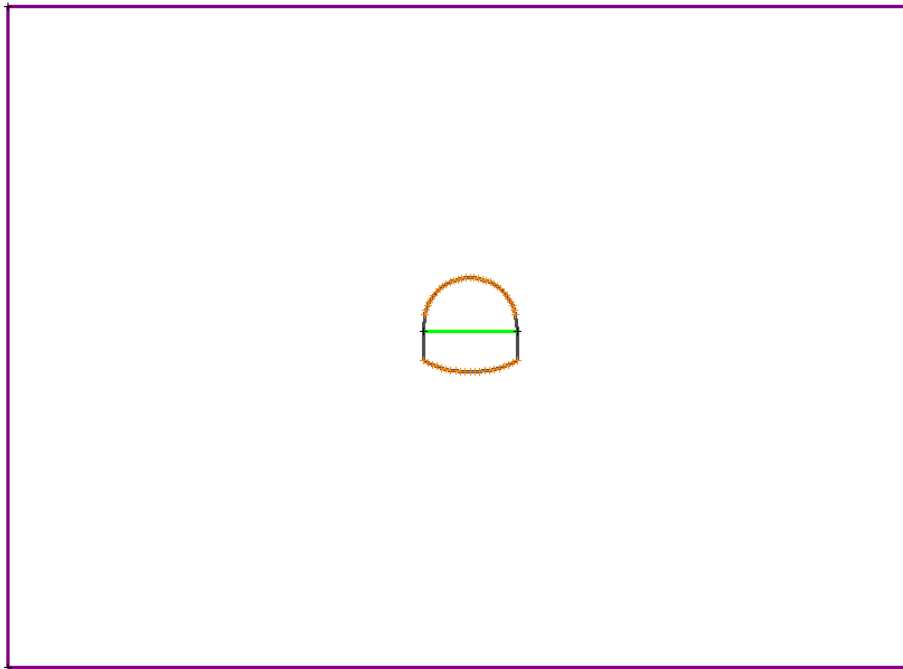


Material Softening Tutorial

The general procedure used in *Phase2* for sequential advance of tunnel excavations is to use a combination of material softening and stress relief. Material softening usually done by reducing the modulus of elasticity, and Poisson's ratio of the materials. Load split is used for stress relief.




Phase2 version 5.0 introduced Stage Materials option in the Material Properties Dialog. The fraction of material softening ends up representing the amount of deformation you expect before installation of support. This can be estimated using simple ground reaction curves, using *RocSupport* or by field measurements and experience. Load split can be customized for every material in the model using load split dialog. Stress relief is required since just softening the material will do nothing. See the *Phase2* FAQ for an [explanation of this phenomenon](#).






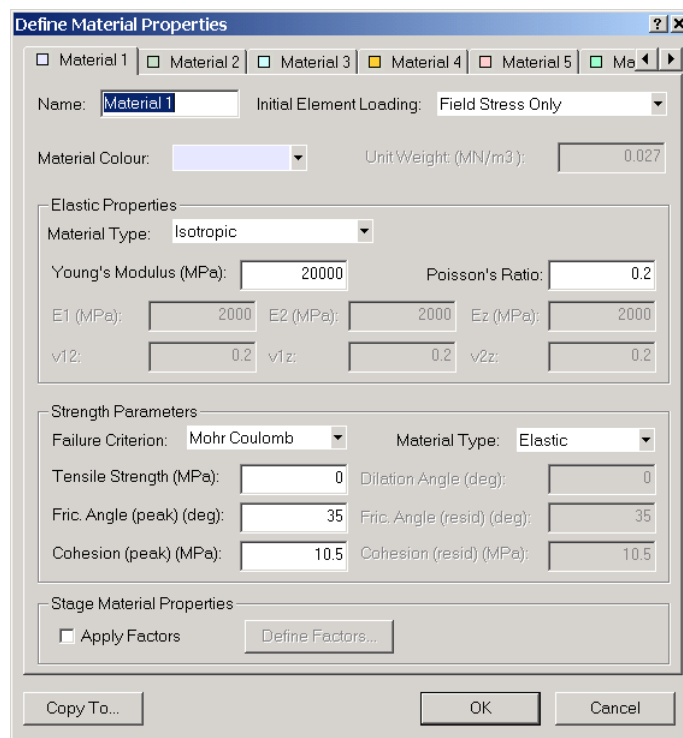
This tutorial involves a tunnel opening to be excavated in two stages. However, for the analysis purpose the tunnel modeled with five stages.

If you wish to skip the model process, the finished product of this tutorial can be found in the (material softening.fea) data file which is available form the download section

Model

1. Start the *Phase²* program.
2. Click the *File* menu and select *Import* from the list of options.
3. From the ensuing sub-menu select *Import DXF*.
4. Click the *Import* button in the pop-up window.
5. Navigate to the directory with your tutorial files and select the (material softening.dxf) file. Click the *Open* button.
6. To save the opened file as a *Phase²* project, click the *File* menu and select the *Save* option. Save your file as “tutorial 5”. (The extension .fea is automatically appended to your file name.)
7. Next click the *File* menu and select the *Project Settings* option. Alternatively you can click the Project Settings button  on the toolbar.
8. In the Project Settings window, change the number of stages to be analyzed from 1 to 5. Select *OK*.
9. Click the *Boundaries* menu and select the *Add Material* option. Alternatively you can click the Add Material button  on the toolbar.
10. Click the right mouse button anywhere in the drawing region of the program. From the ensuing pop-up window select the *Vertex Snap* option.
11. Move the mouse pointer to any point close to the excavation boundary vertex with coordinates (9.3, 3.7) and click the left mouse button. **Tip:** Coordinates of the location of the mouse pointer are given at the bottom right corner of the program window.
12. Next move the cursor over to a point with coordinates close to (20.3, 3.7) and click the left mouse button. Press the *Enter* key on your keyboard to end the process of adding a material boundary. Your screen should look like the figure below:
13. Please save your file again.
14. Click the *Boundaries* menu and select the *Add External* option. Alternatively you can click the Add External button  on the toolbar.
15. In the resulting pop-up window, replace the value of 3 in the Expansion Factor field with 4. Select *OK*.

16. Next select the *Mesh* menu from the program menu bar. Choose the *Discretize* option from the resulting list. Alternatively click the Discretize button  on the toolbar.
17. Click the *Mesh* menu again and select the *Mesh* option from the ensuing list. Alternatively click the Mesh button  on the toolbar.
18. Next go back to the *Mesh* menu and select the *Mesh Refinement* option. From the ensuing sub-menu select the Define Mesh Quality sub-option. Set the Minimum Interior Angle at 10° and the maximum angle at 130° in the resulting pop-up window. Click *OK*.
19. Revisit the *Mesh* menu and select the *Mesh Quality* option. All elements in your mesh should pass the test that checks elements to see whether they satisfy the constraints you specified in the previous step.
20. Select the *Properties* menu from the program menu bar and select the *Define Materials* option. Alternatively you can click the Material Properties button  on the toolbar. In the resulting window, enter the property values as shown below:



Define Material Properties

Material 1 Material 2 Material 3 Material 4 Material 5 Material 6

Name: Initial Element Loading:

Material Colour: Unit Weight (MN/m³):

Elastic Properties

Material Type:

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

E1 (MPa): E2 (MPa): E3 (MPa):

v12: v13: v23:

Strength Parameters

Failure Criterion: Material Type:

Tensile Strength (MPa): Dilatation Angle (deg):

Fric. Angle (peak) (deg): Fric. Angle (resid) (deg):

Cohesion (peak) (MPa): Cohesion (resid) (MPa):

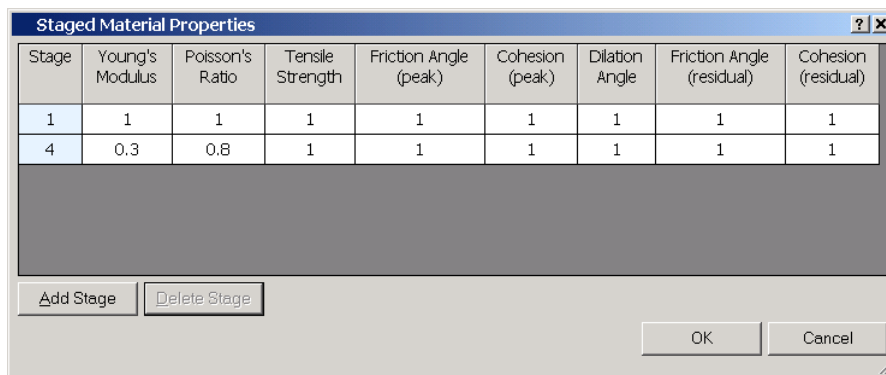
Stage Material Properties

Apply Factors

21. Click the Material 2 tab and rename the material as Zone 1. Input the same deformation and strength properties as previously entered for the host rock material You may change the colour of the zone material to red, for example. This is just for visualization purposes – to make it easier to identify the different assigned materials in the model.
22. Select the *Apply Factors* checkbox in the lower section of dialog. Click the *Define Factors* button. The Staged Material Properties dialog opens up.
23. In the dialog enter the stage number, and Young’s modulus and Poisson’s ratio reduction factors shown on the diagram below:



24. Click *OK* to close the Staged Material Properties dialog.
25. Repeat steps 21 for Material 3, changing the name to Zone 2 and the colour to yellow.
26. Select the *Apply Factors* checkbox in the lower section of dialog and click the *Define Factors* button.
27. In the ensuing Staged Material Properties dialog enter the stage number, and the Young’s modulus and Poisson’s ratio reduction factors shown on the diagram below:

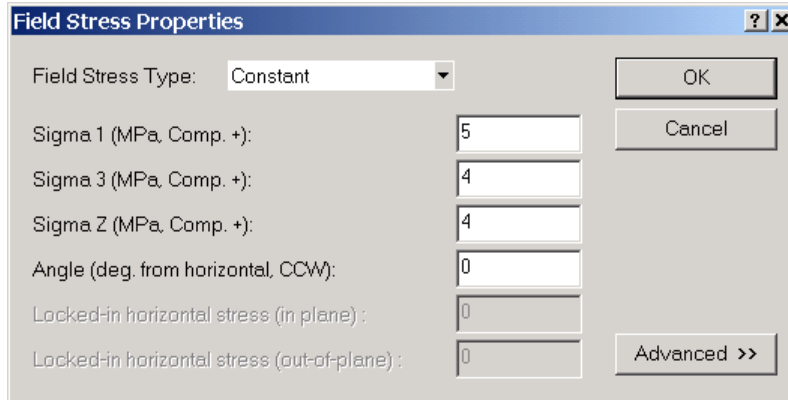


28. Click *OK* to close the Staged Material Properties dialog.

29. Click *OK* to close the Define Material Properties dialog.

30. To set the field stresses for the problem, select the *Loading* menu and click the *Field Stress* option.

31. Enter the values shown on the figure below:

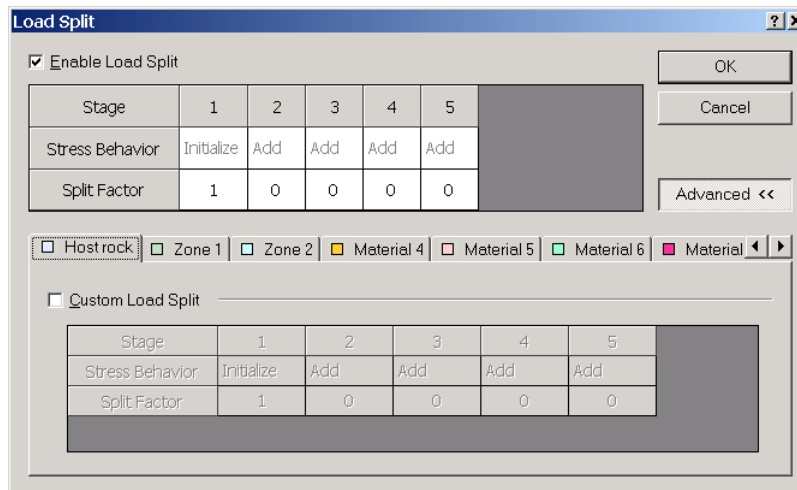


The dialog box titled "Field Stress Properties" contains the following fields and controls:

- Field Stress Type: Constant (dropdown menu)
- Sigma 1 (MPa, Comp. +): 5 (text input)
- Sigma 3 (MPa, Comp. +): 4 (text input)
- Sigma Z (MPa, Comp. +): 4 (text input)
- Angle (deg. from horizontal, CCW): 0 (text input)
- Locked-in horizontal stress (in plane): 0 (text input)
- Locked-in horizontal stress (out-of-plane): 0 (text input)
- Buttons: OK, Cancel, Advanced >>

32. Click the *Loading* menu and click the *Load Split* option. The Load Split dialog opens up.

33. In the dialog enter the values shown on the images below for the Host Rock, Zone 1 and Zone materials.



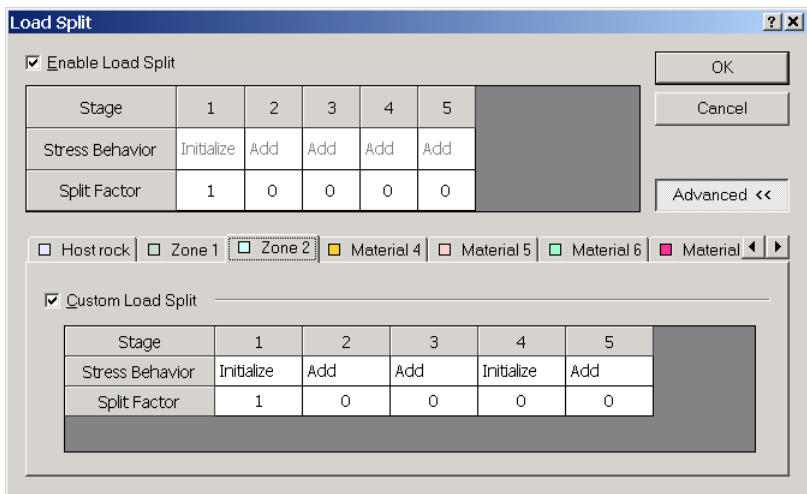
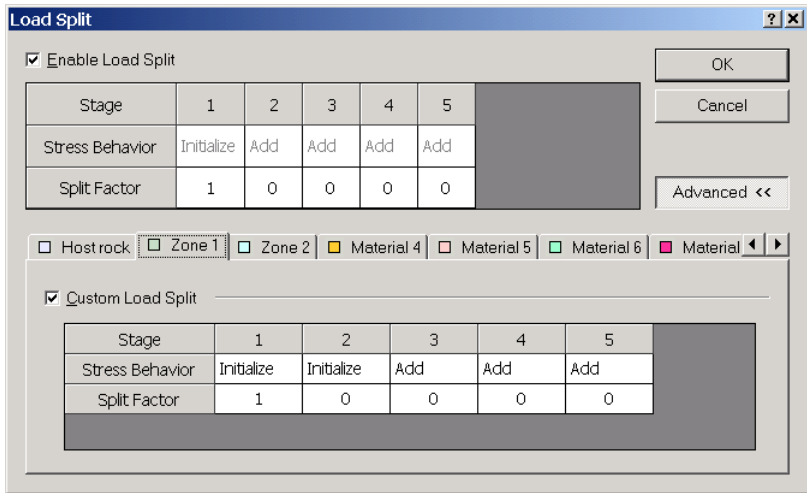
The dialog box titled "Load Split" contains the following elements:

- Enable Load Split
- Buttons: OK, Cancel, Advanced <<
- Material selection row: Host rock, Zone 1, Zone 2, Material 4, Material 5, Material 6, Material
- Custom Load Split
- Table 1 (Main):


Stage	1	2	3	4	5
Stress Behavior	Initialize	Add	Add	Add	Add
Split Factor	1	0	0	0	0

- Table 2 (Custom Load Split):

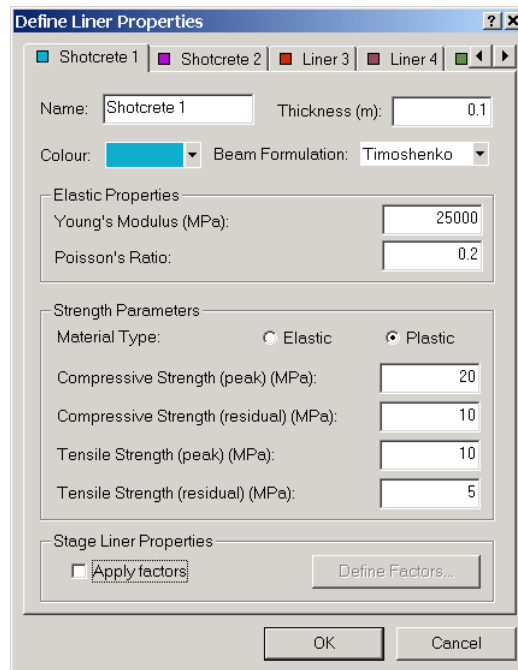
Stage	1	2	3	4	5
Stress Behavior	Initialize	Add	Add	Add	Add
Split Factor	1	0	0	0	0



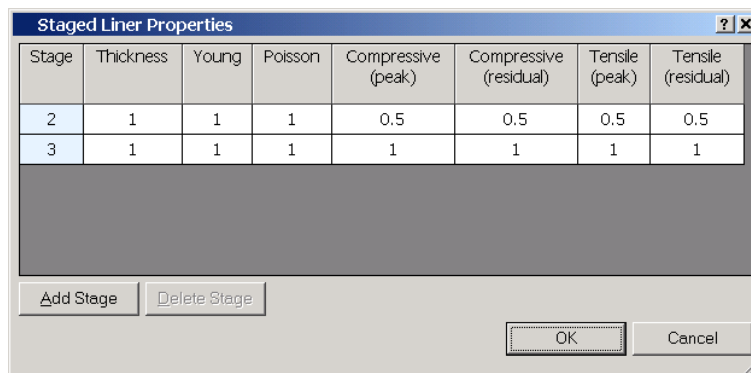
34. Click the *OK* button to close the dialog.

35. We are now going to set the properties for the shotcrete liners that will be used to support the excavation. We will also model the time-dependent strength gain of the applied shotcrete. Click the *Properties* menu and select the *Define Liners* option. Alternatively you can select the Define Liners button  on the toolbar.

36. For Shotcrete 1 enter the values shown on the figure below.



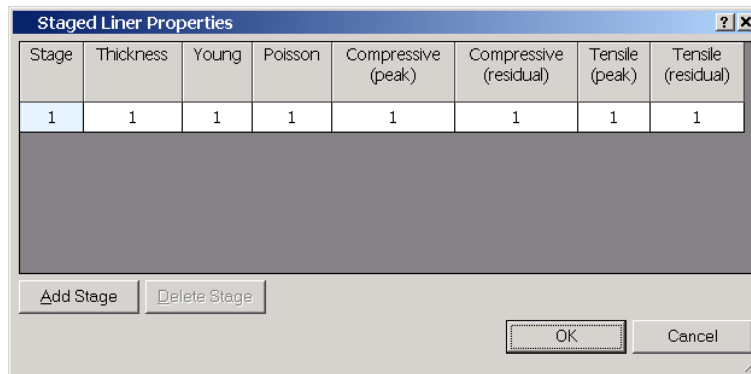
37. Next select the *Apply Factors* checkbox and click the *Define Factors* button. Enter the values shown on the figure below in the ensuing Staged Liner Properties dialog.





38. Click OK to close the dialog.

39. Click on the Shotcrete 2 tab and enter the same values as for Shotcrete 1.

40. Next select the *Apply Factors* checkbox and click the *Define Factors* button. Enter the values shown on the figure below in the ensuing Staged Liner Properties dialog.



41. Click the *OK* button to close the dialog.
42. Click *OK* in the to close the Define Liner Properties dialog.
43. If Stage 1 is not the active drawing region, click the Stage 1 tab (stage tabs are located below the drawing region of the program).
44. Select the *Properties* menu from the program menu bar and select the *Assign Properties* option. Alternatively you can click the Assign Properties button  on the toolbar.
45. Select the Host Rock material in the dialog that comes up, and click anywhere in the region outside the proposed excavation to assign that region as host rock.
46. Next set the upper part of the tunnel as Zone 1 material, and the lower section as Zone 2 material.
47. In Stage 2 of our model we shall apply shotcrete to the boundary of the upper tunnel section (other than the floor) and soften the material in Zone 1. Click the Stage 2 tab to make it active.
48. Select the *Support* menu and click the *Add Liner* option. Draw a window that encompasses the upper tunnel section. Alternatively click the Add Liner button  on the toolbar. (Make sure its bottom edge barely goes into the lower section.) Hit the *Enter* key on your keyboard to end the process of selecting elements to which shotcrete should be applied.
49. In Stage 3 we will excavate the upper tunnel section. Click the Stage 3 tab. Click the Assign Properties button. On the Assign Properties dialog click the Excavate option near the bottom of the window and click anywhere within the upper tunnel section.
50. In Stage 4 we will apply shotcrete to the boundary of the lower tunnel section and soften material in that region. Click the Stage 4 tab.

51. Select the *Support* menu and click the *Add Liner* option. Draw a window that encompasses the lower tunnel section. Hit the *Enter* key on your keyboard to end the process of selecting elements to which shotcrete should be applied.
52. Click the Stage 5 tab and excavate the lower tunnel section in similar fashion to what was done for the upper section (step 28).
53. Save your file again and run the analysis. To compute the model, click the *Compute* button on the toolbar.