

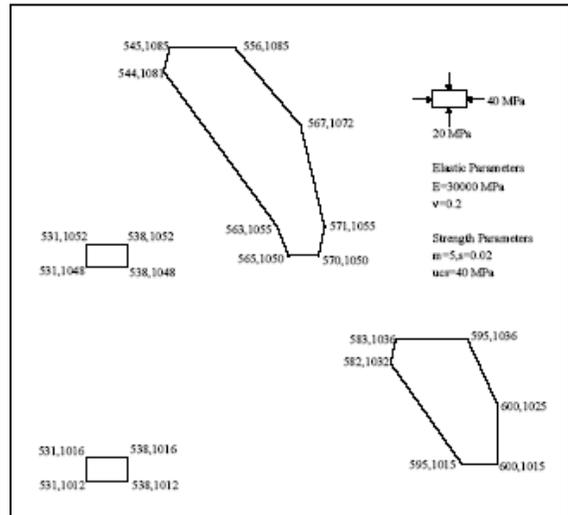
Lab Practical - Boundary Element Stress Analysis

Part A – The Basics

In this exercise, we will model a simple set of excavations using the Rocscience program EXAMINE^{2D}.

The first step is to define the excavation geometry and parameters required for the analysis. If you have not already done so, start the program.

The following steps are laid out in the standard order usually followed to perform a boundary-element analysis.



1) PROJECT SETTINGS

The first step is to define the project settings. The Project Settings option is used to configure the main analysis parameters for your model (e.g. Units, Field Stress Type, Strength Criterion, etc). You can select Project Settings from the toolbar or the Analysis menu.

Select: ANALYSIS → PROJECT SETTINGS

Under the General tab, select the following:

- *Units:* Metric, stress as MPa
- *Field Stress Type:* Constant
- *Elastic Properties:* Isotropic
- *Strength Criterion:* Generalized Hoek-Brown, with the “Use GSI, mi, D” checkbox selected

Select the Analysis tab in Project Settings. We will use all the default options, including:

- *Number of Boundary Elements:* 100 (Note: This process will create approximately a total of 100 boundary elements around the perimeter for *all* the excavations in our model.)

2) SET LIMITS

The next step in any computer-aided design process is setting the drawing limits of the region so that the limits encompass the excavation geometry.

Select: VIEW → LIMIT

Enter the minimum and maximum x-y coordinates as shown below. These limits will approximately centre the model in the drawing region.

Enter **MINIMUM X,Y COORDINATES: 500,1000**
Enter **MAXIMUM X,Y COORDINATES: 630,1100**

2) DEFINE THE EXCAVATION GEOMETRY

Now define the excavation geometry. EXAMINE^{2D} defines excavations by allowing the user to interactively draw the boundaries using the mouse, or by typing the coordinates in the prompt line at the bottom right corner of the screen. Enter the four excavation boundaries using the following sequence:

Excavation #1:

Select: *BOUNDARIES* → *ADD EXCAVATION*

```
Enter Excavation CCW, Starting vertex: 570,1050  
Vertex [@=relative, a=arc, c=close,u=undo,Esc=cancel]: 571,1055  
Vertex [@=relative, a=arc, c=close,u=undo,Esc=cancel]: 567,1072  
Vertex [@=relative, a=arc, c=close,u=undo,Esc=cancel]: 556,1085  
Vertex [@=relative, a=arc, c=close,u=undo,Esc=cancel]: 545,1085  
Vertex [@=relative, a=arc, c=close,u=undo,Esc=cancel]: 544,1081  
Vertex [@=relative, a=arc, c=close,u=undo,Esc=cancel]: 563,1055  
Vertex [@=relative, a=arc, c=close,u=undo,Esc=cancel]: 565,1050  
Vertex [@=relative, a=arc, c=close,u=undo,Esc=cancel]: c
```

And again, excavation #2:

Select: *BOUNDARIES* → *ADD EXCAVATION*

```
Enter Excavation CCW, Starting vertex: 600,1015  
Vertex [@=relative, a=arc, c=close,u=undo,Esc=cancel]: 600,1025  
Vertex [@=relative, a=arc, c=close,u=undo,Esc=cancel]: 595,1036  
Vertex [@=relative, a=arc, c=close,u=undo,Esc=cancel]: 583,1036  
Vertex [@=relative, a=arc, c=close,u=undo,Esc=cancel]: 582,1032  
Vertex [@=relative, a=arc, c=close,u=undo,Esc=cancel]: 595,1015  
Vertex [@=relative, a=arc, c=close,u=undo,Esc=cancel]: c
```

And again, excavation #3:

Select: *BOUNDARIES* → *ADD EXCAVATION*

```
Enter Excavation CCW, Starting vertex: 531,1048  
Vertex [@=relative, a=arc, c=close,u=undo,Esc=cancel]: 538,1048  
Vertex [@=relative, a=arc, c=close,u=undo,Esc=cancel]: 538,1052  
Vertex [@=relative, a=arc, c=close,u=undo,Esc=cancel]: 531,1052  
Vertex [@=relative, a=arc, c=close,u=undo,Esc=cancel]: c
```

And again, excavation #4:

Select: *BOUNDARIES* → *ADD EXCAVATION*

```
Enter Excavation CCW, Starting vertex: 531,1012  
Vertex [@=relative, a=arc, c=close,u=undo,Esc=cancel]: 538,1012  
Vertex [@=relative, a=arc, c=close,u=undo,Esc=cancel]: 538,1016  
Vertex [@=relative, a=arc, c=close,u=undo,Esc=cancel]: 531,1016  
Vertex [@=relative, a=arc, c=close,u=undo,Esc=cancel]: c
```

4) STRESS GRID

You have now finished defining the excavation geometry. As you probably noticed, EXAMINE^{2D} automatically generated a Stress Grid and computed the boundary element analysis as soon as you finished the first excavation. The stress grid defines the locations within the rock mass where stresses and displacements are to be calculated. In this exercise, we will define our own rectangular grid of stress points, which encompass the entire geometry. Therefore:

Delete the automatically generated stress grid (right-click on the grid and select delete).

Select: *STRESSGRID* → *ADD STRESS GRID*

In the prompt line, enter the following coordinates:

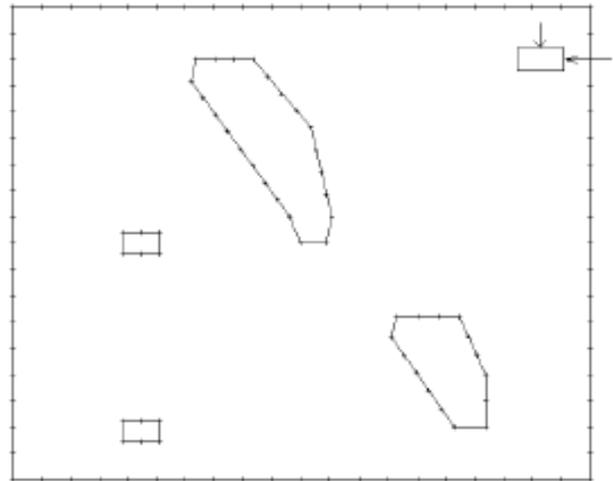
*Select Grid Corner #1: **510,1005***

*Select Grid Corner #2: **620,1095***

In the dialog window:

*Enter number of Intervals in X direction: **23***

*Enter number of Intervals in Y direction: **19***



With the boundary element analysis, it's important to understand that this stress grid only represents the points where stresses and displacement values will be calculated. It does not represent an external boundary that has an influence on the calculated values. As will be seen in next week's exercise, this is in contrast to differential techniques (e.g. finite element, finite difference, etc.).

5) FIELD STRESS

In EXAMINE^{2D} there are two possible far field stress distributions: *CONSTANT*, and *GRAVITATIONAL*, that can be used to define the initial in situ stress field. For this lab, we will be using a *CONSTANT* far field stress field.

Under the *IN SITU FIELD STRESS* in the right sidebar, click the drop down arrow and select *Constant Field Stress*. Enter 40 MPa for Sigma 1, and 20 MPa for Sigma 3. Set the out of plane stress, Sigma Z, as the average of Sigma 1 and Sigma 3. Keep the angle at 0 degrees. This specifies the orientation of the

major principal stress (Sigma 1) relative to the positive horizontal axis (measured counter-clockwise from the horizontal axis).

6) FAILURE CRITERION

In most boundary element analyses, the definition of a failure criterion is not required since the solution is elastic. In other words, the failure criterion is not part of the *stress or displacement* calculations but is only used afterwards as an overlay to compare the calculated stresses to the strength of the rock. As such, the use of the failure criterion can be used in determining the zones of overstress based on the stress results from the elastic analysis (i.e. Strength Factor). For this exercise, we will define a set of strength parameters using the Hoek-Brown strength criterion.

In the right sidebar, choose Hoek-Brown (m,s,a) from the drop down arrow. Set m equal to 5.0, s equal to 0.02 and the intact uniaxial compressive strength as 40 MPa. Use the default for a , 0.503.

7) ELASTIC PROPERTIES

Now define the elastic properties of the rock mass.

In the right side bar, choose the Young's modulus as being 30000 MPa and the Poisson ratio equal to 0.2.

8) COMPUTE

As you might have noticed, EXAMINE^{2D} automatically re-computes the analysis each time the input data is changed. Therefore, the displayed contours always correspond to the current input data. However, the *contour range* of the data in the legend does not automatically change when you edit your model.



Therefore, go to Analysis → Compute Contour Range or click the option in the toolbar to update the contour range in the legend.

Save your file. EXAMINE^{2D} model files have an .exa extension. As a result, when prompted for a filename, you do not have to, and shouldn't, type the .exa extension.

9) INTERPRET - STRESSES

This phase of the analysis is the most important. Using the analysis results and the data interpretation facilities in EXAMINE^{2D}, you will proceed with the process of determining the stress distribution within the rock mass. As well, the regions of overstress based on the elastic stress analysis can be determined. This can provide an important insight into the overall stability of the underground openings.

There are a variety of methods for looking at the data. We will proceed in exploring some of these options.

Go to the drop-list in the toolbar. By default, sigma 1 is displayed. To view other data contours, select the drop-down arrow. You will immediately notice a wide range of datasets that can be viewed. The datasets include the principal stresses, strength factor, displacements, user defined functions, and global stress tensor



components. We will first view the distribution of the major principal stress by selecting the SIGMA 1 option.

Using the values calculated at the grid points, solid color contours are created within the user-defined stress grid. Now let's view the direction of the principal stresses at each grid point:

 Select the Stress Trajectories icon in the toolbar. Alternatively, you can also right-click anywhere in the model and select Display Options. Under the General tab, select Stress Trajectories.

The stress trajectories are shown as small cross icons where the long axis of the cross is oriented in the direction of the major principal stress while the short axis is the direction of the minor principal stress. **On your answer sheet, use FLOW LINES to sketch the Sigma 1 trajectories relative to the underground openings.**

10) INTERPRET - FAILURE

Now let's compare the stresses to the rock strength defined earlier (using a Hoek-Brown failure criterion).

From the drop-list in the toolbar, select Strength Factor.

Just a quick note on the strength factor term used here: In two dimensions, the strength factor is calculated by dividing the rock strength in terms of the maximum internal shear $(\sigma_1 - \sigma_3)/2$ at failure for a given confining pressure $(\sigma_1 + \sigma_3)/2$ at a point, by the maximum induced internal shear at that point due to the excavation. In simpler terms, the strength factor is the ratio of material strength to the induced stress.

Note that this definition of strength factor is different from the definition of factor of safety used previously. The term is more applicable to providing a picture of overstress around an excavation. Still, *the strength factor less than 1 encompasses zones that are critically stressed and would fail under the given stress conditions.*

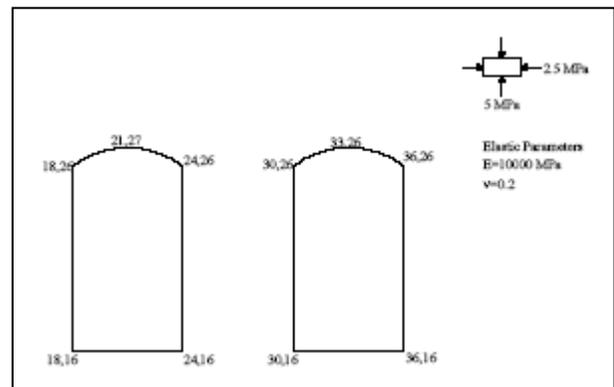
 For visualization purposes, you can filter the contours. In the toolbar, click the Contour Options icon. Also, you can right-click anywhere on the legend and select Contour Options. Under Custom Range, enter 0 as the minimum value and 1 as the maximum value. Select Apply. You should now only see the critical areas having a strength factor of less than 1.

On your answer sheet, comment on the potential for failure in this case.

Part B – Influence of Neighbouring Excavations

This exercise will examine how neighbouring excavations influence one another in terms of induced stresses and failure.

Open a new file in Examine2D



1) SET LIMITS

As mentioned earlier, the first step in any computer-aided design process is setting the drawing limits of the region so that the limits encompass the excavation geometry.

Select: *VIEW* → *LIMIT*

Enter the minimum and maximum x-y coordinates as shown below. These limits will approximately centre the model in the drawing region.

Enter MINIMUM X,Y COORDINATES: 4,2
Enter MAXIMUM X,Y COORDINATES: 50,40

2) DEFINE GEOMETRY

Now define the excavation geometry. Enter the two excavation boundaries using the following sequence:

Select: *BOUNDARIES* → *ADD EXCAVATION*

Enter Excavation CCW, Starting vertex: 18,16
Vertex [@=relative, a=arc, c=close,u=undo,Esc=cancel]: 24,16
Vertex [@=relative, a=arc, c=close,u=undo,Esc=cancel]: 24,26
Vertex [@=relative, a=arc, c=close,u=undo,Esc=cancel]: a
In the Arc Options Dialog, enter the default values:
Arc definition method: 3 points on arc
Number of segments in arc: 8
Enter second arc point: 21,27
Enter third arc point: 18,26
Vertex [@=relative, a=arc, c=close,u=undo,Esc=cancel]: c

Select: *BOUNDARIES* → *ADD EXCAVATION*

Enter Excavation CCW, Starting vertex: 30,26
Vertex [@=relative, a=arc, c=close,u=undo,Esc=cancel]: 30,16
Vertex [@=relative, a=arc, c=close,u=undo,Esc=cancel]: 36,16
Vertex [@=relative, a=arc, c=close,u=undo,Esc=cancel]: 36,26
Vertex [@=relative, a=arc, c=close,u=undo,Esc=cancel]: a
In the Arc Options Dialog, enter the default values:
Arc definition method: 3 points on arc
Number of segments in arc: 8
Enter second arc point: 33,27
Enter third arc point: c

3) DISCRETIZATION

The next step is to discretize the line segments making up the openings, into boundary elements. To do this:

Select: ANALYSIS → PROJECT SETTINGS

Under the Analysis tab, change the default number of boundary elements to 68. EXAMINE^{2D} will now discretize the geometry into (approximately) 68 boundary elements, instead of the former default value of 100. This process will create approximately 68 boundary elements (total) around the perimeter of the two excavations.

Number of Boundary Elements: 68

4) STRESS GRID

You have now finished defining the excavation geometry. As noted previously, EXAMINE^{2D} automatically generated a Stress Grid and computed the boundary element analysis as soon as you finished the first excavation. In this exercise, we will define our own rectangular grid of stress points, which encompass the entire geometry. Therefore:

Delete the automatically generated stress grid (right-click on the grid and select delete).

Select: *STRESSGRID* → *ADD STRESS GRID*

In the prompt line, enter the following coordinates:

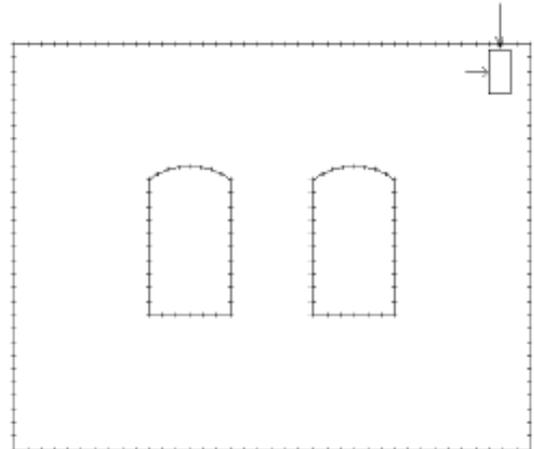
Select Grid Corner #1: 5,5

Select Grid Corner #2: 50,40

In the dialog window:

Enter number of Intervals in X direction: 45

Enter number of Intervals in Y direction: 35



5) FIELD STRESS

Now proceed to enter the far field stress state. For this exercise, we will be using a CONSTANT far field stress field.

Under the INSITU FIELD STRESS in the right sidebar, click the drop down arrow and select Constant Field Stress. Enter 5 MPa for Sigma 1, and 2.5 MPa for Sigma 3. Set the out of plane stress, Sigma Z, as the average of Sigma 1 and Sigma 3. Define the angle as being 90 degrees (i.e. vertical). Note that the stress block icon in the upper right corner of the screen has changed orientation and magnitude.

6) ELASTIC PROPERTIES

Now define the elastic properties of the rock mass.

In the right side bar, set the Young's modulus to 10000 MPa and the Poisson ratio equal to 0.2.

7) INTERPRET - STRESSES

The regions of overstressed rock based on the elastic stress analysis can now be determined. We will first view the distribution of the major principal stress by selecting the SIGMA 1 and SIGMA 3 options.

In the toolbar, click the drop-down list. Select Sigma 1 and Sigma 3 to see stresses around the openings.

To help with your interpretation, you can query the boundary of your excavation in order to show the values of the contours. Right-click on the excavation boundary and select *Query Boundary*. The displayed bars along the excavation boundary represent the relative magnitude of the data value. To display the values of the bars, right-click on the excavation boundary and select Display Options. Under the Query tab, select the Values checkbox.

On your answer sheet, comment on the σ_1 and σ_3 stress states in the pillar between the two excavations.

8) INTERPRET - FAILURE

Now let's compare the stresses to the rock strength using a Mohr-Coulomb failure criterion.

In the right side bar, enter the following strength parameters: tensile strength = 0 MPa; cohesion = 10 MPa; friction = 30°.



To help with your analysis of the strength factor of the pillar rock between the two excavations, you can *Add a Query Material* by selecting the icon from the toolbar or the Query menu. Draw a straight line from the left excavation (center of right wall) towards to right excavation (center of left wall). Right click and select Done. You should now see the Strength Factor values along the pillar.

On your answer sheet, report the strength factor of the pillar rock between the two excavations. Repeat the Interpret Steps above to determine and report the strength factors when the tensile strength is 0 MPa, the friction angle is 30° and cohesion = 5 MPa and 2 MPa.

9) INTERPRET – FAILURE 2

The excavations analyzed in the model are two powerhouse caverns for a hydroelectric project. The designers are concerned by new lab tests that report the rock mass forming the pillar is weaker than expected ($T_o = 0$ MPa, $\phi = 30^\circ$, $c = 1$ MPa). Although the rock is strong enough when only one power house chamber is excavated, it is too weak when the influence of the neighbouring excavation is accounted for, at least with respect to its presently planned location.

Return to the modeller and edit the location of the second excavation to determine what minimum distance of separation is required for the strength factor to be greater than one in the pillar between the excavations (Remember: based on a Mohr-Coulomb failure criterion where $T_o = 0$ MPa, $\phi = 30^\circ$ and $c = 1$ MPa.)

Select Excavation(s) to Move: Click on any part of the right-hand excavation (the line will turn dashed)

To move excavation boundary:

Hover the cursor over the excavation boundary and click. You will see the four-way arrow icon, which indicates that you can move the boundary with the mouse. Click and drag the excavation, and you will see that the analysis contours (in this case Strength Factor) are immediately updated, in real time, as you move the boundary. This capability is referred to as real time contouring or real time stress analysis in Examine2D.

OR

You could move the position of the excavation relative to the horizontal. With the (right-hand) excavation selected, type “m” in the prompt line. Enter 1 to move the boundary at 1m increments and click OK. Use the right arrow in your keyboard to move the boundary. In this case, the excavation will move to the right hand side in the x direction but not in the y direction.

Proceed to check your results and the new strength factor for the pillar rock between the two excavations. Remember to make sure that you are looking at the Mohr-Coulomb Strength Factor and using $T_o = 0$ MPa, $\phi = 30^\circ$ and $c = 1$ MPa.

Repeat this procedure until you find the minimum pillar width (i.e. the minimum distance between the two excavations), where the strength factor is greater than one in the central region of the pillar. ***On your answer sheet, report this minimum pillar width.***