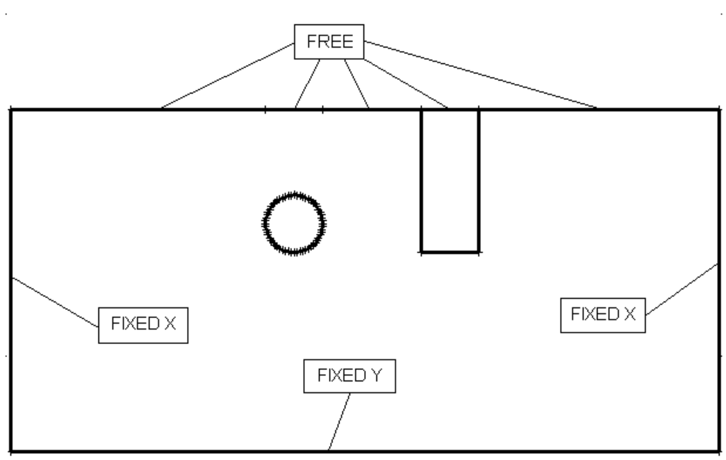
Since this is a surface excavation model, we must specify that the ground surface is a free surface. This is done using the **Free** option in the toolbar or Displacements menu.



Select: Displacements → Free

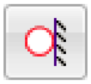
> Select boundary segments to free [enter=done, esc=cancel]: ***Use the mouse to select the five segments representing the ground surface. When finished, right-click and select Done Selection, or press Enter.***

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



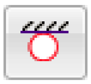
Displacement boundary conditions on external boundary.

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



Select: Displacements → Restrain X

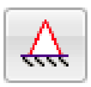
Select boundary segments to restrain in the X direction [enter=done,esc=cancel]: ***Use the mouse to select the left and right edges of the external boundary. Right-click and select Done Selection, or press Enter.***



Select: Displacements → Restrain Y

Select boundary segments to restrain in the Y direction [enter=done,esc=cancel]: ***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: Displacements → Restrain X,Y

1. Right-click the mouse and select Pick by Boundary Nodes 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.

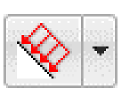
3. Right-click and select Done Selection. Triangular pin symbols now replace the roller symbols at these vertices.

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.

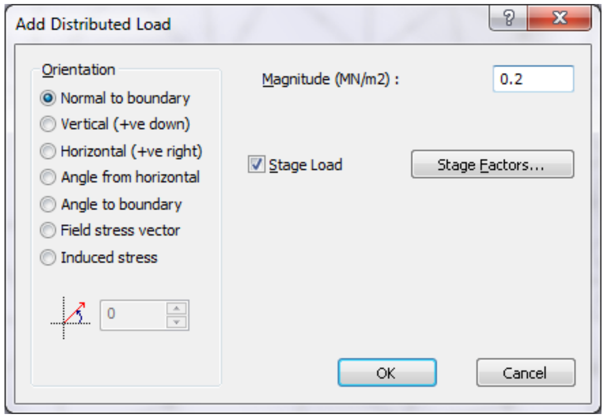
TIP: you can also apply restraints directly by right-clicking on segments or nodes and selecting a restraint option from the popup menu.

Adding a Distributed Load

Now let's add a uniform distributed load to the ground surface segment above the tunnel.

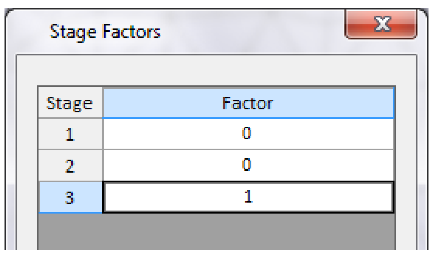


Select: Loading → Distributed Loads → Add Uniform Load



In the Add Distributed Load dialog, enter a Magnitude = 0.2 MN/m2. Select the Stage Load checkbox, and select the Stage Factors button.

In the Stage Factors dialog enter Factor = 0 for Stage 1 and Stage 2, and Factor = 1 for Stage 3. Select OK in both dialogs.



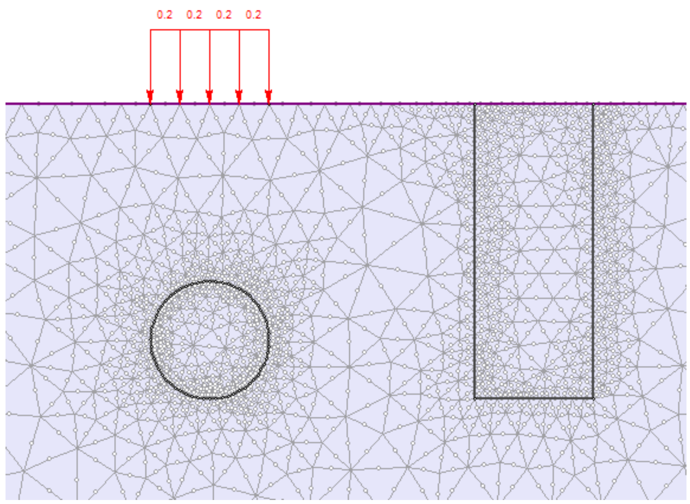
Because of the Factors we have defined, the load will only be applied in the third stage of the analysis, and will not exist in the first or second stages. Factor = 1 means the magnitude will be the same as entered in the Add Distributed Load dialog. Factor = 0 means no load will be applied at that stage.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

Now select the external boundary line segments to be loaded:

`Select boundary segments [enter=done,esc=cancel] :` ***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 Load

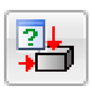
To view the load, select the Stage 3 tab. Since we only applied the load in Stage 3, it is only visible in Stage 3. For display purposes, the size of the load arrows can be scaled by the user in the Display Options dialog. This is left as an optional exercise. Since we are not finished modeling, select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



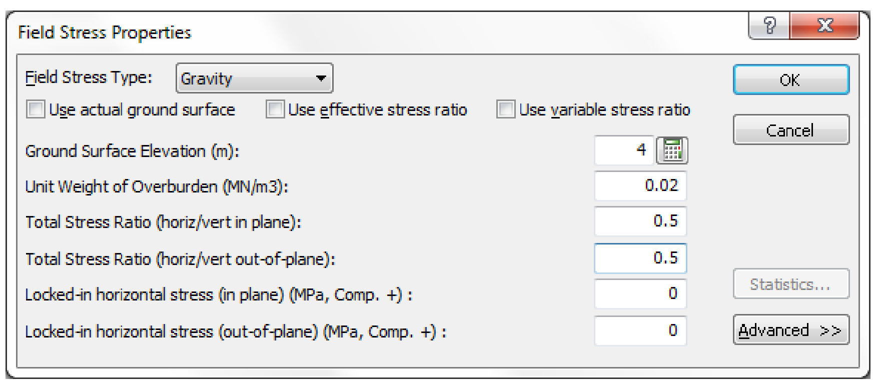
Distributed load added to model

Field Stress

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

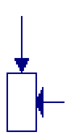


Select: Loading → Field Stress



Enter the above parameters and select OK. Note:

- The Ground Surface Elevation is indicated by a horizontal dotted 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.
- The Unit Weight of Overburden indicates that our material is a soil, rather than rock.

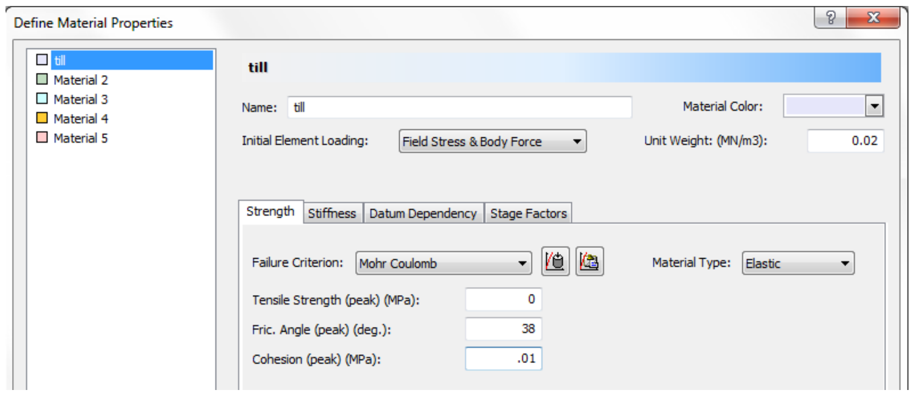
Properties

Select the Materials and Staging workflow tab.

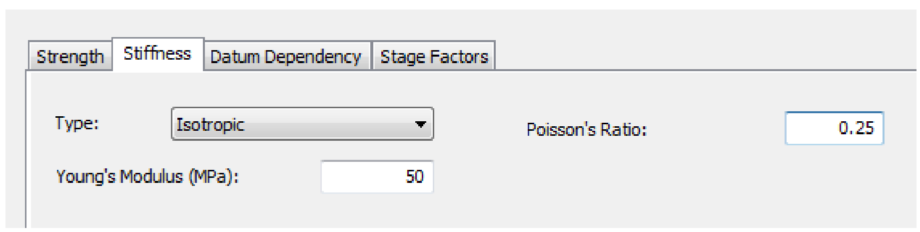
Select Define Materials from the toolbar or the Properties menu.

Select: Properties → Define Materials

With the first material selected, under the Strength tab enter the following: Name = till, Unit Weight = 0.02, Friction Angle (peak) = 38, Cohesion (peak) = 0.01.



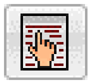
Under the Stiffness tab enter:



Note:

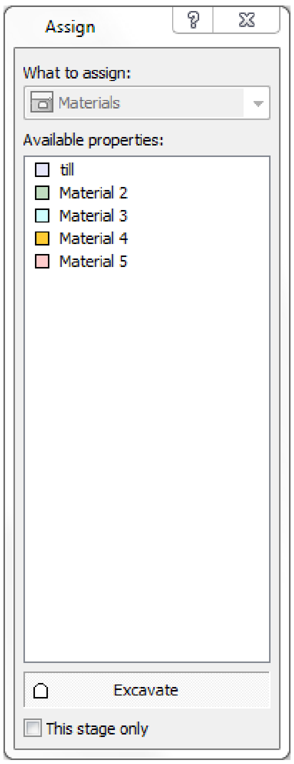
- the Unit Weight of the material is the same as the Unit Weight of Overburden entered in the Field Stress dialog.
- The modulus and strength values we entered are those of a till with high frictional strength.
- For gravity field stress, the default setting for 'Initial Element Loading' (in the Define Material Properties dialog) is '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 selected, we do not have to assign them to the model. By default, *RS2* 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.

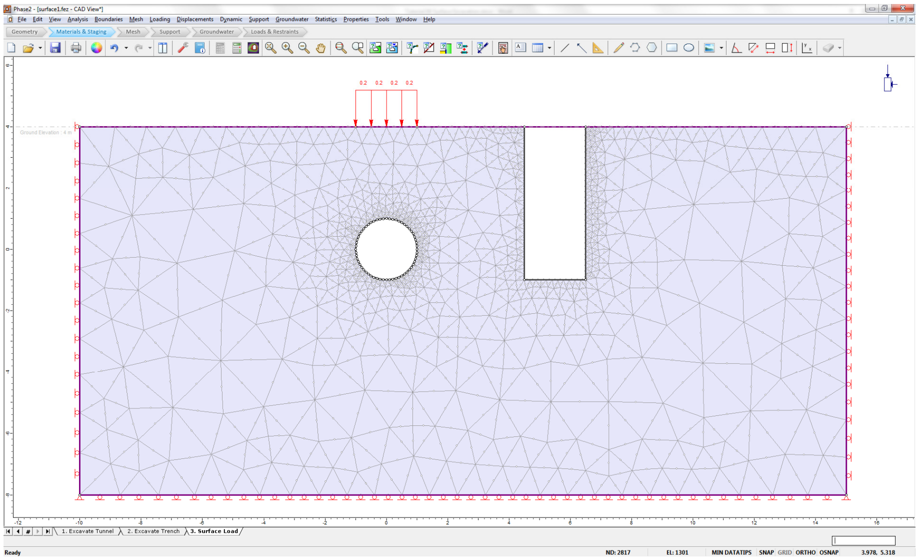
- Select Stage 1 – only the tunnel should be excavated.

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

- Select Stage 3 – the distributed load 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.



Finished model – RS2 Surface Excavation Tutorial

Compute

Before you analyze your model, save it as a file called **surface1.fez**.

Select: File → Save

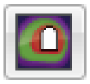
Use the Save As dialog to save the file. You are now ready to run the analysis.

Select: Analysis → Compute

The *RS2* 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

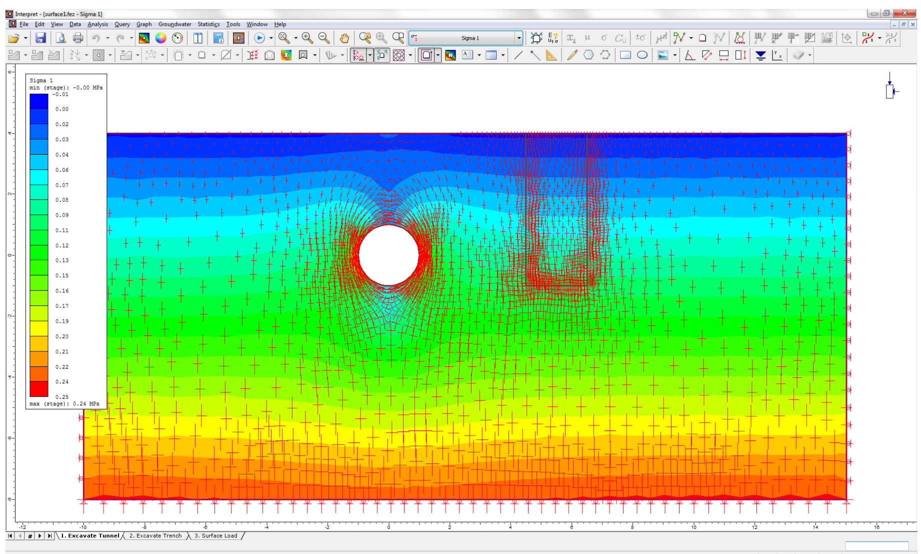
This will start the *RS2* Interpret program.

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 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.



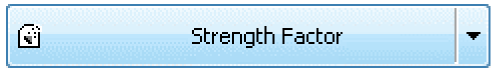
Sigma 1 contours and principal stress trajectories. Gravitational field stress is in effect.

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

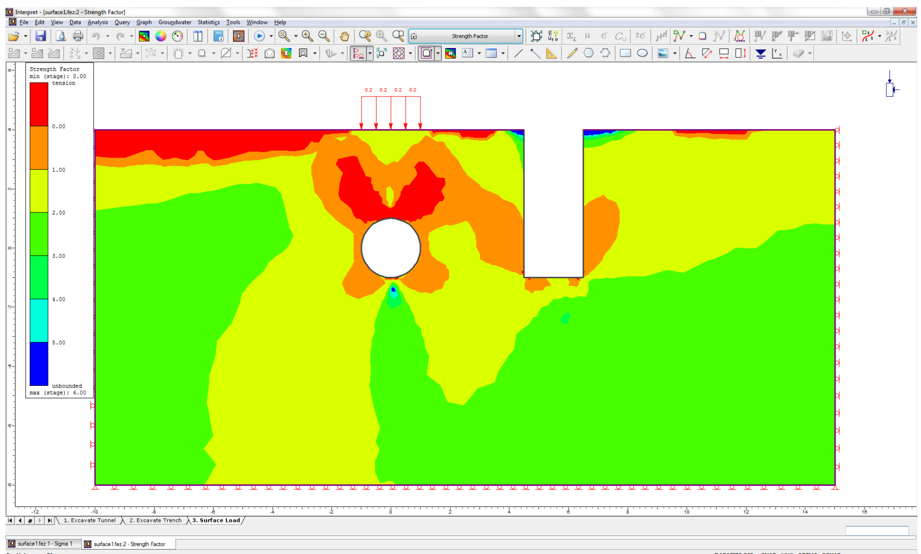
Select the Stage 2 and Stage 3 tabs to view the changing stress distribution with excavation of the trench and loading.

Strength Factor

Change the data type to Strength Factor, and change the number of Contour intervals to 7 in the Contour Options dialog.

Select: 

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 (i.e. contours with strength factor < 1 in orange, and tension zones in red).



Strength factor contours at Stage 3, indicating collapse of material around excavations.

Displacement

Now let's look at displacements.

Select: 

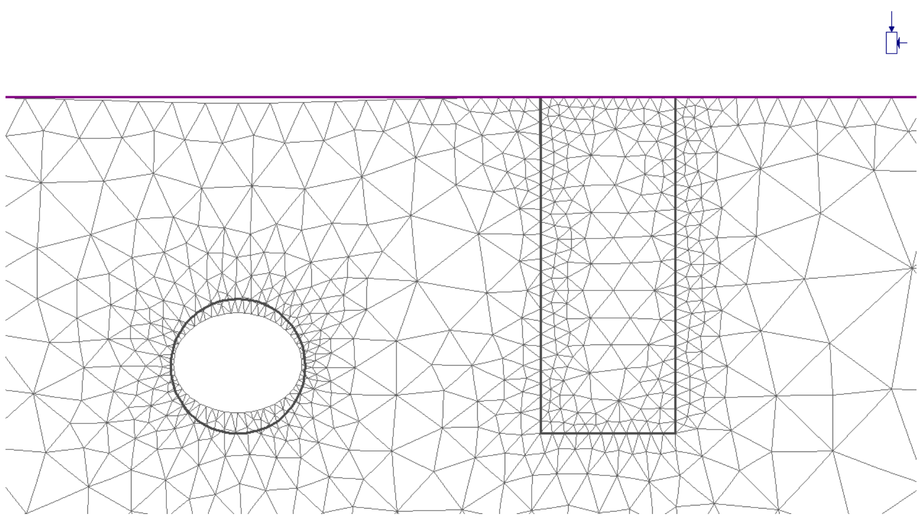
View the displacement contours at each stage, by selecting the stage tabs. Observe the maximum displacement, displayed in the status bar, and where it is occurring on the model.

The Stage 1 maximum displacement, about 3 mm, is occurring at the bottom of the tunnel. The Stage 2 maximum displacement, about 4.7 mm, is occurring at the left side of the trench. The Stage 3 maximum displacement, about 17 mm, is underneath the distributed load.

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.

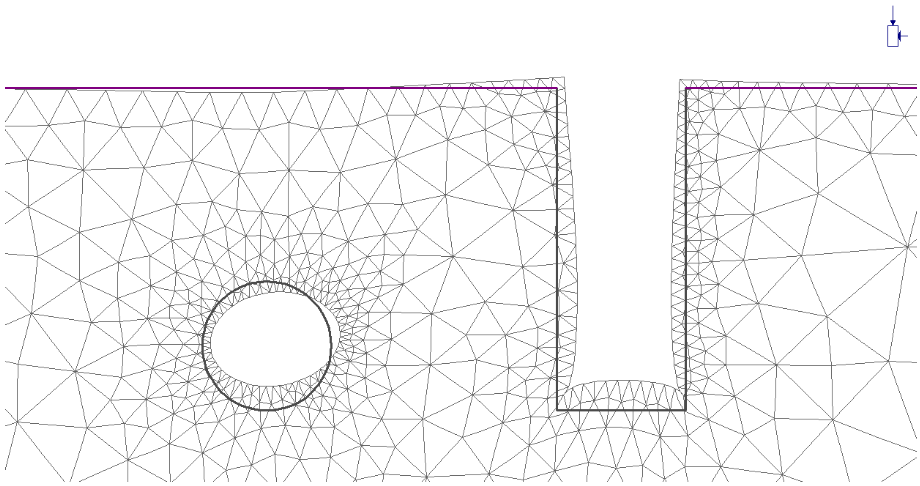
1. Right-click the mouse and select Contour Options.
2. In the Contour Options dialog, set the Mode to Off, and select Done.
3. Right-click the mouse and select Display Options.
4. In the Display Options dialog, select Deform Mesh and Deform Boundaries, and enter a Scale Factor of 100. Select Done.
5. Select Zoom Excavation (in the toolbar, or press F6).
6. Select the Stage 1 tab.



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.

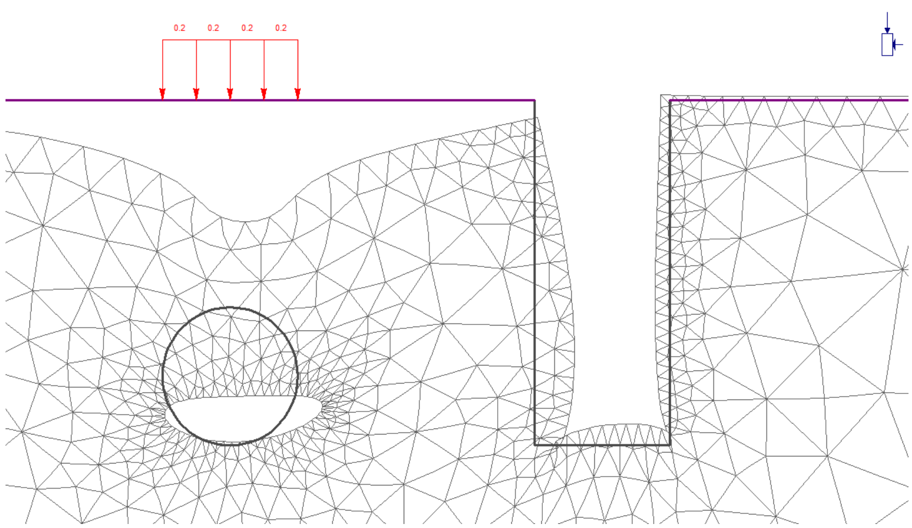


Deformed mesh and boundaries, stage 2. Displacements magnified by 100.

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 load. The maximum displacement is directly beneath the load. The overall displacement of the tunnel has been shifted downward, and the bottom of the tunnel is now almost in its original position.



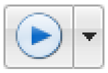
Deformed mesh and boundaries, stage 3. Displacements magnified by 100.

Let's "animate" the results. First, set the timing of the animation.

Select: Data → Stage Settings

In the Stage Settings dialog, set the Minimum Animation Time to 2 seconds. Select OK.

Now select the Animate Tabs option.



Select: Data → Animate Tabs

The stage tabs are now automatically selected for you, giving you an animated display of results at each stage.

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 toolbar.

- If you are not already zoomed in, press F6 to Zoom Excavation.

Query Data

RS2 allows you to query data anywhere in the material 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:

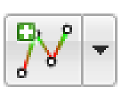
- Create a query, and show the values on the model

- 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

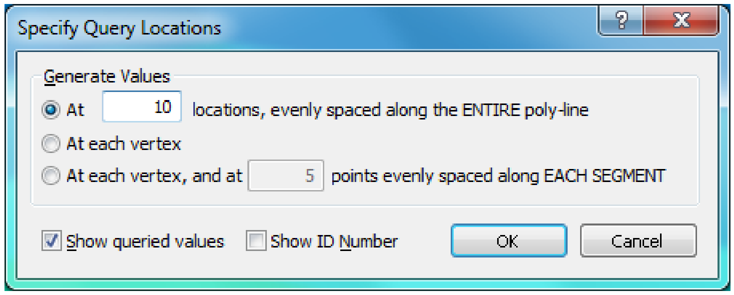
1. It will be handy to use the Snap option in this case, so right-click the mouse and make sure Snap is on.

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:



Choose the options shown above, and select OK.

6. You should now see 10 values along the left edge of the trench, since we entered 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 (e.g., 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.

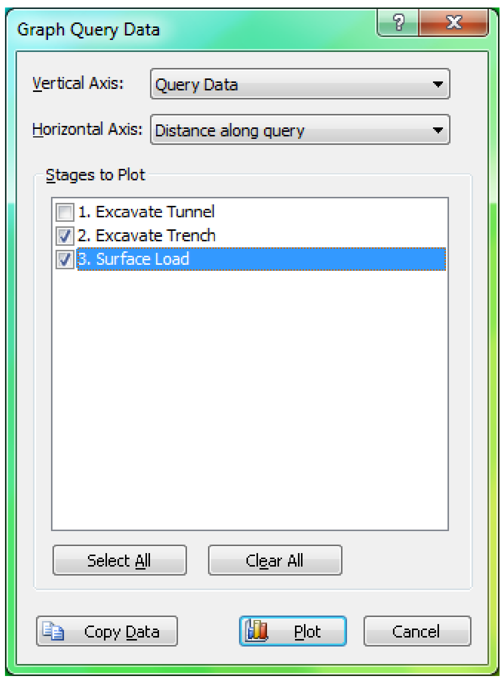
NOTE: the number of decimal places can be independently specified for each data type, and *RS2* will "remember" this information, so you do not have to reset the number of decimal places each time you use the program.

Graphing a Query

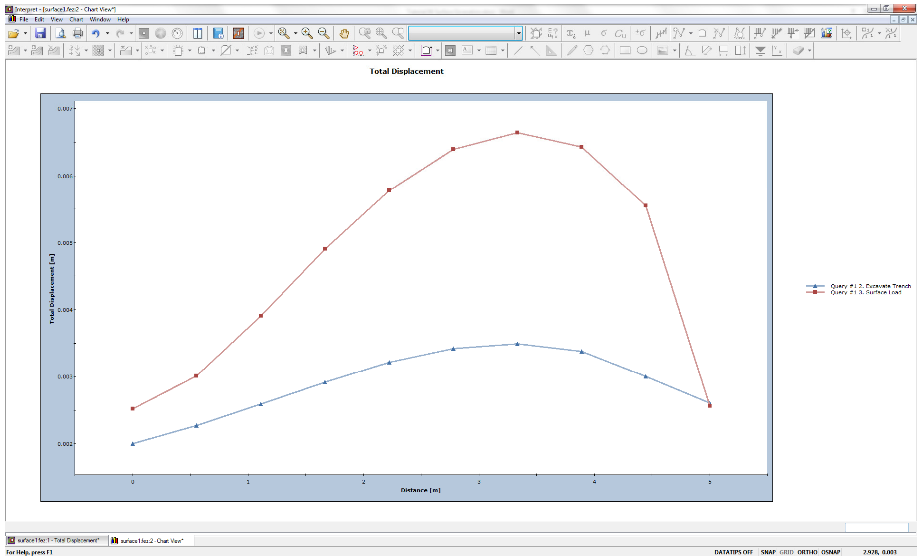
A short cut for graphing data for a single query, is to right-click on the query and select Graph Data.

1. Right click on the query (i.e., 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 Plot button, and a graph of Total Displacement along the query, for both Stage 2 and Stage 3, will be generated.



Trench wall displacement, before and after adding surface load.

This graph shows the “before” and “after” effect of the distributed load 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. We will now edit the query and generate a new graph.

Editing a Query

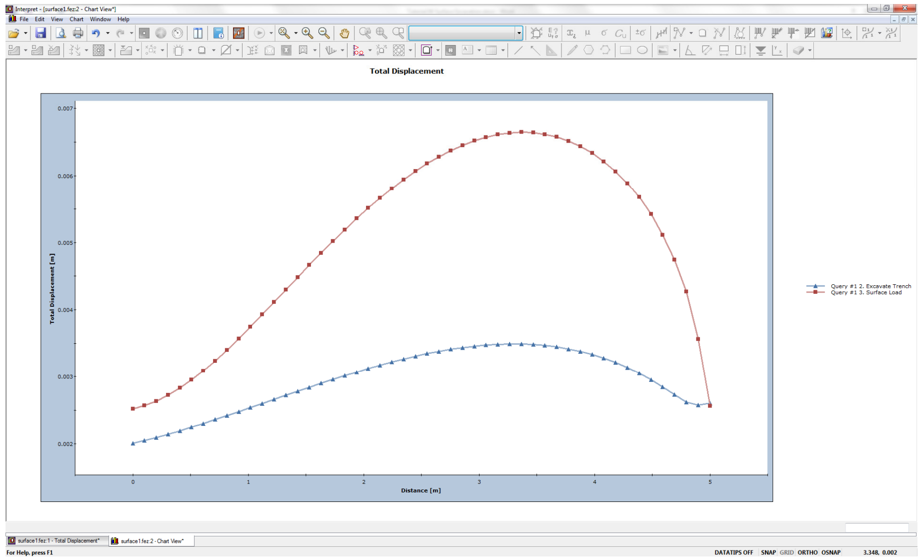
To edit the query:

1. Right click on the query, and select Edit Locations 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.



Trench wall displacement using 50 query locations

Finally, note that the axis ranges and titles can be modified by right-clicking on the graph and selecting Chart Properties. Many other chart options are also available in the right-click menu. This 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.

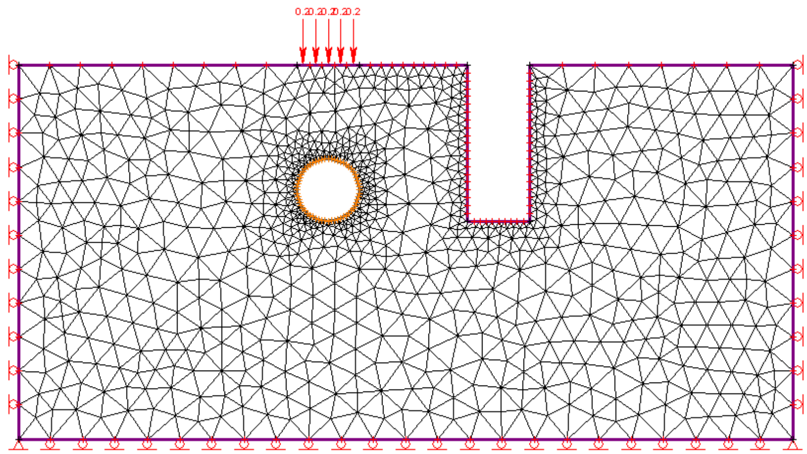
Additional Exercises

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 Number of Excavation Nodes = 60, since the only 'excavation' in this model (according to the *RS2* 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.



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 (i.e. an elastic liner, 0.1 m thickness, modulus = 30000 Mpa). For details about adding liners to excavations, see the Support Tutorial – Step 2.

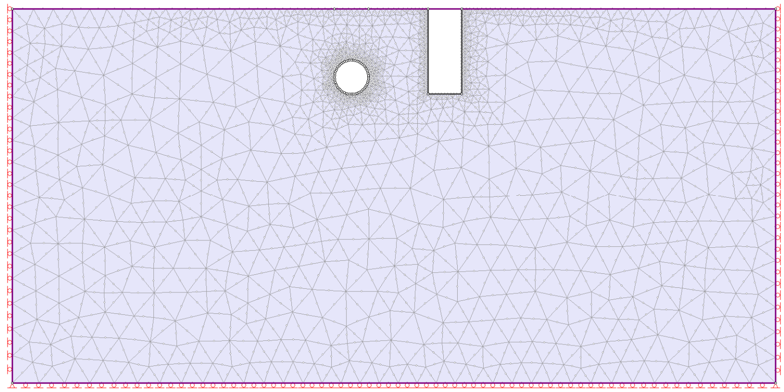
When you look at the analysis results, notice that the zone of tension around the tunnel is reduced in the Strength Factor contour plot, and also the displacements are reduced.

(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 influencing the contours somewhat, at the left, right and bottom edges 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.



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	.0036
Stage 2	.0047	.0065
Stage 3	.0178	.0191

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.