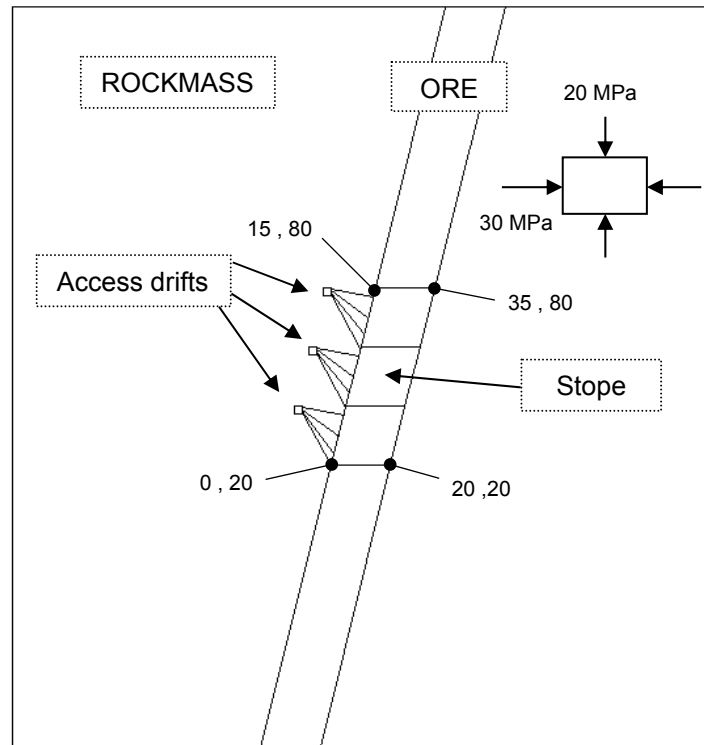


# Materials & Staging Tutorial



This tutorial will demonstrate the use of multiple materials and staging in *RS2*, using material and stage boundaries. The model represents a longhole stope in an orebody which has different properties than the surrounding rock mass.

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. Support installation is covered in more detail in the *RS2* Support tutorial.

The finished product of this tutorial can be found in the **Tutorial 02 Materials and Staging.fez** 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

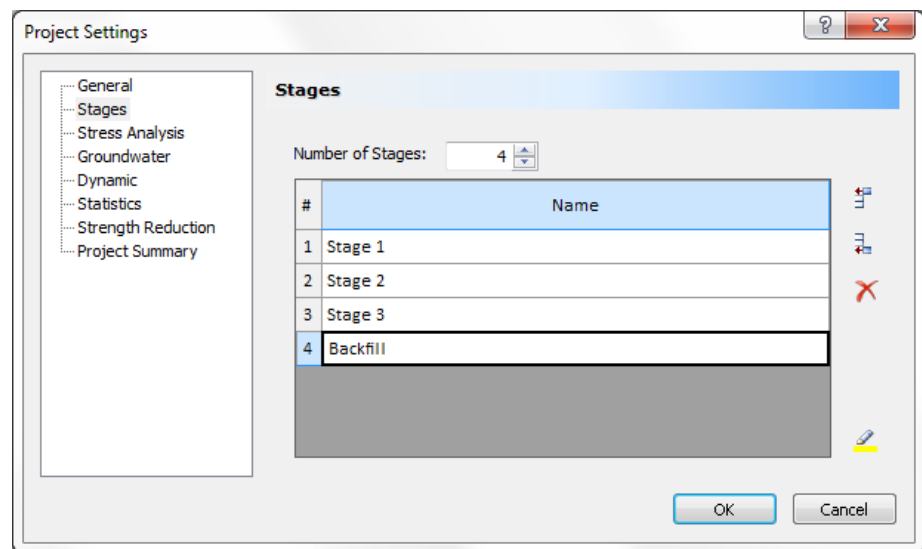
Whenever we are creating a staged model, the first thing we should do is define the number of stages in **Project Settings**, since this affects subsequent modeling options.



Select: Analysis → Project Settings

First select the **General** tab and make sure that the Units option is set to Metric, stress as MPa. This determines the units of length, force, stress and unit weight used in the analysis.

Select the **Stages** tab. Set the **Number of Stages** = 4, to create a total of four stages. Change the name of the fourth stage to backfill, as shown below.



Select the **Project Summary** tab and enter “Materials and Staging Tutorial” as the Project Title.

Do not change any other settings in the dialog. Select OK.

## Entering Boundaries

Geometry

Next we will define the slope and the three access drifts using Excavation boundaries. Select the **Geometry** workflow tab, then select **Add Excavation** from the toolbar or the **Boundaries** menu.

Select: Boundaries → Add Excavation



```
Enter vertex [t=table,i=circle,esc=cancel]: 35 80
Enter vertex [...]: 15 80
Enter vertex [...]: 10 60
Enter vertex [...]: 5 40
Enter vertex [...]: 0 20
```

```
Enter vertex [...]: 20 20
Enter vertex [...]: 25 40
Enter vertex [...]: 30 60
Enter vertex [...,c=close,esc=cancel]: c
```

Press F2 to Zoom All, to center the excavation in the view.



Select: Boundaries → Add Excavation

```
Enter vertex [t=table,i=circle,esc=cancel]: 0 80
Enter vertex [...]: -2.5 80
Enter vertex [...]: -2.5 77.5
Enter vertex [...]: 0 77.5
Enter vertex [...,c=close,esc=cancel]: c
```



Select: Boundaries → Add Excavation

```
Enter vertex [t=table,i=circle,esc=cancel]: -5 60
Enter vertex [...]: -7.5 60
Enter vertex [...]: -7.5 57.5
Enter vertex [...]: -5 57.5
Enter vertex [...,c=close,esc=cancel]: c
```



Select: Boundaries → Add Excavation

```
Enter vertex [t=table,i=circle,esc=cancel]: -10 40
Enter vertex [...]: -12.5 40
Enter vertex [...]: -12.5 37.5
Enter vertex [...]: -10 37.5
Enter vertex [...,c=close,esc=cancel]: c
```

Now let's add the two stage boundaries so that the slope 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 make sure the Snap option is enabled, so that we can snap the stage boundary vertices to the existing excavation vertices.

```
Enter vertex [t=table,i=circle,esc=cancel]: use the mouse to  
click on the excavation vertex at 10 60  
Enter vertex [...]: use the mouse to click on the excavation  
vertex at 30 60  
Enter vertex [...,enter=done,esc=cancel]: right-click and  
select Done
```

Notice that when you are in Snap mode, if you hover the cursor over a vertex, the 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 [t=table,i=circle,esc=cancel]: use the mouse to
click on the excavation vertex at 5 40
Enter vertex [...]: use the mouse to click on the excavation
vertex at 25 40
Enter vertex [...,enter=done,esc=cancel]: right-click and
select Done
```

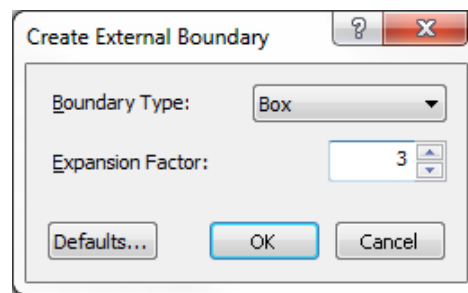
Since we planned ahead and added extra vertices to the stope where the stage boundaries would be, all we had to do was snap to these vertices to add the stage boundaries.

If the stage boundary vertices were not there, we could have still added the stage boundaries using the automatic boundary intersection capability of *RS2*, 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



We will use the default settings of Boundary Type and Expansion Factor so just 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 Snap mode.

```
Enter vertex [t=table,i=circle,esc=cancel]: use the mouse to
click on the excavation vertex at 15 80
Enter vertex [...]: enter the point 55 240 in the prompt line
Enter vertex [...,enter=done,esc=cancel]: press Enter
```



Select: Boundaries → Add Material

```
Enter vertex [t=table,i=circle,esc=cancel]: use the mouse to
click on the excavation vertex at 35 80
Enter vertex [...]: enter the point 75 240 in the prompt line
Enter vertex [...,enter=done,esc=cancel]: press Enter
```



Select: Boundaries → Add Material

```
Enter vertex [t=table,i=circle,esc=cancel]: use the mouse to
click on the excavation vertex at 0 20
Enter vertex [...]: enter the point -40 -140 in the prompt line
Enter vertex [...,enter=done,esc=cancel]: press Enter
```



Select: Boundaries → Add Material

```
Enter vertex [t=table,i=circle,esc=cancel]: use the mouse to
click on the excavation vertex at 20 20
Enter vertex [...]: enter the point -20 -140 in the prompt line
Enter vertex [...,enter=done,esc=cancel]: 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. RS2 automatically intersected these lines with the external boundary, and added new vertices. This capability of RS2 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 RS2 calculated the exact intersection.

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

## Meshing



For this model, we will use the default Mesh Setup parameters. Since we do not need to customize the discretization of the boundaries, we will use the **Discretize and Mesh** shortcut, which automatically discretizes the boundaries and generates the mesh with a single mouse click.

Select the **Mesh** workflow tab, then select **Discretize and Mesh** from the toolbar or the **Mesh** menu.



Select: Mesh → Discretize & Mesh

The mesh will be generated and the status bar will show the number of discretizations, elements and nodes in the mesh.

NODES = 7115 ELEMENTS = 3498
------------------------------

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 *RS2* 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 (i.e. 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 *RS2* Support Tutorial.

Select: File → Import → Import DXF

Navigate to the *RS2* 9.0 Examples > Tutorials folder and open the **Tutorial 02 Bolts.dxf** file. NOTE: the file path depends on your operating system, for example:

- Windows 7/8: C:\Users\Public\Public Documents\Rocscience\RS2 9.0 Examples\Tutorials
- Windows XP: C:\Documents and Settings\All Users\Shared Documents\Rocscience\RS2 9.0 Examples\Tutorials

The dialog should show a preview of the bolts in the DXF file. Select OK. 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 Bolt** option, but that is left as an optional exercise 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.

## Loads & Restraints

### Field Stress

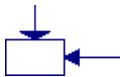
For this tutorial we will use a constant field stress. A constant field stress is a reasonable assumption when excavations are relatively deep (i.e. not near the ground surface). For surface or near surface excavations, gravity field stress is more appropriate, this is covered in later tutorials.

Select the **Loads & Restraints** workflow tab, then select **Field Stress** from the toolbar or the **Loading** menu.



Select: Loading → Field Stress

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 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.

## Materials & Staging

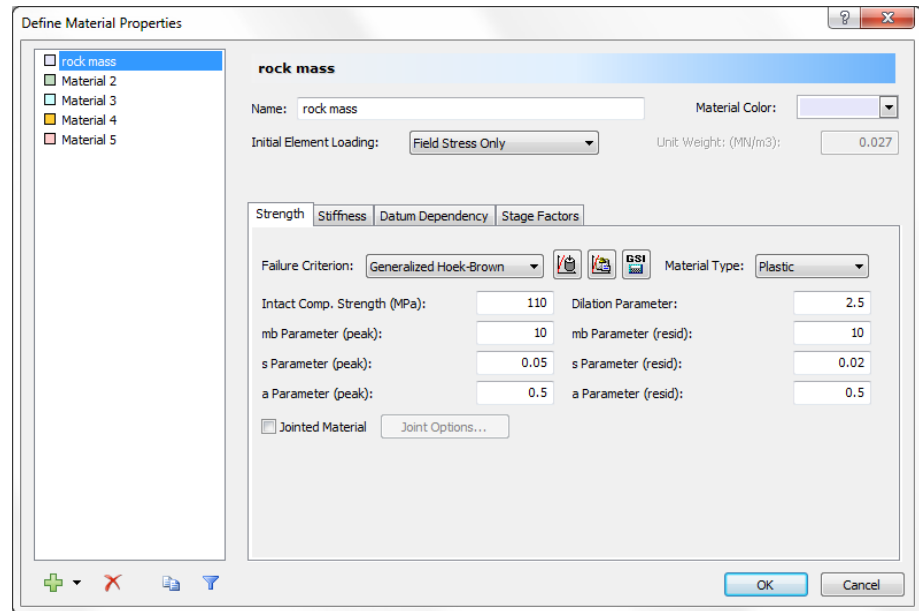
Select the **Materials & Staging** workflow tab.

### Define Material Properties



Select: Properties → Define Materials

With the first material selected in the list at the left of the Define Material Properties dialog, enter the rock mass properties.

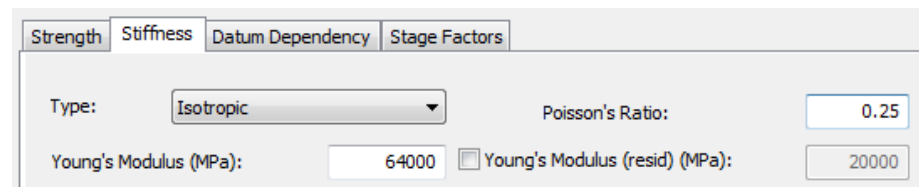


Enter Name = rock mass. Under the **Strength** tab enter the following parameters:

- Failure Criterion = Generalized Hoek-Brown
- Material Type = Plastic
- Comp. Strength = 110
- mb (peak) = 10
- s (peak) = 0.05
- a (peak) = 0.5
- Dilation = 2.5
- mb (residual) = 10
- s (residual) = 0.02
- a (residual) = 0.5

Under the **Stiffness** tab enter:

- Young's Modulus = 64000
- Poisson's Ratio = 0.25



Select the **second** material in the list, and enter the **ore** properties.

Enter Name = ore. Under the **Strength** tab enter the following parameters:



**Define Material Properties**

Material List: ☐ rock mass, ☒ ore, ☐ Material 3, ☐ Material 4, ☐ Material 5

Name: ore Material Color:

Initial Element Loading: Field Stress Only Unit Weight: (MN/m<sup>3</sup>): 0.027

Strength Stiffness Datum Dependency Stage Factors

Failure Criterion: Generalized Hoek-Brown Material Type: Plastic

Intact Comp. Strength (MPa): 54 Dilation Parameter: 0

mb Parameter (peak): 2 mb Parameter (resid): 2

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

a Parameter (peak): 0.5 a Parameter (resid): 0.5

☐ Jointed Material Joint Options...

Under the **Stiffness** tab enter:

Strength Stiffness Datum Dependency Stage Factors

Type: Isotropic Poisson's Ratio: 0.25

Young's Modulus (MPa): 35000 ☐ Young's Modulus (resid) (MPa): 20000

Select the **third** material in the list and enter the **backfill** properties. Enter Name = backfill, Initial Element Loading = Body Force Only, Unit Weight = 0.023. Under the **Strength** tab enter the following parameters:

**Define Material Properties**

Material List: ☐ rock mass, ☐ ore, ☒ backfill, ☐ Material 4, ☐ Material 5

Name: backfill Material Color:

Initial Element Loading: Body Force Only Unit Weight: (MN/m<sup>3</sup>): 0.023

Strength Stiffness Datum Dependency Stage Factors

Failure Criterion: Generalized Hoek-Brown Material Type: Plastic

Intact Comp. Strength (MPa): 7.5 Dilation Parameter: 1.5

mb Parameter (peak): 6 mb Parameter (resid): 6

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

a Parameter (peak): 0.5 a Parameter (resid): 0.5

☐ Jointed Material Joint Options...

Under the **Stiffness** tab enter:

Strength Stiffness Datum Dependency Stage Factors

Type: Isotropic Poisson's Ratio: 0.025

Young's Modulus (MPa): 2000 ☐ Young's Modulus (resid) (MPa): 20000

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. The unit weight of a material must be defined when the initial element loading includes body force.

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

## Define Bolt Properties



Select: Properties → Define Bolts

Select the first bolt type from the list at the left of the dialog. Enter Name = cables. Select **Bolt Type** = Plain Strand Cable. Enter Out-of-Plane Spacing = 2. Leave all other parameters at the default values. Select OK.

If you zoom in to the access drifts, you will notice that face plates are now displayed at the upper end of each cable. For more information about the Plain Strand Cable model see the *RS2* Help system and references.

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

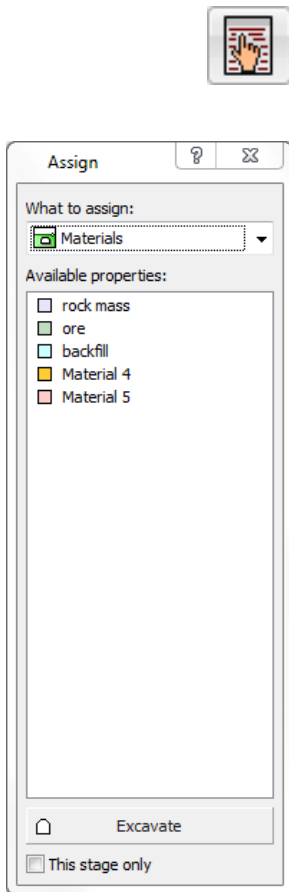
Select: Properties → Assign Properties

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 top of the Assign dialog.
3. Select the “ore” material in the Assign dialog. (Notice that the material names are the names you entered when you defined the three materials – i.e. rock mass, 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’.



*Stage 1 – material assignment and excavation of bottom section of stope and access drifts*

**NOTE:** since we defined the “rock mass” properties using the *first* material in the Define Materials dialog, the “rock mass” properties do not need to be assigned by the user. The properties of the first material in the Define Materials dialog are automatically assigned to all elements of the model. Therefore the rock mass, 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. You can also rotate the mouse wheel to zoom in or out.

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

*Stage 2 – excavation of mid-section of stope.*

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.

*Stage 3 – excavation of top-section of stope.*

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.

*Stage 4 – backfill of entire stope.*

13. Select the Stage 4 tab.
14. Select the “backfill” material 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 type 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 top 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 displayed in a lighter shade of colour.

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

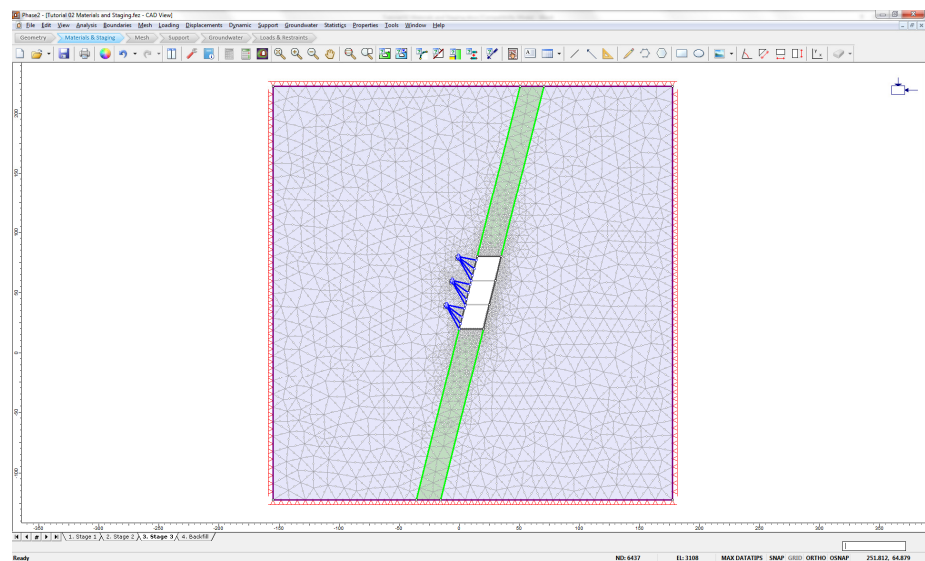
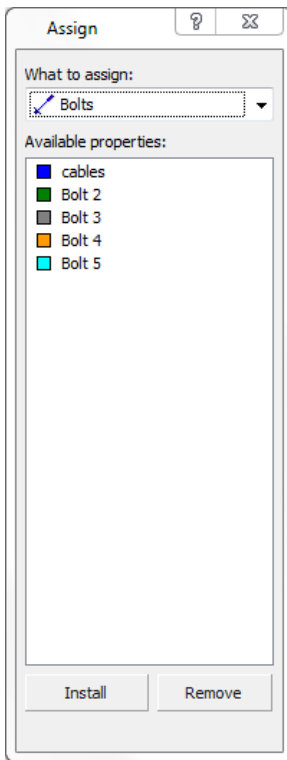
8. Select the Stage 2 tab. Both the lower and middle sets of bolts should be installed.

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.



*Finished model – RS2 Material & Staging Tutorial*

## Compute

---



Before you analyze your model, save it as a file called **matstg.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. 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: Analysis → Interpret

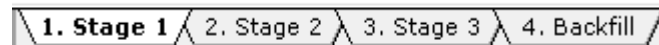
This will start the *RS2* Interpret program.

### Viewing Stages

---

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

Viewing results at different stages in *RS2* 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.

**TIP:** you can also use the Page Up / Page Down keys to change the viewing stage.



Toggle on the principal stress trajectories, using the button provided in the 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.

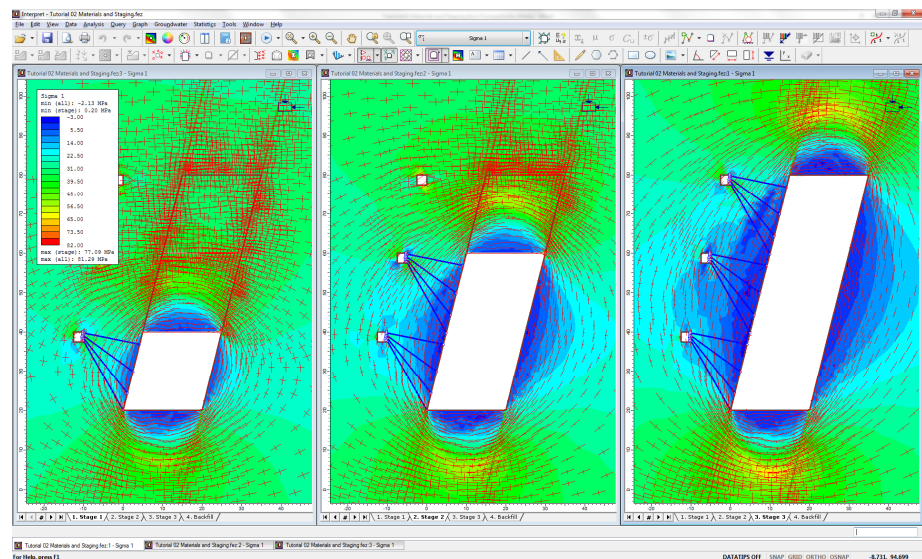
5. Display the stress trajectories in each view.

6. Hide the legend in the right and middle views (use View → Legend Options, or right-click on a Legend and select Hide Legend).



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.



*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.

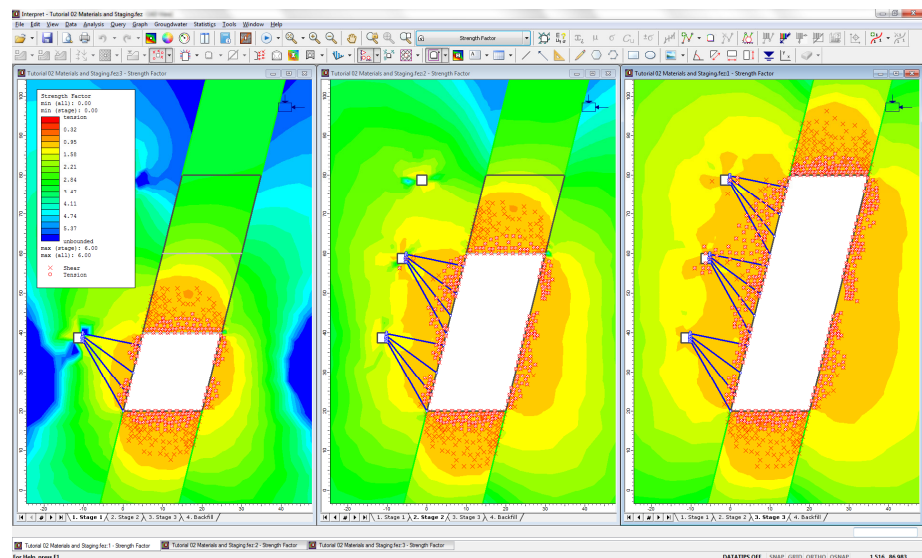


1. Display the Strength Factor in each view.
2. Toggle Stress Trajectories OFF, and Yielded Elements ON, using the Display toolbar buttons, in each view.



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 (i.e. in the orebody), although there is yielding in the rockmass as well.



### *Strength factor contours and yielded elements*

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 Options), 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

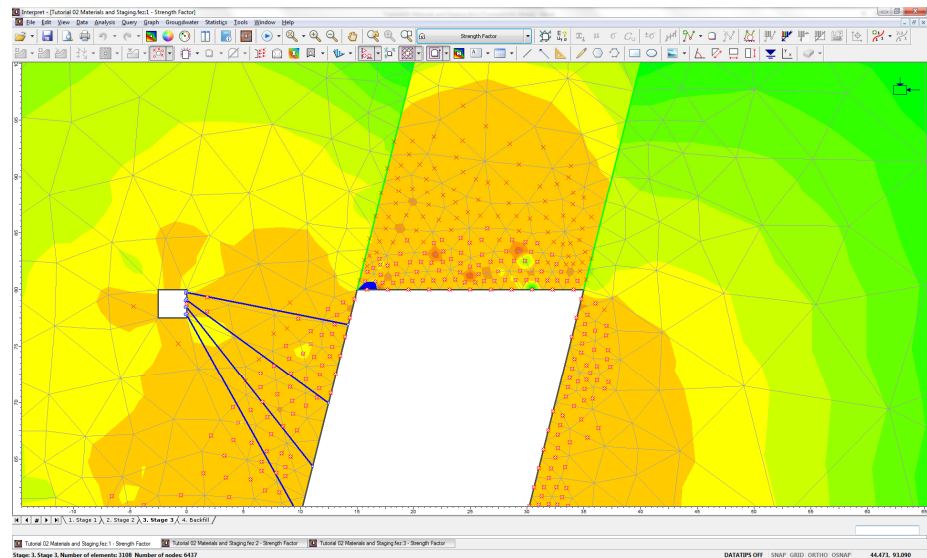


```
Enter first window point[esc=quit]: 0 100
Enter second window point[esc=quit]: 50 60
```

You will see that there are actually two symbols used for the yielded element markers – failure in shear is indicated by an × marker, and failure in tension is indicated by a ○ marker. This is indicated in the Legend. If tensile failure is accompanied by shear failure, the symbols overlap.



Display the mesh by selecting the Elements button in the toolbar. Note that for 6-noded triangles, the Yielded Element symbols occur at the mid-side element nodes.



### *Strength Factor contours and yielded elements, Stage 3.*

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. You can also pan by holding down the mouse wheel and moving the mouse.

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

---

Now look at the total displacements.

Select:  Total Displacement ▼

Select the Stage 1 tab. The maximum total displacement for Stage 1 is about 22 mm, as indicated in the status bar.

Maximum Total Displacement = 0.0219 m

Select the Stage 2 tab.

Maximum Total Displacement = 0.0286 m

Select the Stage 3 tab.

Maximum Total Displacement = 0.0347 m

Select the Stage 4 tab.

Maximum Total Displacement = 0.0361 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.

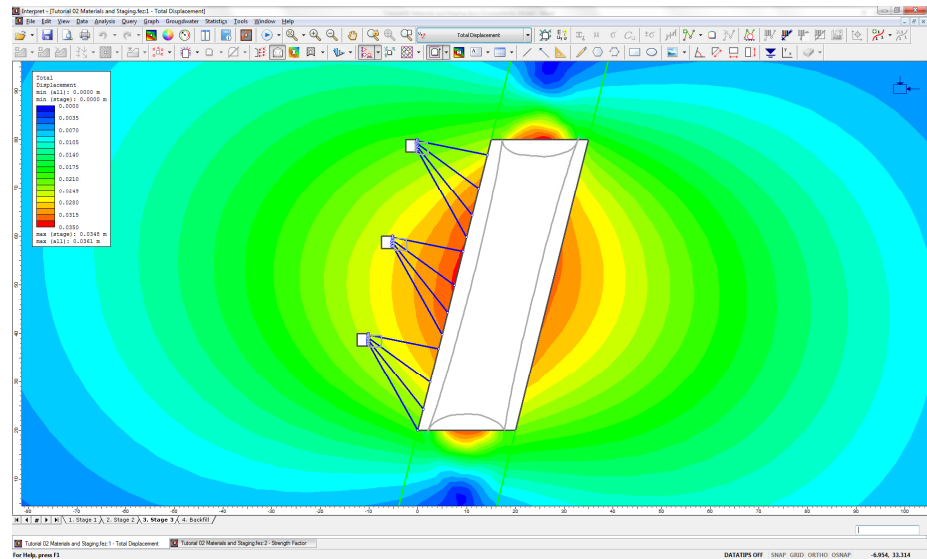
In the Display Options dialog, select the Stress tab, toggle on Deform Boundaries, enter a Scale Factor of 100 and select Done.

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 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.



*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 toolbar. Then close two of the views. Then maximize the remaining view, and select Zoom Excavation, if necessary. Turn off the Deformed Boundaries display by selecting the Deformed Boundaries toolbar button.

## Stage 4

Remember that in the fourth stage of this model we backfilled the entire slope 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 *RS2* Help system for more information about Initial Element Loading.

## Bolts

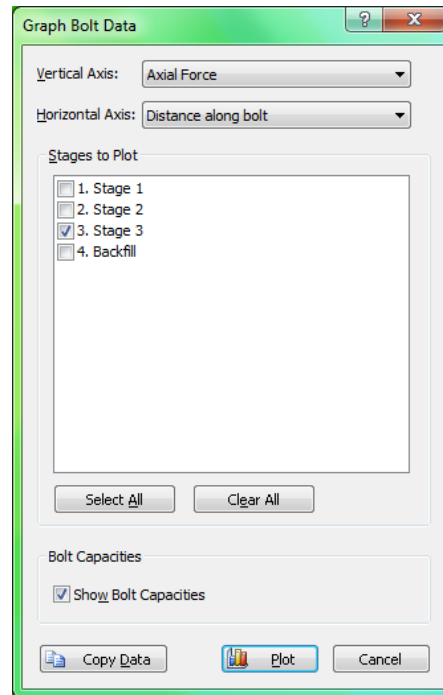
Now let's see what's going on with our bolts. They don't seem to have any obvious effect on the stress or strength contours, so let's see what other information we can gather, using the Graph Bolt Data option. First, select the Stage 3 tab.



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 dotted line when selected), right-click the mouse and select Graph Selected, and you will see the following dialog:



In the Graph Bolt Data dialog, select Create Plot, and a graph of Axial Force for the selected bolts will be generated.

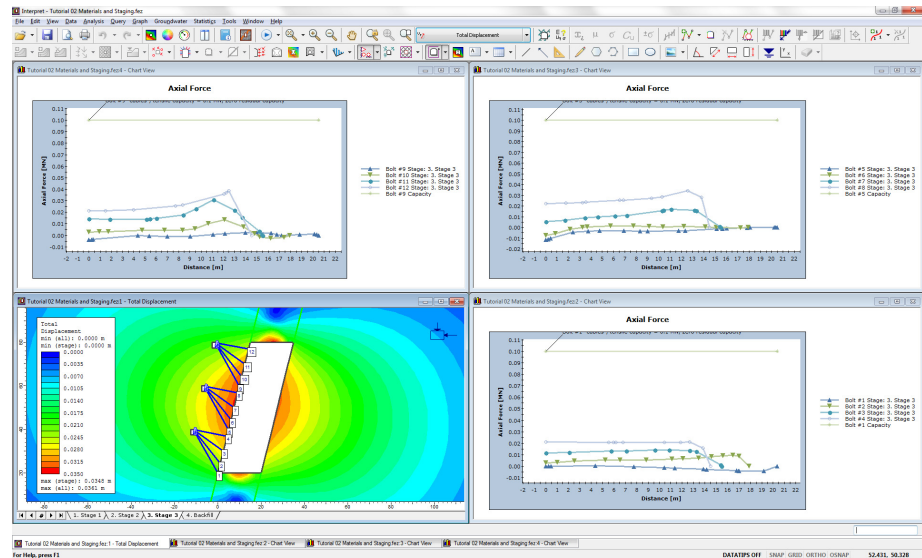
Now repeat the above procedure for the middle and the top sets of four bolts, to generate two more graphs.

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. Right click on the model and select Display Options. Under Support, turn on Bolt Numbers and close the dialog. You will notice numbers have been assigned to each bolt. The bolt numbers on the model correspond to the bolt numbers on the graphs, allowing you to identify the bolts.



*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 the Add Bolt option or DXF import.*

Also notice on the plots that the peak capacity of the bolts (0.1 MN) is indicated by a horizontal line. The force in all 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 toolbar. In the status bar at the bottom left of the screen you will see:

No yielded bolt elements

As we expected, no bolts have yielded (if there were yielded bolts, the yielded sections would be highlighted with a different colour).

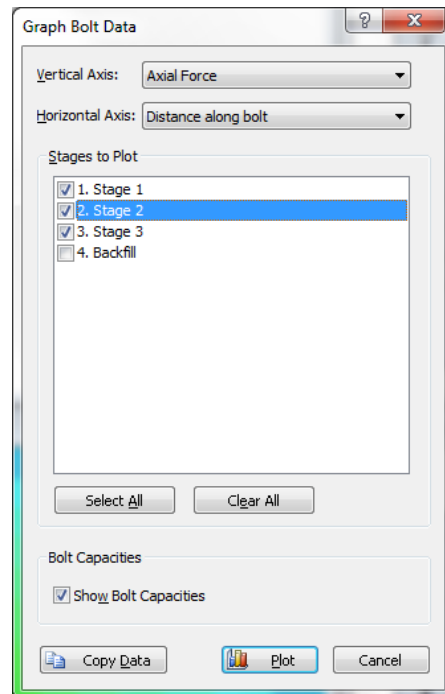
While we are looking at bolt data, let's illustrate one more feature, the ability to plot data from multiple stages on a single graph. First, maximize the model view (if you are still looking at the tiled view of all the graphs).



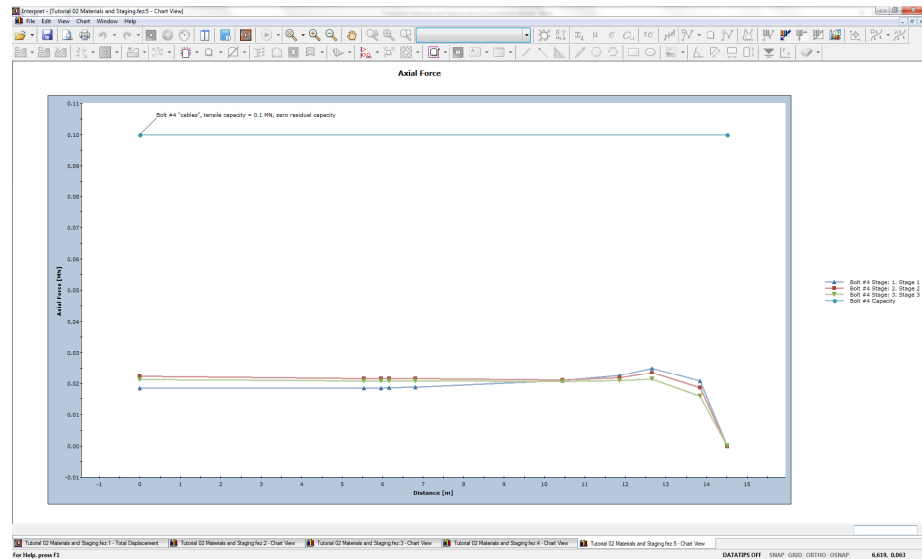
Select: Graph → Graph Bolt Data

Pick bolts to graph[enter=done,\*=all, esc=quit]: ***select bolt #4 (the fourth bolt from the bottom of the model).***

Right-click the mouse and select Graph Selected as before, except this time select the first three stages to plot, using the checkboxes.



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.



### *Axial force in Bolt 4 at Stages 1, 2 and 3.*

In this case, the axial force in the bolt, near the face plate, increases from stage 1 to stage 2, and does not change significantly from stage 2 to stage 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 *RS2* Support Tutorial) this will often be the case.

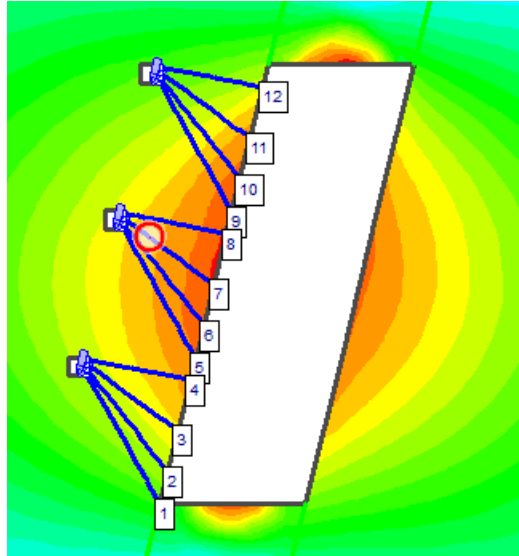
Examining the load in the bolts allows you to design bolt support by varying bolt parameters (diameter, etc) to obtain optimal stress in the bolt system.

Let's demonstrate one more interactive feature of the bolt graphs. Tile the views. Notice that as you hover the mouse cursor over points on the bolt graphs:

- a popup box displays the bolt data corresponding to that point
- on the model view, you will see a red circle displayed on the exact bolt element corresponding to the point on the bolt graph, as shown in the following figure.

Place the mouse cursor over different points on the bolt graph and observe the location of the red circle on the model view.





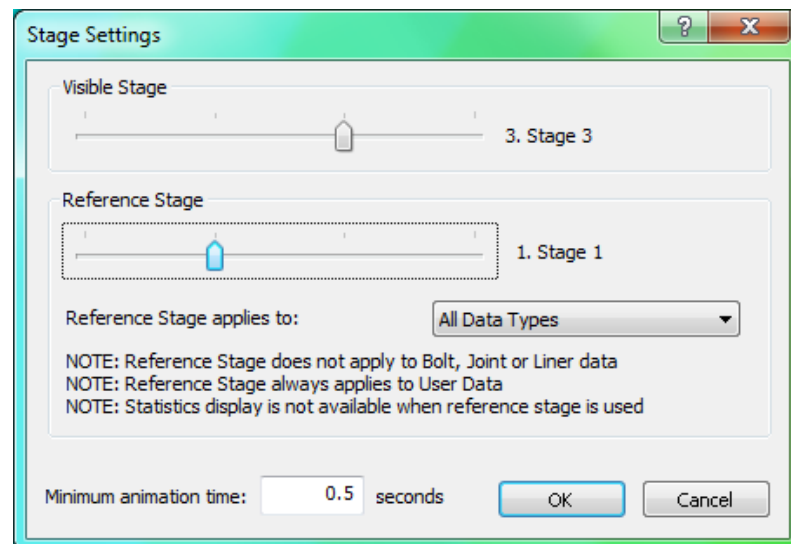
*Interactive red circle corresponding to selected data point on bolt graph.*

Close the bolt graph views and maximize the model view.

## 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. The **( r )** in the stage tabs indicates that you are viewing differential results with respect to a reference stage.



We will not explore differential results further in this tutorial, but the user is encouraged to explore this on their own. See the *RS2* Help system for information about how to interpret differential results.

## Log File and Load Step Plot

---

Before we conclude this tutorial, let's examine the Log File which is created during a *RS2* stress analysis, and the Load Step plot.

Select: Analysis → Log File

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

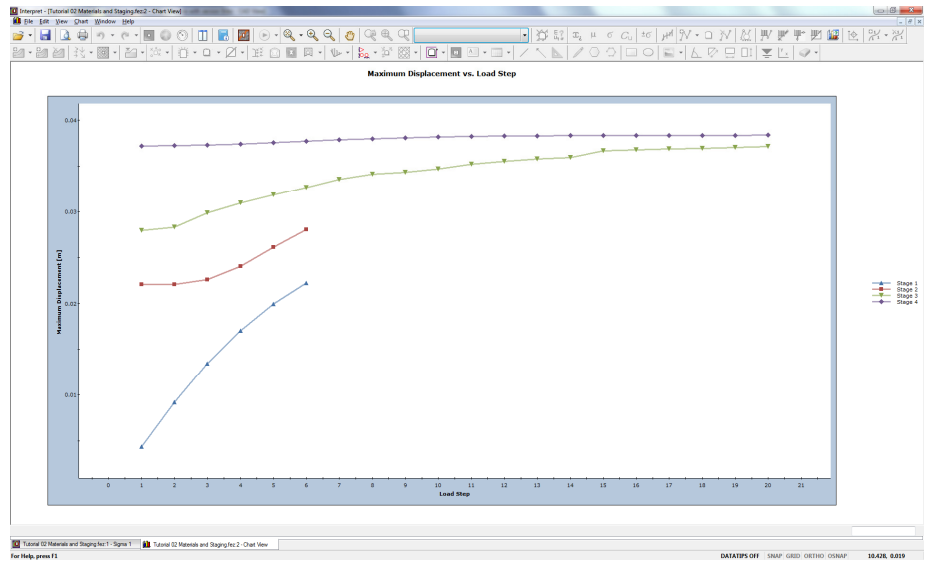
After a plastic analysis, it is a good idea to check the log file to make sure that the solution converged within the specified tolerance. For this example, the final calculated tolerance is less than 0.001 for each load step, indicating convergence within our specified tolerance. The tolerance, number of load steps, and maximum number of iterations, can all be user specified in the Project Settings dialog when you create the model.

Close the log file view.

Now view the Load Step Plot, which graphically illustrates the data in the Log File.



Select: Analysis → Load Step Plot



### *Load Step plot for stress analysis*

The Load Step Plot plots the maximum displacement at each load step, for each stage.

That concludes the 'Materials and Staging' tutorial.