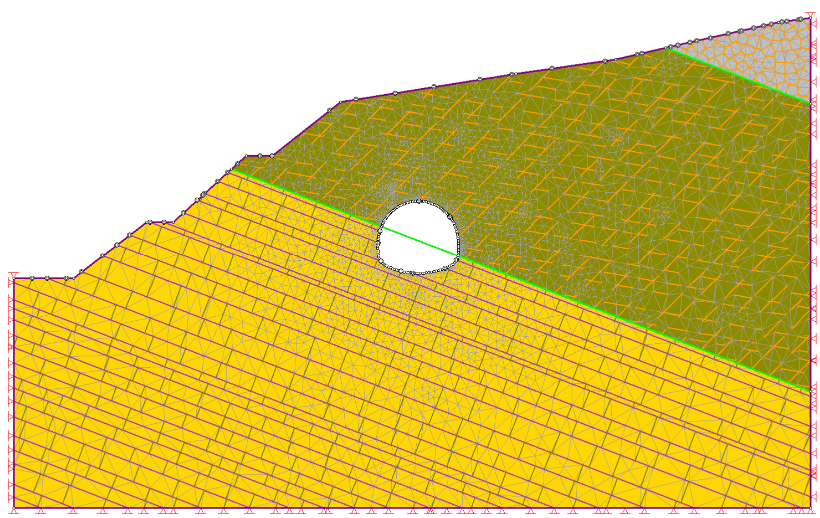
Application of Joint Networks

This tutorial will demonstrate how to specify joint networks or discrete fracture networks (DFN) in *RS2* using the automatic joint network generator. It will also demonstrate a few techniques for analyzing the effects of the joints on model results. The model involves the stability analysis of a transportation tunnel near a slope in blocky rock. There are three different zones of blocky rock in the model.

The finished *RS2* model of this tutorial can be found in the **Tutorial 29 Application of Joint Networks.fez** file, which can be accessed by selecting File > Recent Folders > Tutorials Folder from the *RS2* main menu.

Topics Covered

- Adding joint networks to different model regions
- Specifying joint end conditions
- Interpreting joint movements



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.

Open the file **Tutorial 29 Joint Networks (initial).fez**.

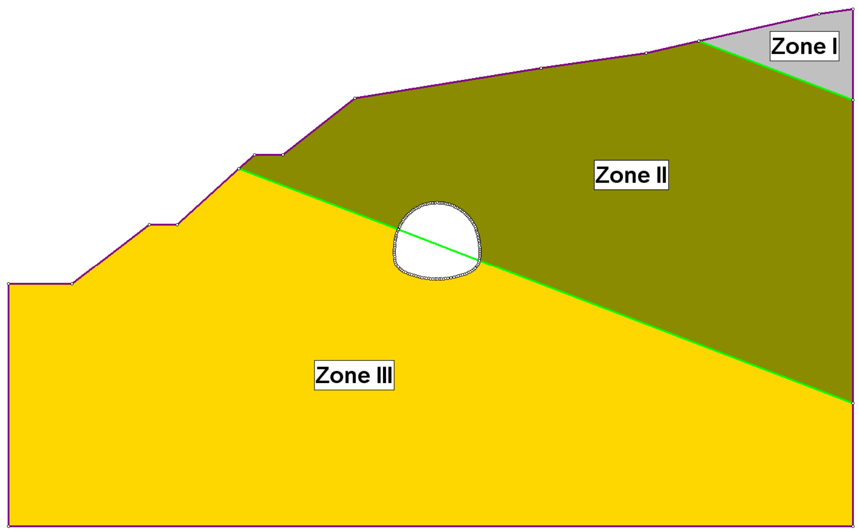
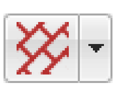


Figure 29-1: Basic geometry (startup model) for the tutorial example.

As seen in Figure 29-1 above, the model consists of three material zones. (For the rest of the tutorial we will refer to these zones as Zones I, II and III as labeled on the figure.) In each of these zones we will be applying a joint network.

Adding a Voronoi Joint Network to Zone I

We will apply a joint network to the zone at the upper right corner of the model.



Select: Boundaries → Joint Networks → Add Joint Network

Notice that the mouse cursor immediately changes shape. Click the left mouse button at any location within Zone I. A hatched pattern appears in the selected zone. Hit Enter or right-click and select Done to complete zone selection. The **Add Joint Network** dialog (Figure 29-2) pops up.

In the dialog, input data fields are grouped under headings. Labels (descriptions) of the input fields are shown on the left cells while the corresponding actual input fields are on the right.

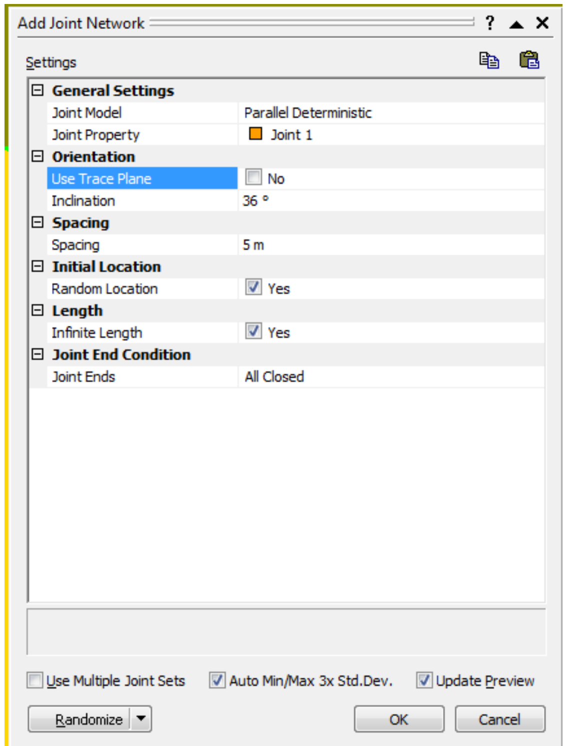


Figure 29-2: Input dialog for joint networks.

Towards the bottom of the dialog, there are three checkboxes. We shall examine their roles in subsequent sections of this tutorial.

Automatic Preview of Joint Networks

The third checkbox in the lower section of the joint network dialog is labeled **Update Preview**. A preview of the currently active joint network is displayed on the *RS2* model being developed, as soon as the joint network dialog appears. When the **Update Preview** checkbox is on (the default status) a joint network is immediately redrawn the moment any change is made to a parameter in the dialog. When it is off, the preview remains but will not be updated until the dialog is closed. Leave the **Update Preview** checkbox on.

The network is shown in a rectangular window slightly larger than the selected zone. In addition to allowing you to see the geometry of the joint network you are specifying, the preview feature allows you to zoom into any part of a model for a closer look without leaving the joint network dialog.

We will zoom into an area of the model around Zone I to take a closer look at the currently displayed joint network. You can use key strokes and shortcuts shown in the table below to zoom around Zone I.

Action	**Shortcut Key**
Zoom In	Home
Zoom Out	End
Pan to the Left	Left arrow
Pan to the Right	Right arrow
Pan Up	Up arrow
Pan Down	Down arrow
Zoom All	F2
Zoom Excavation	F6

Click anywhere on the *RS2* model. Use any combination of the **Home** and **Arrow** keys to zoom into Zone I. Press **F2** when you are done to reset the view to the model extents.

Specifying Input for Voronoi Joint Network

Under the **General Settings** heading in the dialog, click on the **Joint Model** input field (the cell on the right with the descriptor **Parallel Deterministic**). From the resulting dropdown list select the **Voronoi** joint model. Notice that the joint network displayed on the model immediately changes. (Depending on the speed of your computer the preview may not be as quick.)

Change the Density (number of Voronoi cells per unit area) to 0.4. Once the network of Voronoi cells is displayed, click on the Regularity input field and change the option from **Irregular** to **Medium Regular**.

Changing Joint End Conditions

By default, the ends of all joints in a network are assumed to be closed, i.e. no relative movement (joint sliding or opening) can occur at a joint end. Options exist in the joint network dialog for removing this constraint. In our example, we will specify that joint ends be open at the ground surface and at intersections with the excavation (tunnel).

Click on the input data cell (cell with the default end condition setting of **All Closed**). Select **Open at Boundary Contacts** from the dropdown list. Once this option is selected, five new rows of data appear. Each row describes end conditions for joints that intersect a type of boundary. (There are five types of boundaries in *RS2*.)

In a model, the first time the **Open at Boundary Contact** option is selected, joint ends are specified as open at the ground surface and at excavation boundaries. Since this is what we are interested in, we will accept these defaults. Your dialog should now look like exactly like that shown on Figure 29-3 Select the **OK** button to close the dialog.

This completes the specification of a joint network for Zone I.

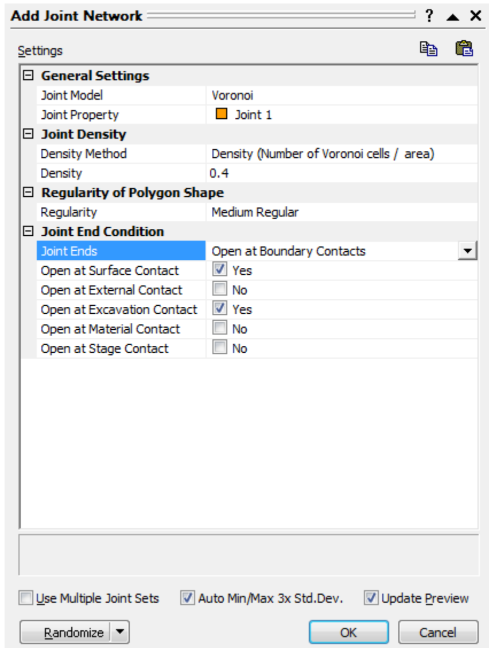


Figure 29-3: Appearance of dialog after input of all parameters for Voronoi joint network.

Adding a Network Comprising 2 Joint Sets to Zone II

We shall next apply a joint network to Zone II of the model. This time the network will comprise two sets of parallel joints.

Select: Boundaries → Joint Networks → Add Joint Network

Click the left mouse button anywhere within Zone II and hit Enter to complete selection. The joint network dialog pops up.

As mentioned earlier, there are three checkboxes towards the bottom of the joint networks dialog. We already looked at the function of the third (**Update Preview**) checkbox. We shall now examine the roles of the first two in the specification of our new joint network.

The first checkbox with the label **Use Multiple Joint Sets** allows us to specify a network with more than one joint set. The second checkbox, with the description **Auto Min/Max 3x Std. Dev.**, is on by default. It automatically calculates lower and upper bounds (relative minimum and relative maximum values) for the values generated from a statistical distribution based on the specified standard deviation.

As its name suggests, the **Auto Min/Max 3x Std. Dev.** option calculates both relative minimum and relative maximum as three times the standard deviation. However there are exceptions. If a relative minimum or maximum calculated this way will result in an invalid bounding value (for example, if it leads to negative spacing) the minimum or maximum will be assigned a lower value that maintains a valid bound.

Select the **Use Multiple Joint Sets** checkbox. A new panel appears on the left side of the dialog as soon as the option is turned on. The panel shows an automatically selected joint set – Joint Set 1.

Setting the Parameters for Joint Set 1

For Joint Set 1, change the joint model to **Parallel Statistical**. Leave the joint property field as Joint 1. Under the Orientation section of the dialog leave the **Use Trace Plane** option as the default of No. Enter an inclination angle of 45^o.

Next, we are going to specify a distribution for the spacing of the parallel joints in Set 1. Enter a mean spacing of 3 m. Leave the choice of distribution as Normal. Set the standard deviation to 0.8.

Because the **Auto Min/Max 3x Std. Dev.** option is on, the relative minimum and relative maximum values for the distribution are automatically calculated. Both relative minimum and relative maximum are automatically set to a value of 2.4 (3 x 0.8).

The next set of parameters we will input relate to the lengths of the joints. Change the **Infinite Length** option from Yes to No. Once you do this several new input data fields appear for entering parameters for the distribution of joint lengths. Specify the following values:

- Mean = 4 m
- Distribution = Lognormal, and
- Standard deviation = 1 m.

The relative minimum and maximum values are again set automatically.

For persistence, specify the following distribution parameters:

- Mean = 0.7
- Distribution = Normal, and
- Standard deviation = 0.1.

The relative minimum is automatically set to 0.3 (3 x 0.1). The relative maximum is however set to only 0.2. This is because a persistence of 1 is not allowed.

Change the **Joint End Condition** to **Open at Boundary Contacts** as we did for the previous network. This completes the definition of Joint Set 1.

Setting the Parameters for Joint Set 2

When we selected the **Use Multiple Joint Sets** option, two buttons labeled **Add** and **Delete** appeared beneath the left panel with joint sets information. These buttons are for adding sets to a network or deleting them.

Click on the **Add** button. A second joint set labeled Joint Set 2 shows up on the left panel just below Joint Set 1. It is highlighted. This indicates that Joint Set 2 is active and its parameters can be specified or altered.

Notice that there are checkmarks beside the joint set labels. A checkmark indicates that a joint set is actually drawn (or applied) in the selected material zone. If it is deselected, although the data (parameters) for the joint set remain, joints from the set are not applied to the material zone. We will leave the checkmarks as they are.

For Joint Set 2 input the parameters shown in Table 29-1. Leave all other parameters as the defaults provided in the dialog.

Click on the **OK** button to complete the definition of the joint network for Zone II.

Table 29-1: Input parameters for Joint Set 2 (for Zone II)

Parameter	Value
General Settings	
Joint Model	Parallel Statistical
Joint Property	Joint 1
Orientation	
Use Trace Plane	No
Inclination	-10^{o}
Spacing	
Mean	2 m
Distribution	Normal
Standard deviation	0.5
Length	
Infinite Length	No
Mean	2 m
Distribution	Normal
Standard deviation	1 m
Persistence	
Mean	0.5
Distribution	Normal
Standard deviation	0.1
Joint End Condition	
Joint Ends	Open at Boundary Contacts

Adding a Cross-Jointed Network to Zone III

We shall next apply a fracture network consisting of bedding planes with cross joints to Zone III.

Select: Boundaries → Joint Networks → Add Joint Network

Click the left mouse button anywhere within Zone III and hit enter to complete selection. The joint network dialog pops up.

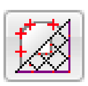
Specify the parameters indicated in Table 29-2 for the cross-jointed network. Leave any other parameters not specified as the defaults provided in the dialog.

Click on the **OK** button to close the joint network dialog and apply the joint network to Zone III.

This completes the specification of joint networks for our model.

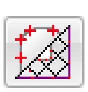
Table 29-2: Input parameters for Cross-Jointed Network (for Zone III)

Parameter	**Value**
General Settings	
Joint Model	Cross Jointed
Bedding Joint Property	Joint 2
Cross Joint Property	Joint 3
Orientation	
Use Trace Plane	No
Bedding Inclination	-21°
Cross Joint Inclination	69°
Bedding Spacing	
Mean	2 m
Distribution	Normal
Standard deviation	0.8
Cross Joint Spacing	
Mean	5 m
Distribution	Normal
Standard deviation	1.0
Joint End Condition	
Joint Ends	Open at Boundary Contacts



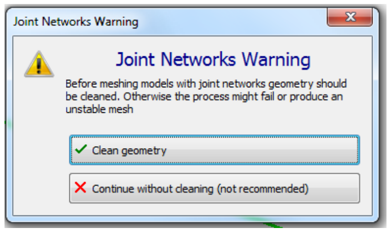
Meshing

We will use the **Discretize and Mesh** option in *RS2* to automatically discretize the boundaries in our model and generate a mesh with one mouse click.



Select: Mesh → Discretize & Mesh

If a "Joint Networks Warning" dialog appears, click "Clean geometry". This process is recommended when using joint networks to ensure good mesh quality.



In the Geometry Cleanup dialog, accept the default values and click OK. Select Yes to the discretization and mesh prompt which follows.

While the mesh is being generated the mesh generation status window shown below opens up.

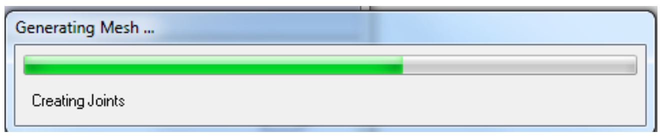


Figure 29-4: Status window indicating progress of mesh generation.

Upon completion of the mesh generation process the status window disappears, and your model should appear like the image in Figure 29-5.

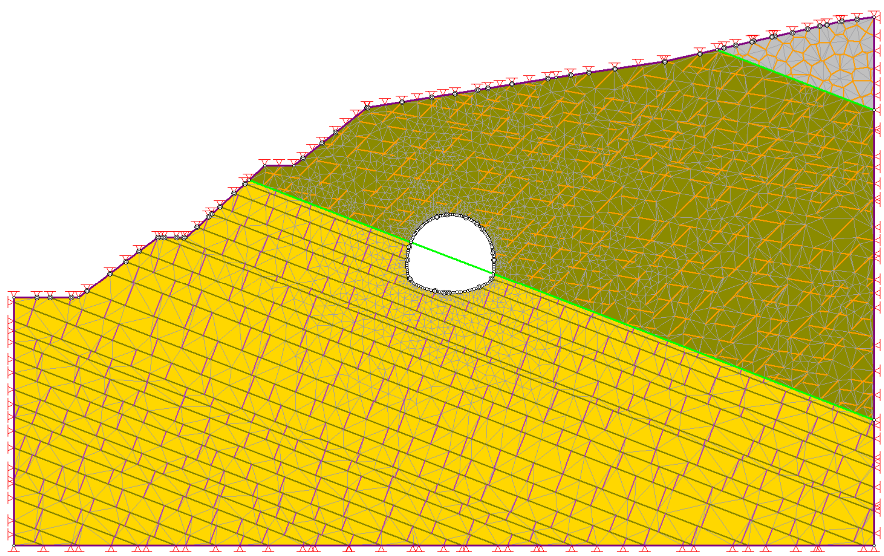
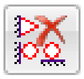


Figure 29-5: Appearance of model at end of mesh generation.

Boundary Conditions

In Figure 29-5 we see that the slope (ground) surface is not free; it has pinned (fixed, zero displacement) boundary conditions. This occurs because by default all nodes on the external boundary are pinned.

To remove these conditions select the **Free** option in the **Displacements** menu.



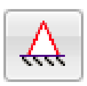
Select: Displacements → Free

The following message appears in the status window at the bottom right corner of the program:

```
Select boundary segments to free [enter=done, esc=cancel]:
```

Use the mouse to select all the line segments that define the ground surface. When finished, right-click on the mouse and select Done Selection, or simply press **Enter**. The triangular pin symbols should now be gone from the slope surface.

It may be necessary to re-apply the pinned boundary conditions to the uppermost nodes of the left and right external boundaries of the model. To do so:



Select: Displacements → Restrain X,Y

1. Right-click the mouse. Select **Pick by Boundary Nodes** from the resulting popup menu. This will change the mode of application of boundary conditions from segments to nodes.
2. Select the topmost vertex of the left external boundary with coordinates (-50, 34.5) and the corresponding vertex on the right boundary with coordinates (70, 73.761).
3. Right-click and select **Done Selection**. Triangular pin symbols now appear at these vertices.

Your final model should now look like Figure 29-6 below:

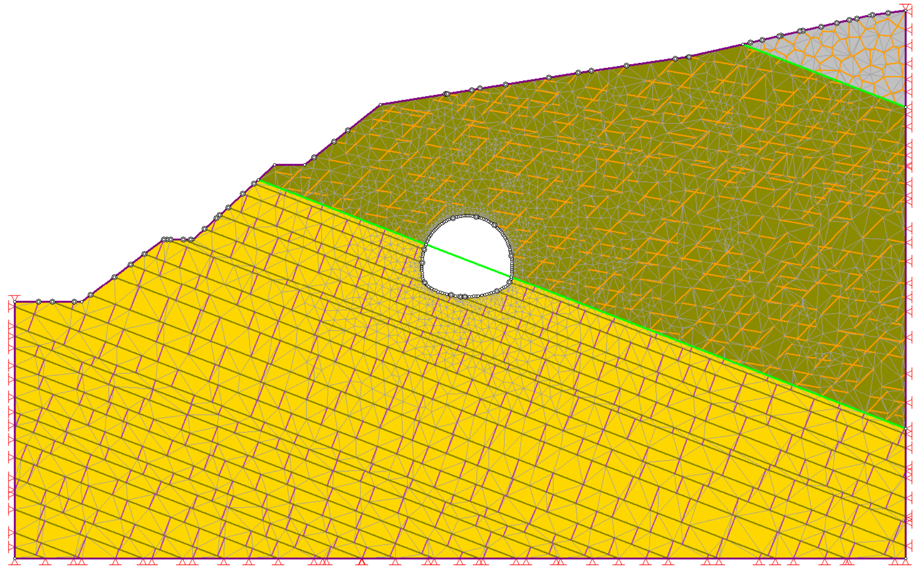
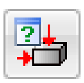


Figure 29-6: Final slope with tunnel model after specification of all appropriate boundary conditions.

Field Stress

For this tutorial we will specify a gravitational field stress. This assumption is reasonable for slope problems or surface and near-surface excavations.



Select: Loading → Field Stress

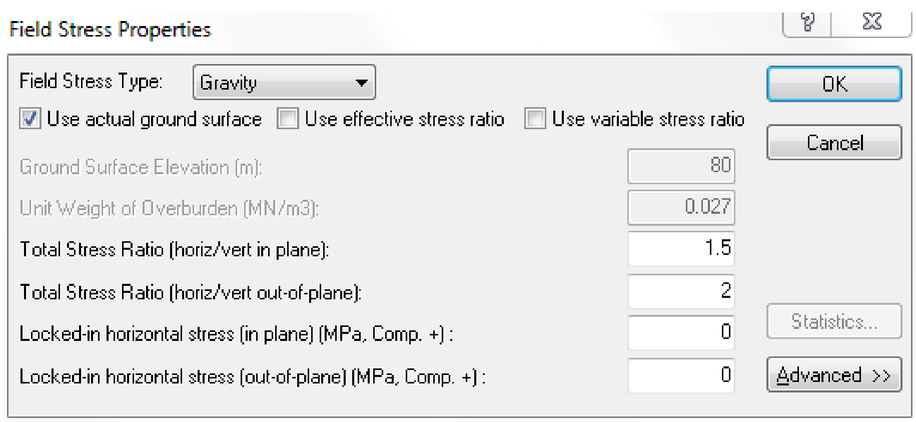


Figure 29-7: Dialog with field stress properties specified for the tutorial.

Enter the parameters indicated in the field stress dialog (Figure 29-7) above. Select **OK**. (Notice that in this tutorial we assume horizontal stresses to be larger than the vertical stresses.)

Material and Joint Properties

The material and joint strength and deformation properties used in this tutorial have already been provided in the starting file **Tutorial 29 (initial).fez**. If you would like to see the material properties assumed in the model you can do the following:

Select: Properties → Define Materials

Click on the tabs named Zone I, Zone II and Zone III to see what the properties are.

To review the joint properties:

Select: Properties → Define Joints

Note that all the materials and joints are assigned plastic properties, i.e. they will fail if the stresses at a location exceed the material or joint strength.

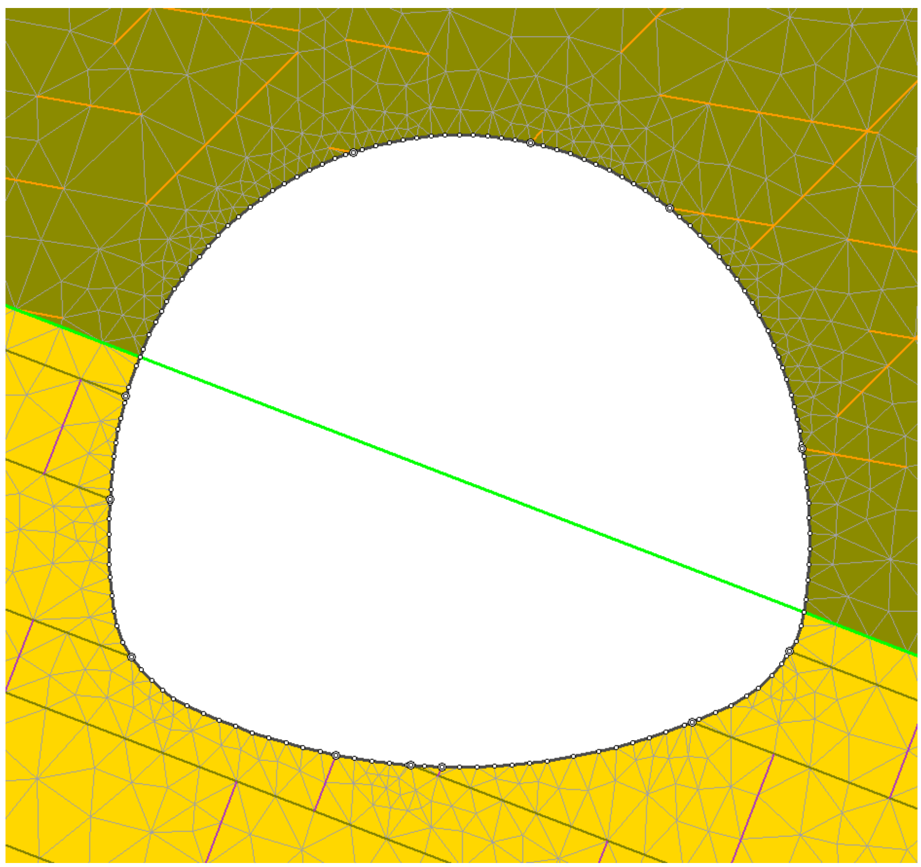


Figure 29-8: Representation of open ended joints with concentric circles.

Visual Representation of Joint End Conditions

Because we specified joint ends to be open at the ground surface and at excavation boundaries, any such ends (nodes) are represented by a symbol of two concentric circles.

As an example, we shall zoom in on the tunnel to look how the ends of joints that intersect the excavation boundary are represented.

Select: View → Zoom → Zoom Excavation

Your view should be similar to that in Figure 29-8. The concentric circle symbols representing open joint ends are visible on the excavation boundary, wherever a joint intersects the excavation.

Compute

Before you analyze the model, you must save it under any name of your choice.

Select: File → Save As

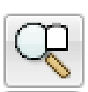
Enter your file name (e.g. ***Tutorial 29 – Final Model .fez***). You are now ready to run the analysis.

Select: Analysis → Compute

The *RS2* Compute engine will begin to run the analysis. Since we are using Plastic materials and joints, depending on the speed of your computer, the analysis may take some time.

Once the computation is done, you can view the results in the *RS2* Interpret.

Interpret



To view the results of the analysis:

Select: Analysis → Interpret

This will start the *RS2* Interpret program.

We will briefly look at aids for interpreting how networks of joints affect stresses, strains and displacements in the model.

Contours of Major Principal Stress – Sigma1

When Interpret first opens up it displays contours of major principal stress, Sigma 1. Go to Contour Options, choose Custom Range, change the Max value to 3 and the number of Intervals to 20, and select Done.

You should see the following contours (Figure 29-9). The effects of the joints on the Sigma 1 contours are visible – the contours are jagged and not as smooth as those for models without joints.

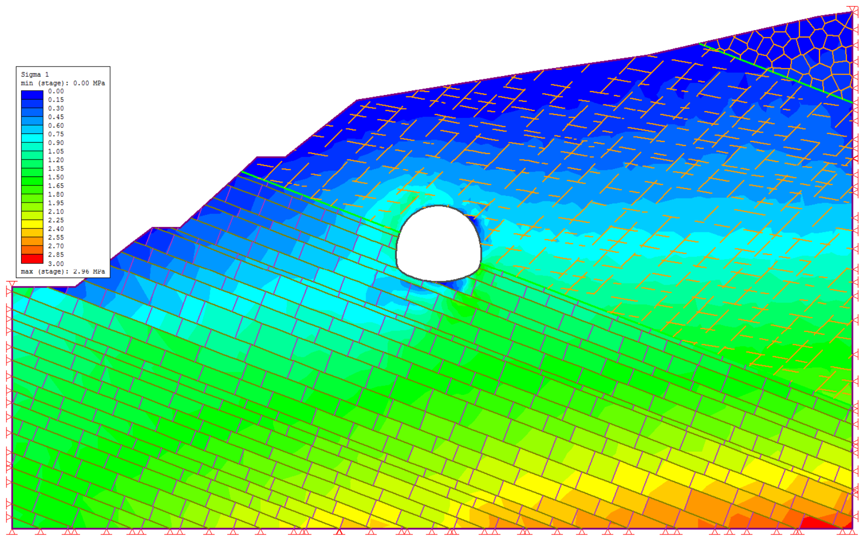


Figure 29-9: Sigma 1 contours. Notice the jagged nature of the contours.

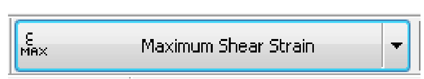
Contours of Maximum Shear Strain

We will next look at contours of maximum shear strain. On the Interpret toolbar

Click on Sigma 1:

From the resulting list of quantities

Select:



The contours reveal that very little shear strain occurs in the intact rock materials (most of the model is coloured in blue). Most of the shear displacements must be occurring along the joints.

We shall zoom in on the excavation and examine the distribution of shear straining in that region.

Select: View → Zoom → Zoom Excavation

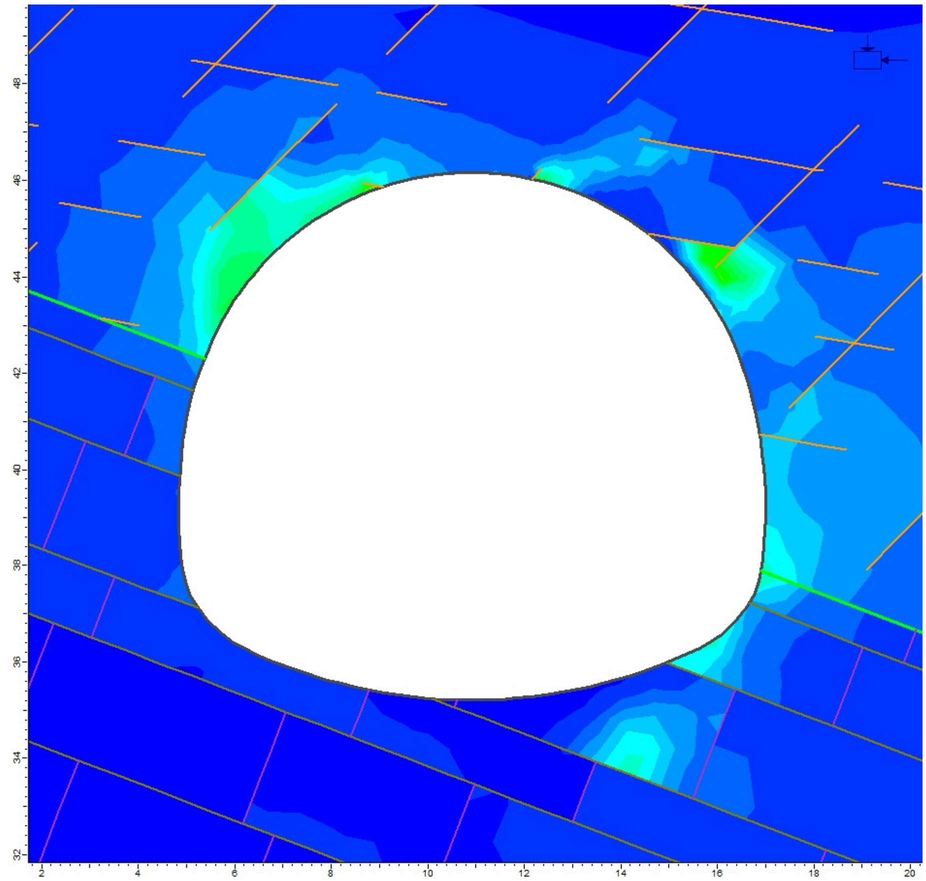


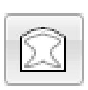
Figure 29-10: Contours of maximum shear strain in vicinity of tunnel.

In Figure 29-10 it can be seen that concentrations of shear strain occur at some joint ends and joint intersections with the tunnel.

Deformed Boundary

The *RS2* Interpret program can display an exaggerated view of the deformed shapes of excavation, joint and external boundaries. This feature is very useful in understanding behaviour.

Select: View → Display Options



In the resulting dialog select the Deform Boundaries checkbox. You can access the Deform Boundaries option quicker by simply clicking on the **Display Deformed Boundaries** toolbar button shown in the margin.

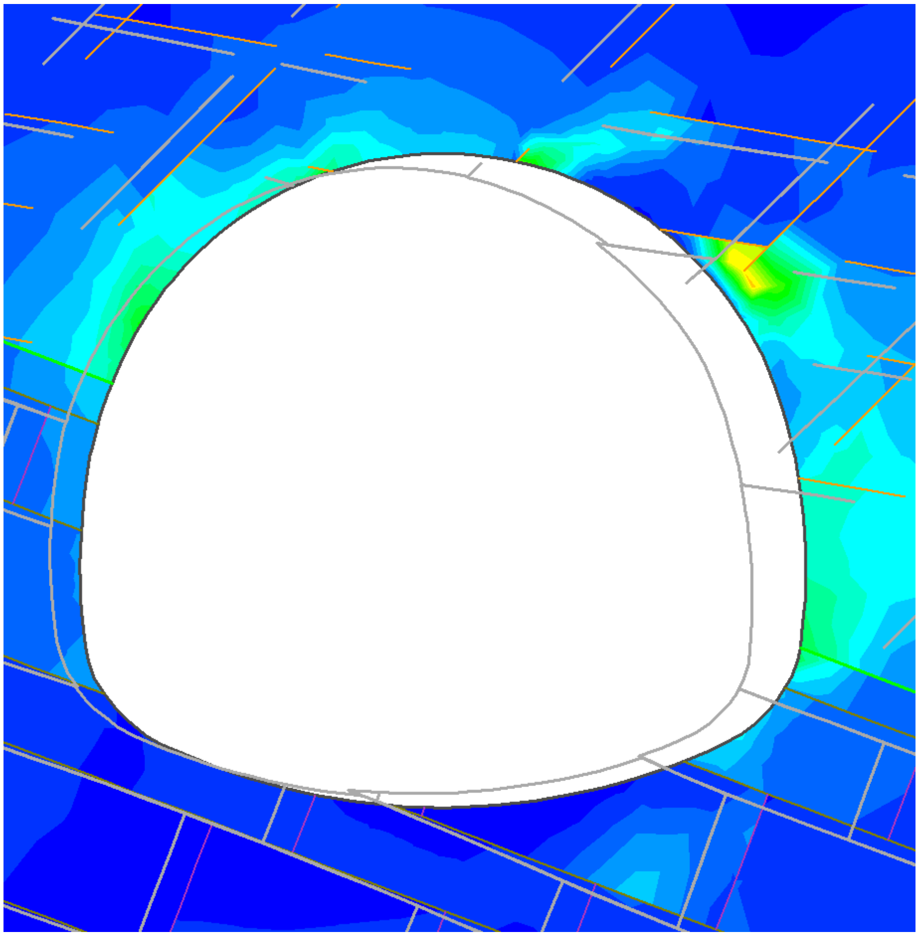


Figure 29-11: Plot of (exaggerated) deformed boundaries. Notice the differential movements of some joint ends which intersect the tunnel.

Gray lines that show how boundaries deform are drawn on the screen. From the deformed boundaries (shown on Figure 29-11), slip at joint ends that intersect the tunnel is visible. If we had left the joint end conditions as closed, these differential displacements would not have occurred.

This concludes the tutorial; you may now exit the *RS2* Interpret and *RS2* Model programs.