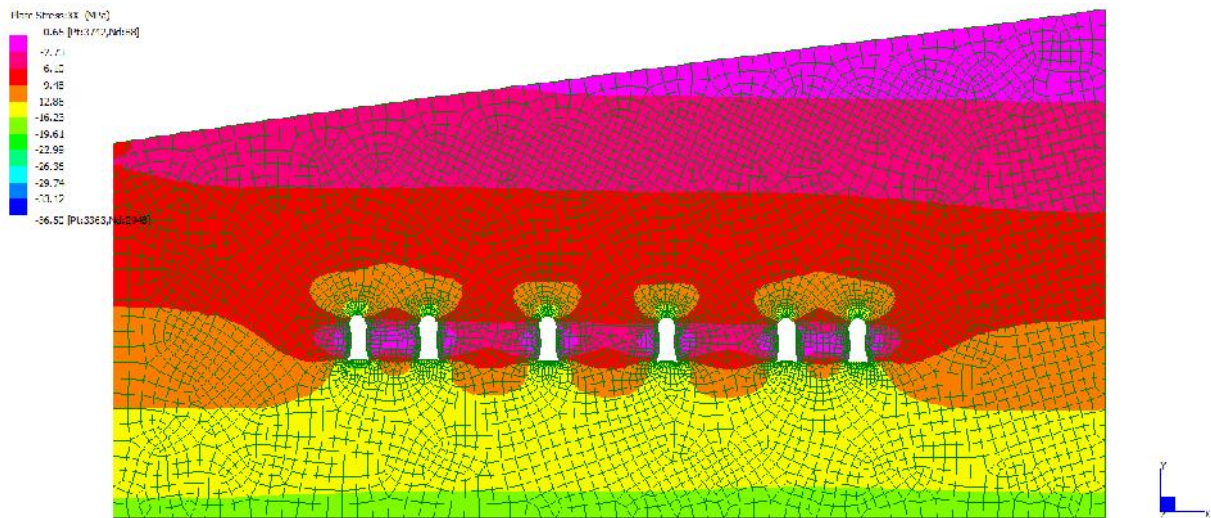


LABORATORY MANUAL

NUMERICAL MODELLING LAB

**for
VI- B. Tech (ME)**



**DEPARTMENT OF MINING ENGINEERING
IIT(ISM), DHANBAD**

Course Type	Course Code	Name of Course	L	T	P	Credit
DP10	MNC 307	Numerical Modelling / Remote Sensing & GIS Lab. (Modular)	0	0	2	2

Course Objective
To provide skills in operating latest software in numerical modelling, remote sensing and GIS applications
Learning Outcomes
<ul style="list-style-type: none"> will be able to design various structures in rock will be able to process various remote sensing data apply GIS and SAR to predict mine subsidence

Sl. No.	Major Topics	No. of Practicals	Learning outcomes
Numerical Modelling			
1	To perform finite element analysis of stress around a circular tunnel	1	basics of applying 2D numerical modelling
2	Study effect of mesh size on stress distribution around the circular tunnel and comparing the numerical solution with closed form solution	1	influence of mesh size on results of numerical modelling
3	Modelling of underground excavations in massive rockmass	1	understanding of 3D numerical modelling
4	To perform finite element analysis of a rib pillars and Modelling of sequence of excavation and design of stopes/cavern	1	solving metal mining geotechnical problems
5	Modelling of mechanical behaviour of pillars under different geo-mining conditions	1	modelling of coal mine pillars and understanding its strength behaviour
6	Modelling of a hydroelectric cavern and gas oil storage cavern	1	solving problems w.r.t. large civil underground structure
Remote Sensing & GIS Lab.			
7	Introduction to different types of remote sensing data products	1	Remote sensing data products and their uses in various applications
8	Visual interpretation of various features and Analysis on a satellite data.	1	data processing of satellite data and interpretation of results
9	Demonstration on various GIS software's and their salient features	1	An overview of the capabilities of GIS platform for a series of mining applications
10	Georeferencing of various maps and Satellite image & Digitisation for documentation of Mine Lease Boundaries	1	Geo-referencing and digitisation of cadastral map with lease boundary of mining areas
11	Preparation of Land Use/ Land Cover Map from the satellite data	1	Land pattern usages and change detection within lease area of mine
12	Mine subsidence modeling using Spaceborne SAR interferometry technique	1	Radar image processing to estimate subsidence.

List of Practicals for Numerical Modelling Lab

1. To perform finite element analysis of stress around a circular tunnel
2. Study effect of mesh size on stress distribution around the circular tunnel and comparing the numerical solution with closed form solution
3. Modelling of underground excavations in massive rockmass
4. To perform finite element analysis of a rib pillars and Modelling of sequence of excavation and design of stopes/cavern
5. Modelling of mechanical behaviour of pillars under different geo-mining conditions
6. Modelling of a hydroelectric cavern and gas oil storage cavern

Evaluation of Practical (weightage):

1. Regular Submission of Practical Note Book with observations (30%)
2. Performance & Attendance (20%)
3. End Sem Practical Examination: 50% (Practical Exam + Viva Voce)

Do's & Dont's:

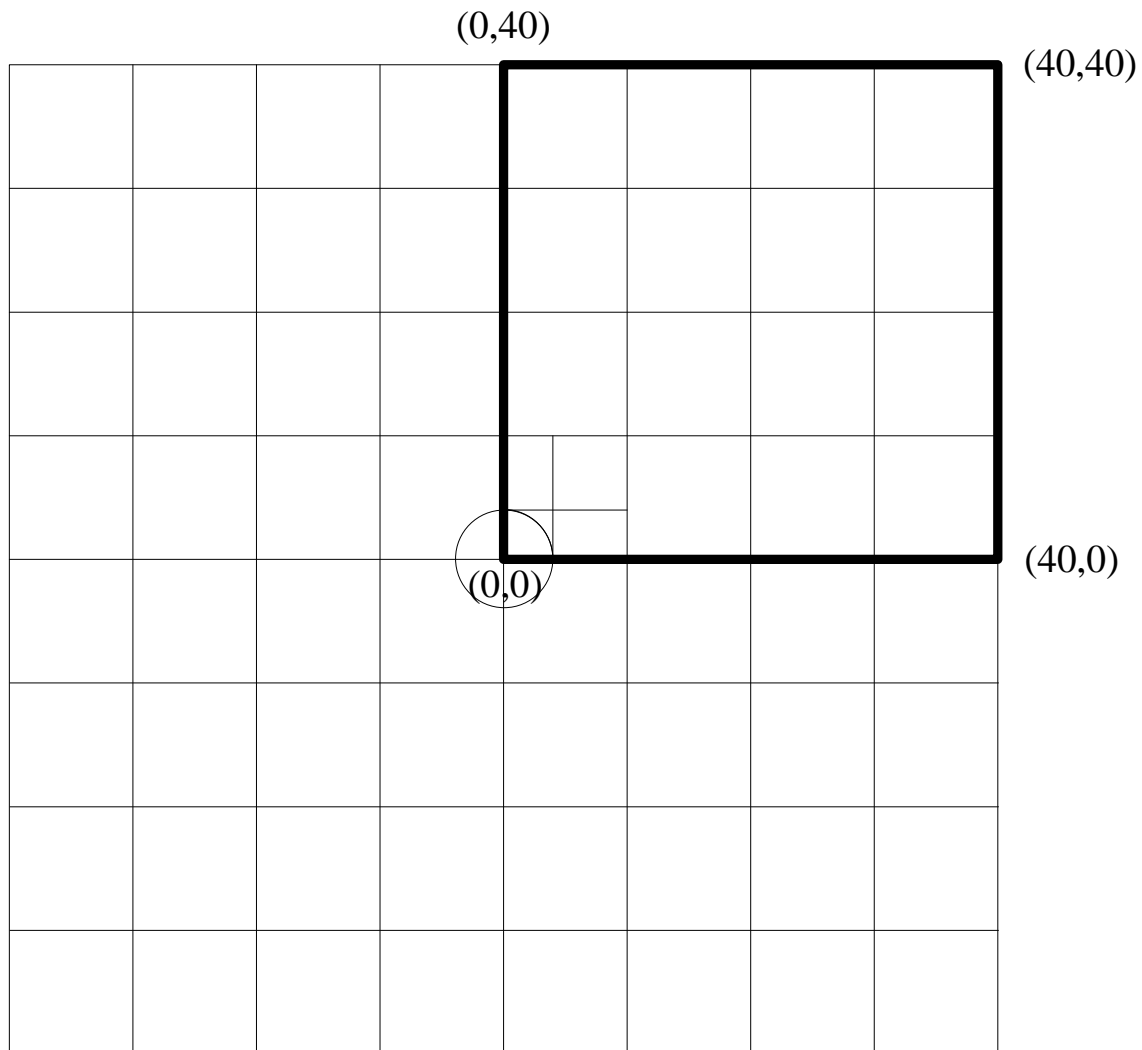
The following instructions may be followed while conducting practicals in the above mentioned laboratories:

1. Read the laboratory manuals/procedures supplied to you in advance, carefully and completely and make your own observations on the type, procedure and care that needs to be exercised in each practical.
2. Listen to the instructions of the instructor and associated scholar meticulously. Any doubts with regards to usage please do not hesitate to contact any one of the above.
3. Do not leave any of your belongings in the laboratory and check before you leave.
4. Do not roam around the room, distract other students or startle other students.
5. Do not open any irrelevant internet sites on lab computer.
6. Do not use a flash drive on lab computers.
7. Do not upload, delete or alter any software on the lab PC.
8. Bring Notebook, pen/ pencil, scale, calculator during each practical class.
9. Understand the assignment and Aim of the experiment.
10. Team work is a must to complete the assignment in a scheduled time frame.

NM - 1: To Perform Finite Element Analysis of Stress around A Circular Tunnel

Aim: Perform Finite Element Analysis of Stresses around of a circular tunnel of 8m dia. Observe the changes in stress distribution around the opening. $\sigma_v = 5$ MPa, $\sigma_h = 10$ MPa, $E = 10$ GPa, $\nu = 0.25$, material isotropic elastic, $\rho = 2600$ kg/m³

Model: Prepare the model using pre-processing modules of the FEM package.



Course mesh: Minimum size of element near the boundary 1.5m x 1.5m

Grading : Proper fine grading of the mesh towards the opening.

Observation Table

Table 1: σ_{xx} & σ_{yy} at the nodes along x-axis for the course mesh
(size of the elements at the boundary =)

Distance from center, m (1)	Closed form solution (2)		Course mesh (3)		Difference (2 – 3)		Remarks
4	σ_{xx}	σ_{yy}	σ_{xx}	σ_{yy}	σ_{xx}	σ_{yy}	

Table 2: σ_{xx} & σ_{yy} at the nodes along y-axis for the course mesh
(size of the elements at the boundary =)

Distance from center, m (1)	Closed form solution (2)		Course mesh (3)		Difference (2 – 3)		Remarks
4	σ_{xx}	σ_{yy}	σ_{xx}	σ_{yy}	σ_{xx}	σ_{yy}	

Topics Covered

- Generate FEM mesh
- Boundary conditions
- Material properties
- Post processing

INSTRUCTION SHEET

2D- FEM modelling of stress around a circular tunnel of 8m diameter using Strand7

STEPS TO BE FOLLOWED:

1. Click on the icon of Strand7 server mode on the desktop
2. Click on menu **file – new**
3. Screen will be displayed for selecting model unit as displayed below
4. Choose SI unit and change the Modulus/ Stress to MPA, Force to kN, let the other units as it is. Click on OK button

CREATING FEM MESH

1. Go to the menu **Create – node**. The first node will have coordinate (0,0,0) (X, Y, Z). Press apply button. The second node will have the coordinate (40, 0, 0). Again press apply button. Repeat the above for third and fourth coordinates. The third node will have coordinate of (40,40,0) and the forth node will have coordinate of (0,40,0).
2. Press **F3** button in keyboard to view all the four nodes.
3. Go to the menu **Create – element**. A dialog box appears at the left side of screen. Choose the **Quad4** element for 4 noded quadrilateral elements from the box. Click on the first node. A rubber band will appear. With the rubber band move the mouse to the second node and select by clicking the left button of the mouse, subsequently select third node and finally fourth node. The rubber band will vanish. **Move anticlockwise from 1st node in selecting the nodes.**
4. Click on menu **Tool – subdivide**. A dialog box appears in the left side of screen. Divide the quad4 element into 16 small square (each square of 10m length) by entering the value A = 4 and B = 4 under division in the dialog box and press apply.
5. In order to create a circular tunnel of 8m diameters, click on the **Tool – grade plates and bricks**. Select the **quarter circle cut** from the grade bar. Select **first two sides** of first quad (small square) element starting from node (0,0,0). Enter the **ratio (4/10)** into ratio box under the grade bar.
6. Go to the Menu **Tool – Subdivide** for further **subdividing** the first square containing the circular element into 4 subdivisions along the both side A and B. Enter A = 4 and B = 4 in the subdivision dialogue box. Select the all the element present in the first square and then press apply button in the subdivision window.
7. Select all other square and subdivide each square into 16 small squares.
8. Go to menu **Tool – align – plate axis** and select all elements to be aligned with the global axis.
9. Go to menu **Tool – clean-mesh** for removing the extra unused nodes and plates.

DEFINING LOAD CASES

1. Go to menu Global – load and Freedom cases. Select freedom cases and check 2D Plane in auto set. Select Primary load cases and edit case name to Load case1: Sh=-10MPa. Select new load case and edit it to Load case 2:Sv=-5MPa,
2. Go to menu **Attribute – plate – pre- (load) stress** and apply pre-stress of -10 MPa (compression) along the x- axis as **Load case 1 Sh=-10MPa** and -5 MPa (compression) along the y – axis as **Load case 2:Sv=-5MPa** separately. (You have to select the appropriate load cases).

DEFINING BOUDBARY CONDITIONS

1. Go to the menu **attribute - node - restrain** and fix the boundary of the model by restricting the translation either along the X, and/or Y and/or Z direction depending on the selection as per boundary conditions. For example, select the nodes of the lower horizontal boundary, select **X-sym** from **Node Attribute** menu and press **apply**.

DEFINE MATERIAL PROPERTY

1. Now go to the menu **property - plate** and for **plate property 1** select **2-D plain strain** condition and enter the following properties of plate
Select the material type as **Isotropic** and enter: Modulus = 10×10^3 MPa,
Density = 2600 kg/m^3 , Poisson's ratio = 0.25
Edit the name of the plate as **Sandstone**.

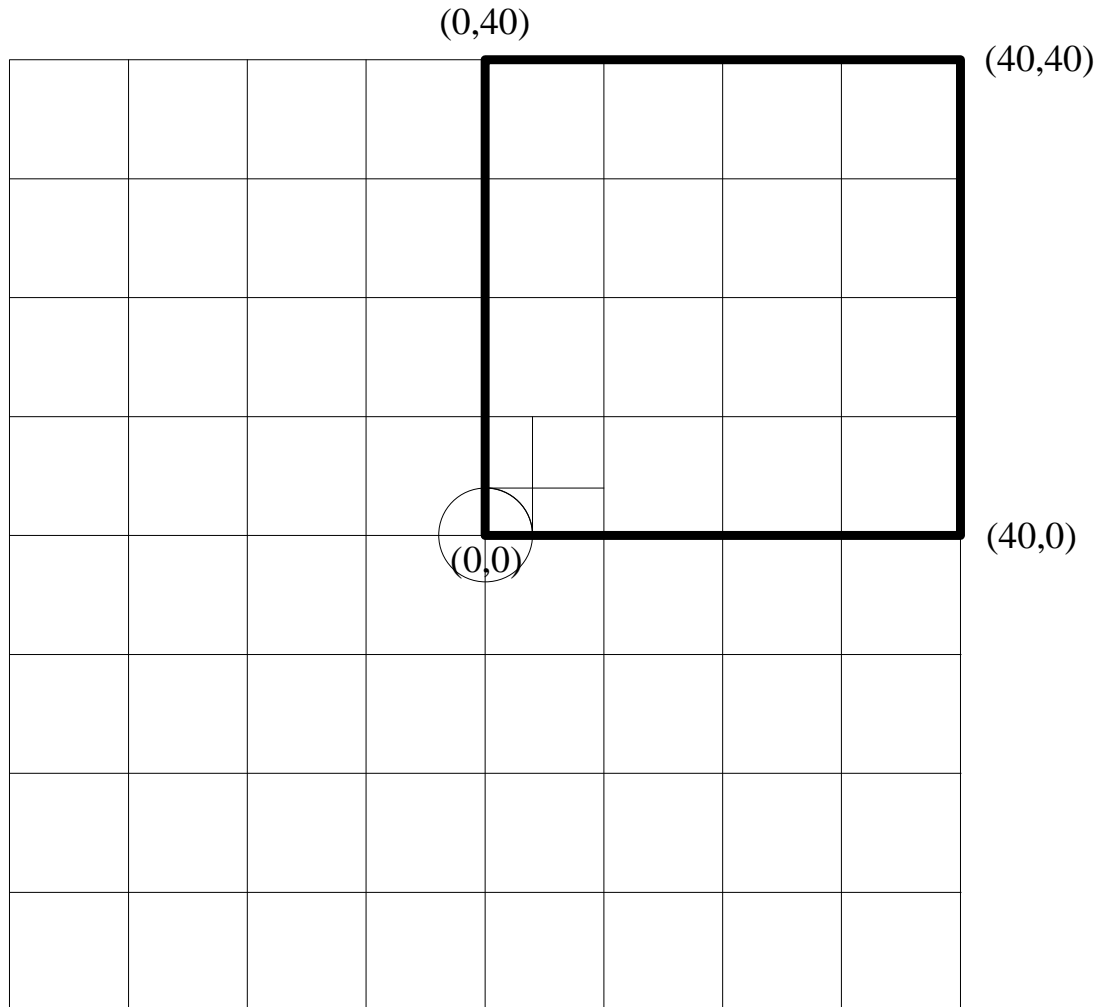
SOLVE

1. Now go to menu **solver – linear static** and select **calculate – plate stress**
2. Finally click on the button **solve**
3. Wait for the dialog to appear
4. Once the solution is done, combine the result cases using menu **result-linear load case combination**.
5. Open the result file by going to menu- **result- open the result file**. Also view the log file.
6. You should be able to see the results.

NM - 2: Study the effect of mesh size on stress distribution around the circular tunnel and comparing the numerical solution with closed form solution

Aim: Perform Finite Element Analysis of Stresses around of a circular tunnel of 8m dia. Study effect of mesh size on stress distribution around the opening. $\sigma_v = 5$ MPa, $\sigma_h = 10$ MPa, $E = 10$ GPa, $\nu = 0.25$, material isotropic elastic, $\rho = 2600$ kg/m³

Model: Prepare the model using pre-processing modules of the FEM package.



Course mesh: Minimum size of element near the boundary $1.5\text{m} \times 1.5\text{m}$

Fine mesh: Minimum size of element near the boundary $0.25\text{ m} \times 0.25\text{m}$.

Refine the FE mesh by subdividing and sub-grading.

Grading : Proper fine grading of the mesh towards the opening.

Observation Table

Table 1: σ_{xx} & σ_{yy} at the nodes along x-axis for the course mesh
(size of the elements at the boundary =)

Distance from center, m (1)	Closed form solution (2)		Course mesh (3)		Difference (2 – 3)		Remarks
4	σ_{xx}	σ_{yy}	σ_{xx}	σ_{yy}	σ_{xx}	σ_{yy}	

Table 2: σ_{xx} & σ_{yy} at the nodes along y-axis for the course mesh
(Size of the elements at the boundary =)

Distance from center, m (1)	Closed form solution (2)		Course mesh (3)		Difference (2 – 3)		Remarks
4	σ_{xx}	σ_{yy}	σ_{xx}	σ_{yy}	σ_{xx}	σ_{yy}	

Table 3: σ_{xx} & σ_{yy} at the nodes along x-axis for the fine mesh
(Size of the elements at the boundary =)

Distance from center, m (1)	Closed form solution (2)		Course mesh (3)		Difference (2 – 3)		Remarks
4	σ_{xx}	σ_{yy}	σ_{xx}	σ_{yy}	σ_{xx}	σ_{yy}	

Table 4: σ_{xx} & σ_{yy} at the nodes along y-axis for the fine mesh
(Size of the elements at the boundary =)

Distance from center, m (1)	Closed form solution (2)		Course mesh (3)		Difference (2 – 3)		Remarks
4	σ_{xx}	σ_{yy}	σ_{xx}	σ_{yy}	σ_{xx}	σ_{yy}	

Topics Covered

- Generate FEM mesh
- Boundary conditions
- Material properties
- Post processing

INSTRUCTION SHEET

2D- FEM modeling of stress around a circular tunnel of 8m diameter using Strand7

STEPS TO BE FOLLOWED:

5. Click on the icon of Strand7 server mode on the desktop
6. Click on menu **file – new**
7. Screen will be displayed for selecting model unit as displayed below
8. Choose SI unit and change the Modulus/ Stress to MPA, Force to kN, let the other units as it is. Click on OK button

CREATING FEM MESH

10. Go to the menu **Create – node**. The first node will have coordinate (0,0,0) (X, Y, Z). Press apply button. The second node will have the coordinate (40, 0, 0). Again press apply button. Repeat the above for third and fourth coordinates. The third node will have coordinate of (40,40,0) and the forth node will have coordinate of (0,40,0).
11. Press **F3** button in keyboard to view all the four nodes.
12. Go to the menu **Create – element**. A dialog box appears at the left side of screen. Choose the **Quad4** element for 4 noded quadrilateral elements from the box. Click on the first node. A rubber band will appear. With the rubber band move the mouse to the second node and select by clicking the left button of the mouse, subsequently select third node and finally fourth node. The rubber band will vanish. **Move anticlockwise from 1st node in selecting the nodes.**
13. Click on menu **Tool – subdivide**. A dialog box appears in the left side of screen. Divide the quad4 element into 16 small square (each square of 10m length) by entering the value $A = 4$ and $B = 4$ under division in the dialog box and press apply.
14. In order to create a circular tunnel of 8m diameters, click on the **Tool – grade plates and bricks**. Select the **quarter circle cut** from the grade bar. Select **first two sides** of first quad (small square) element starting from node (0,0,0). Enter the **ratio (4/10)** into ratio box under the grade bar.
15. Go to the Menu **Tool – Subdivide** for further **subdividing** the first square containing the circular element into 5 subdivisions along the both side A and B. Enter $A = 4$ and $B = 4$ in the subdivision dialogue box. Select the all the element present in the first square and then press apply button in the subdivision window.
16. Select all other square and subdivide each square into 16 small squares.
17. Go to menu **Tool – align – plate axis** and select all elements to be aligned with the global axis.
18. Go to menu **Tool – clean-mesh** for removing the extra unused nodes and plates.

DEFINING LOAD CASES

3. Go to menu Global – load and Freedom cases. Select freedom cases and check 2D Plane in auto set. Select Primary load cases and edit case name to Load case1: Sh=-10MPa. Select new load case and edit it to Load case 2:Sv=-5MPa,
4. Go to menu **Attribute – plate – pre- (load) stress** and apply pre-stress of -10 MPa (compression) along the x- axis as **Load case 1 Sh=-10MPa** and -5 MPa (compression) along the y – axis as **Load case 2:Sv=-5MPa** separately. (You have to select the appropriate load cases).

DEFINING BOUDBARY CONDITIONS

2. Go to the menu **attribute - node - restrain** and fix the boundary of the model by restricting the translation either along the X, and/or Y and/or Z direction depending on the selection as per boundary conditions. For example, select the nodes of the lower horizontal boundary, select **X-sym** from **Node Attribute** menu and press **apply**.

DEFINE MATERIAL PROPERTY

1. Now go to the menu **property - plate** and for **plate property 1** select **2-D plain strain** condition and enter the following properties of plate
Select the material type as **Isotropic** and enter: Modulus = 10×10^3 MPa,
Density = 2600 kg/m^3 , Poisson's ratio = 0.25
Edit the name of the plate as **Sandstone**.

SOLVE

1. Now go to menu **solver – linear static** and select **calculate – plate stress**
 2. Finally click on the button **solve**
 3. Wait for the dialog to appear
 4. Once the solution is done, combine the result cases using menu **result-linear load case combination**.
 5. Open the result file by going to menu- **result- open the result file**. Also view the log file.
 6. You should be able to see the results.
-

Closed form solutions: Stress around a circular tunnel

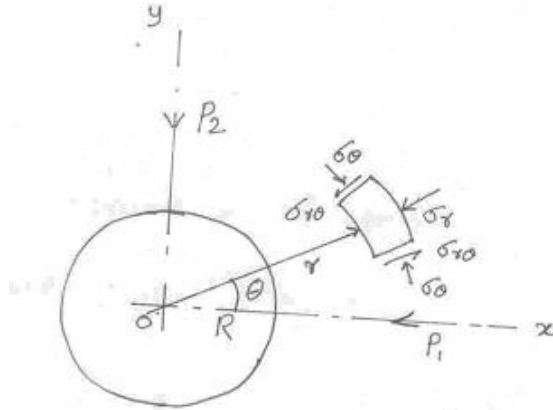


Figure: Stress at a distance r around a circular tunnel.

With reference to the above Figure, where principal stresses P_1 and P_2 are acting at infinity along x and y axes respectively, the stresses at a radial distance r at angle θ are given as

$$\sigma_r = \frac{1}{2}(P_1 + P_2) \left(1 - \frac{R^2}{r^2} \right) + \frac{1}{2}(P_1 - P_2) \left(1 - \frac{4R^2}{r^2} + \frac{3R^4}{r^4} \right) \cos 2\theta \quad (1)$$

$$\sigma_\theta = \frac{1}{2}(P_1 + P_2) \left(1 + \frac{R^2}{r^2} \right) - \frac{1}{2}(P_1 - P_2) \left(1 + \frac{3R^4}{r^4} \right) \cos 2\theta \quad (2)$$

$$\sigma_{r\theta} = -\frac{1}{2}(P_1 - P_2) \left(1 + \frac{2R^2}{r^2} - \frac{3R^4}{r^4} \right) \sin 2\theta \quad (3)$$

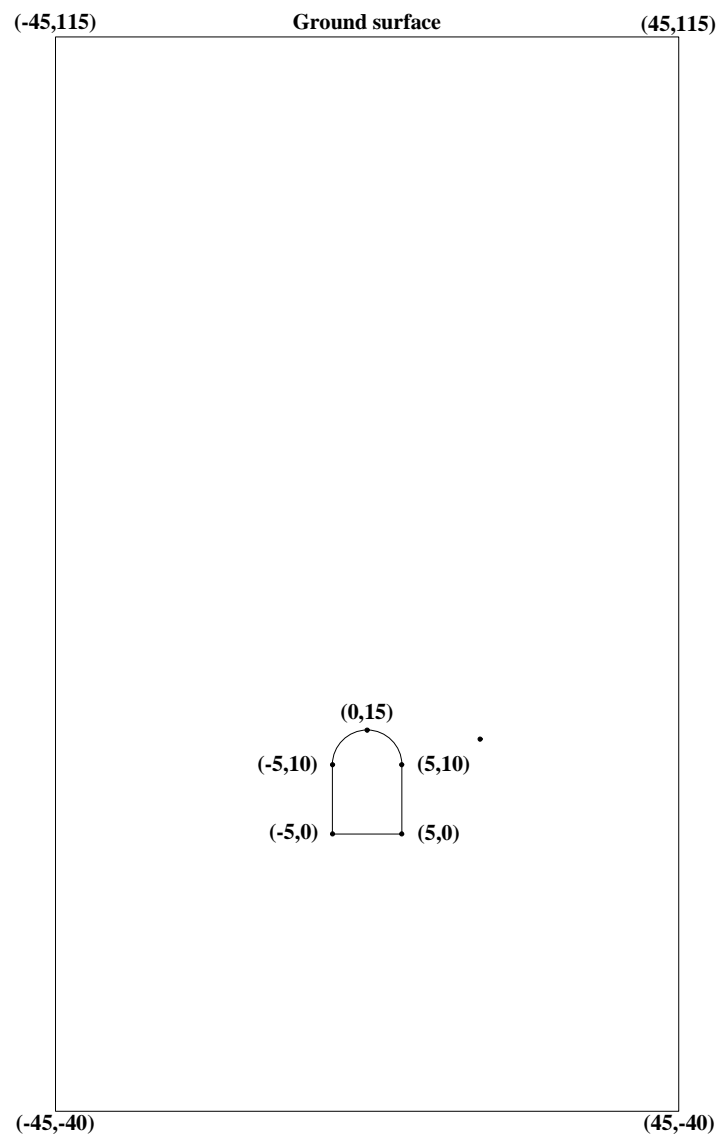
where, σ_r radial stress, σ_θ is tangential stress and $\sigma_{r\theta}$ is shear stress.

NM – 3: Modelling of underground excavations in massive rockmass

Aim: Perform Finite Element Analysis of Stresses around of an underground excavation in massive rockmass and observe the changes in stress distribution around the excavation.

Rock propertires are: density = 2650 kg/m^3 , Young's modulus = 5 GPa, poisson's ratio = 0.22.
Insitu vertical stress along Y-axis is 2 times the horizontal stress.

Model: Prepare the model using pre-processing modules of the FEM package.



Observation Table

Table 1: σ_{xx} & σ_{yy} at the nodes along x-axis
(size of the elements at the boundary =)

Distance from center of the excavation, m	Insitu results (1)		FEA results (2)		Difference (1 – 2)		Remarks
	σ_{xx}	σ_{yy}	σ_{xx}	σ_{yy}	σ_{xx}	σ_{yy}	

Table 2: σ_{xx} & σ_{yy} at the nodes along y-axis
(size of the elements at the boundary =)

Distance from center of the excavation, m	Insitu results (1)		FEA results (2)		Difference (1 – 2)		Remarks
	σ_{xx}	σ_{yy}	σ_{xx}	σ_{yy}	σ_{xx}	σ_{yy}	

Topics Covered

- Generate FEM mesh
- Boundary conditions
- Material properties
- Stages
- Post processing

INSTRUCTION SHEET

2D- FEM modelling of stress around an underground excavation using Strand7

STEPS TO BE FOLLOWED:

9. Click on the icon of Strand7 server mode on the desktop
10. Click on menu **file – new**
11. Screen will be displayed for selecting model unit as displayed below
12. Choose SI unit and change the Modulus/ Stress to MPA, Force to kN, let the other units as it is. Click on OK button

CREATING FEM MESH

19. Go to the menu **Create – node**. The first node will have coordinate (-5,0,0) in (X, Y, Z). Press apply button. The second node will have the coordinate (5, 0, 0). Again press apply button. Repeat the above for third and fourth coordinates. The third node will have coordinate of (5,10,0) and the forth node will have coordinate of (-5,10,0).
20. Press **F3** button in keyboard to view all the four nodes.
21. Go to **Tools > Points and Lines**. Select Find Circle Centre. Select the top 3 nodes using cursor as 1, 2, 3 points. Click Apply. Circle centre is obtained.
22. Go to the menu **Create – element**. A dialog box appears at the left side of screen. Choose the **Beam element** from the box. Click on the first node. A rubber band will appear. With the rubber band move the mouse to the second node and select by clicking the left button of the mouse. The rubber band will vanish. Finally, close the dialog box. Geometry of excavation is created.
23. Create the domain of the problem: Go to the menu **Create – node**. The first node will have coordinate (-45,-40,0) in (X, Y, Z). Press apply button. The second node will have the coordinate (45, -40, 0). Again press apply button. Repeat the above for third and fourth coordinates. The third node will have coordinate of (45,115,0) and the forth node will have coordinate of (-45,115,0).
24. Select 'Toggle Beam Select'. Select All. Go to **Tools > Geometry Tools> Face from Beam Polygon**. Enter 5 in 'Edge Tol'. Click Apply. Close the dialog box. Right click on (**hide entity**) in vertical toolbar on the left side window. A dialog box appears. Select 'Show Wireframes'. Click Apply.
25. Select 'Toggle Face Select'. Select All. Go to **Tools > Geometry Tools > Graft Faces to Edges**. A dialog box appears. Click Apply.
26. Go to **Tools > Clean > Geometry**. Select 'Leave One Face'. Click Apply.
27. Go to **Global > Groups**. A dialog box appears. Create group under 'Model' using 'New'. Rename as excavation.
28. Go to **Global > Stages**. A dialog box appears. Create 2 stages using 'New'. Rename them as Virgin and excavation.
29. **Meshing:** Go to **Attributes > Vertex > Mesh Size**. Enter 1 in 'Value'. Click Apply. Go to **Tools > Automeshing> Surface Mesh**. A dialog box appears. Enter 10 in 'Maximum Edge Length'. Click Mesh.
30. Go to menu **Tool – align – plate axis** and select all elements to be aligned with the global axis.

DEFINING LOAD CASES

1. Go to menu **Global – load and Freedom cases**. Select freedom cases and check **2D Plane** in auto set. Select **Primary load cases** and edit case name to **Load case1: Gravity**. Select new load case and edit it to **Load case 2:Sh = 2×Sv**

2. Go to menu **Attribute – plate – pre- (load) stress** and apply pre-stresses separately. (You have to select the appropriate load cases).
3. Select 'Toggle Plate Select'. Select All. Go to Tools > Align > Plate Axes. Select 'x' and 'X'. Click Apply.

DEFINING BOUDBARY CONDITIONS

1. Go to the menu **attribute - node - restrain** and fix the boundary of the model by restricting the translation either along the X, and/or Y and/or Z direction depending on the selection as per boundary conditions. For example, select the nodes of the lower horizontal boundary, select **Fix** from **Node Attribute** menu and press **apply**.

DEFINE MATERIAL PROPERTY

1. Now go to the menu **property - plate** and for **plate property 1** select **2-D plain strain** condition and enter the following properties of plate
 Select the material type as **Isotropic** and enter: Modulus = 50×10^3 MPa, Density = 2650 kg/m^3 , Poisson's ratio = 0.22
Edit the name of the plate.

SOLVE

1. Now go to menu **solver – nonlinear static** and select **calculate – plate stress**
 - a. Click **load increment**
 - b. Select **staged analysis**
 - c. **Solve**
2. Open the **result file**. In **contour** option uncheck **interpolation to nodes**.
3. Check the correctness of the stress analysis result: check deformation and stresses at various places and judge their correctness.

NM – 4a: To perform finite element analysis of a rib pillars

Aim: Perform FE analysis of a 24m wide rib pillars at a depth of 200m between two horizontal panels of 100m wide each. observe the changes in stress distribution around the excavations and rib pillar for the property set - 1.

Property Set-1

Pillar rock

Modulus of elasticity, E = 2 GPa

Poisson' ratio = 0.15

Unit Weight, ρ = 1400 kg/m³

Roof and floor rock

Modulus of elasticity, E = 10 GPa

Poisson' ratio = 0.25

Unit Weight, ρ = 2200 kg/m³

In-situ Stress: gravity load: (9.8 m/sec² acceleration)

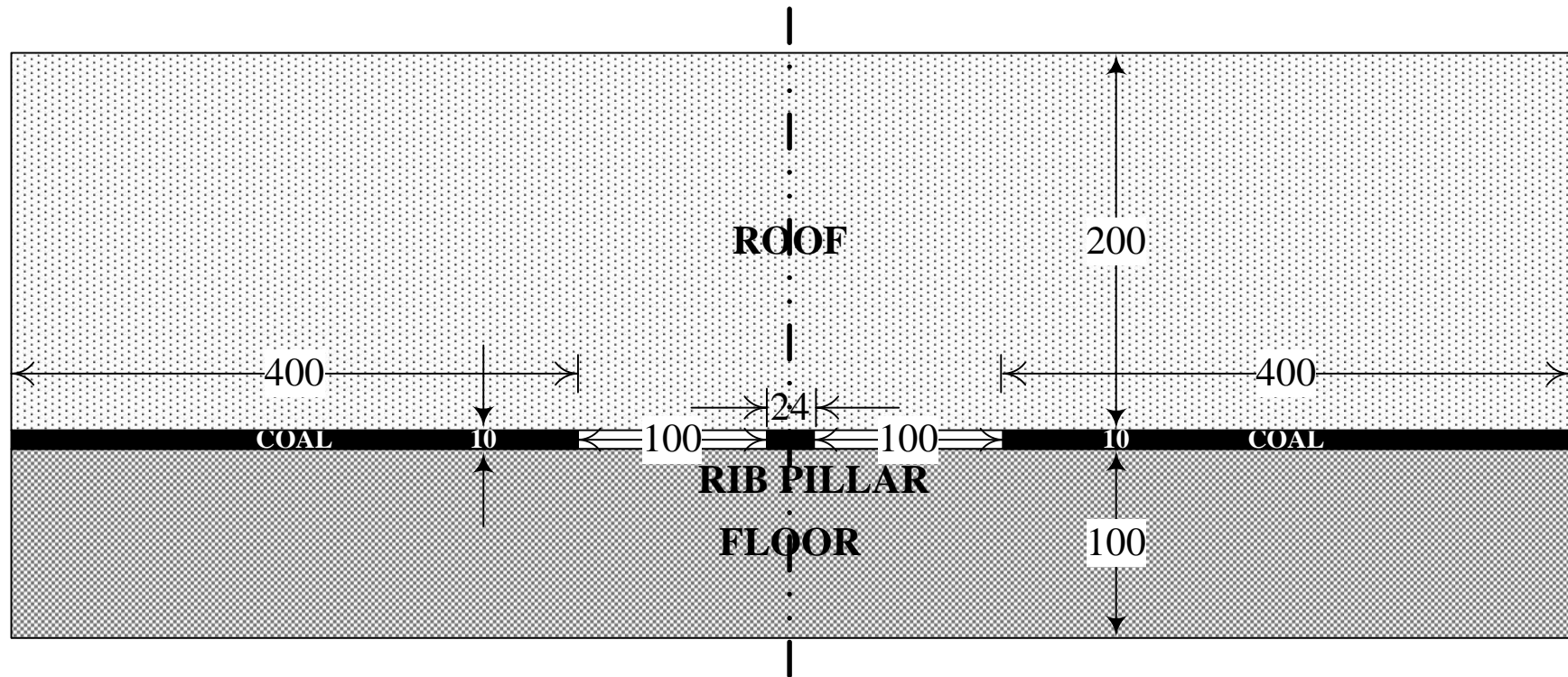
Observation table

Table 1: Pillar height = -----

Distance from middle section of the pillar	σ_1 (1)	σ_2 or σ_3 (2)
0		
12m / 24m		

Topics Covered

- Generate FEM mesh
- Groups
- Stages
- Boundary conditions
- Material properties
- Post processing



INSTRUCTION SHEET

2D- FEM modelling of stress around an excavation using Strand7

FEM Software:

Steps

- Make a proper sketch of the model showing symmetry if any and the boundary conditions. Write down coordinates which define the geometry. Divide geometry into appropriate blocks (plates). Draw the FE mesh by hand. Note down number of sub-divisions in each plate (block).
- Generate **coarse FEM mesh** by extrusion:
 - a) create node 0,0
 - b) extrude the node 1 by increment x direction to form beams
 - c) Extrude beams in y direction to form plates for floor, coal seam and roof.
- Use **subdivide** and **sub grade** options of the menu **tool** proper grading of the mesh i.e. finer mesh in the region of interest and coarser away from the region of interest
- Simulation of mining of the seam in stages of 3 m using tool **global group**
 - a) Open tool **global group**
 - a) Select a new group and rename it slice-1
 - b) Select bottom 3 m slice in the panel
 - c) Assign selected slice to the selected group
 - d) Repeat b, c and d till groups are made of all slices.
 - b) After formation of the group open tool **global stage**
 - a) Select a new group and rename it virgin
 - b) select a new sub-group of virgin and rename it **pillar height=3m and click slice-1**
 - c) select a new sub-group of virgin again and rename it **pillar height=6m and click slice-1 and slice -2**
 - d) Repeat c till stages are defined for all slices.
- Thus, generate 1m x 1m mesh size in pillar and larger size elements in the region away from the pillar. Assign boundary conditions.
- Give material properties for coal seam and sandstone
- For solving the problem, select nonlinear static solver.
 - a) Click **load increment**
 - b) Select **staged analysis**
 - c) **Solve**
- Open the **result file**. In **contour** option uncheck **interpolation to nodes**.
- Check the correctness of the stress analysis result: check deformation and stresses at various places and judge their correctness.
- Note down the principal stresses in middle section of the pillar.

NM – 4b: Modelling of sequence of excavation and design of stopes/cavern

Aim: Perform FE analysis of a 24m wide rib pillars at a depth of 200m between two horizontal panels of 100m wide each. Study effect of height of the pillar on its strength for following height of extraction: 3m, 6m, 12, 18m, 24m.

Property Set-1

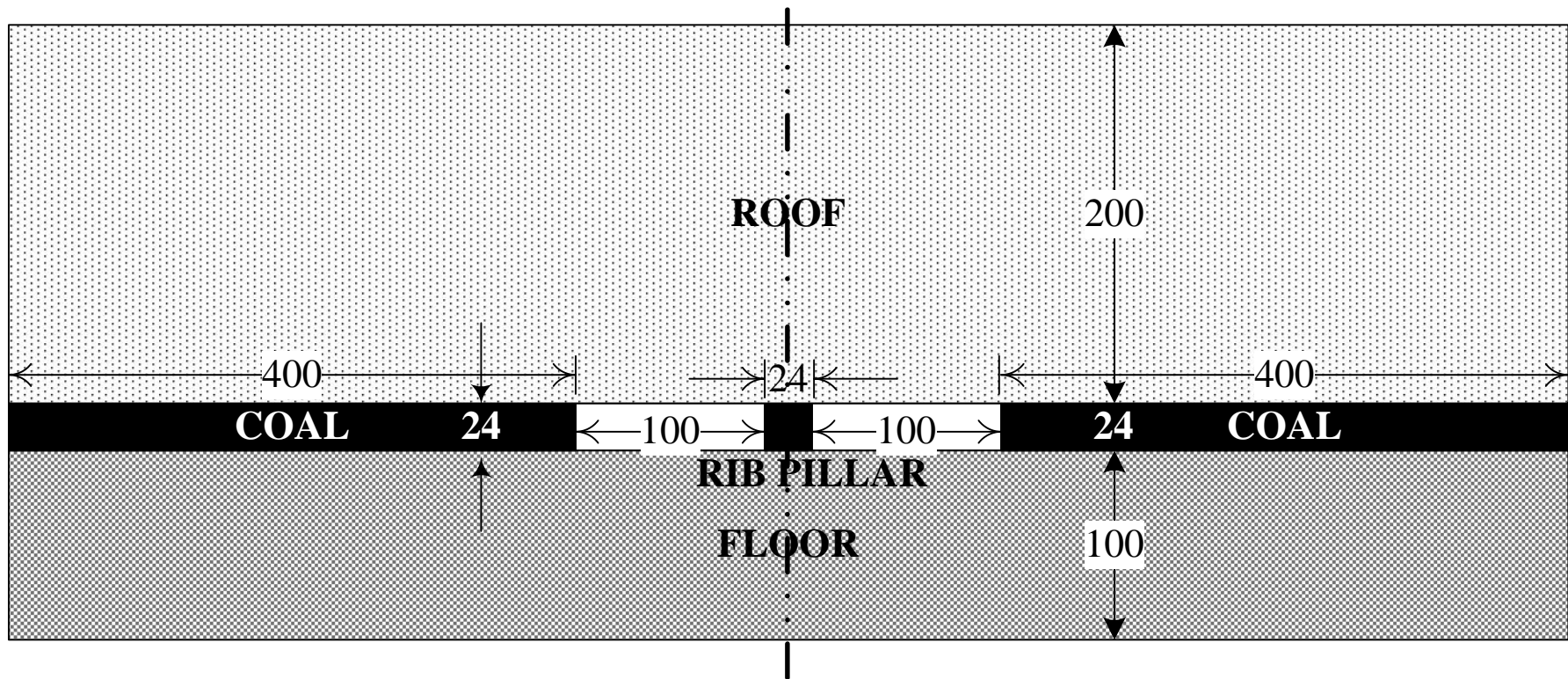
Pillar rock

Modulus of elasticity, E	= 2 GPa
Poisson' ratio	= 0.15
Unit Weight, ρ	= 1400 kg/m ³

Roof and floor rock

Modulus of elasticity, E	= 10 GPa
Poisson' ratio	= 0.25
Unit Weight, ρ	= 2200 kg/m ³

In-situ Stress: gravity load: (9.8 m/sec² acceleration)



Observation table

Table 1: Pillar height = 3m

Distance from Centre of the pillar	σ_1 (1)	σ_2 or σ_3 (2)
0		
12m		

Table 2: Pillar height = 6m

Distance from Centre of the pillar	σ_1 (1)	σ_2 or σ_3 (2)
0		
12m		

Table 3: Pillar height = 12m

Distance from Centre of the pillar	σ_1 (1)	σ_2 or σ_3 (2)
0		
12m		

Table 4: Pillar height = 18m

Distance from Centre of the pillar	σ_1 (1)	σ_2 or σ_3 (2)
0		
12m		

Table 5: Pillar height = 24m

Distance from Centre of the pillar	σ_1 (1)	σ_2 or σ_3 (2)
0		
12m		

Topics Covered

- Generate FEM mesh
- Groups
- Stages
- Boundary conditions
- Material properties
- Post processing

INSTRUCTION SHEET

FEM Software:

Steps

- Make a proper sketch of the model showing symmetry if any and the boundary conditions. Write down coordinates which define the geometry. Divide geometry into appropriate blocks (plates). Draw the FE mesh by hand. Note down number of sub-divisions in each plate (block).
- Generate **coarse FEM mesh** by extrusion:
 - d) create node 0,0
 - e) extrude the node 1 by increment x direction to form beams
 - f) Extrude beams in y direction to form plates for floor, coal seam and roof.
- Use **subdivide** and **sub grade** options of the menu **tool** proper grading of the mesh i.e. finer mesh in the region of interest and coarser away from the region of interest
- Simulation of mining of the seam in stages of 3 m using tool **global group**
 - a) Open tool **global group**
 - e) Select a new group and rename it slice-1
 - f) Select bottom 3 m slice in the panel
 - g) Assign selected slice to the selected group
 - h) Repeat b, c and d till groups are made of all slices.
 - b) After formation of the group open tool **global stage**
 - a) Select a new group and rename it virgin
 - b) select a new sub-group of virgin and rename it **pillar height=3m and click slice-1**
 - c) select a new sub-group of virgin again and rename it **pillar height=6m and click slice-1 and slice -2**
 - d) Repeat c till stages are defined for all slices.
- Thus, generate 1m x 1m mesh size in pillar and larger size elements in the region away from the pillar. Assign boundary conditions.
- Give material properties for coal seam and sandstone
- For solving the problem, select nonlinear static solver.
 - a) Click **load increment**
 - b) Select **staged analysis**
 - c) **Solve**
- Open the **result file**. In **contour** option uncheck **interpolation to nodes**.
- Check the correctness of the stress analysis result: check deformation and stresses at various places and judge their correctness.
- Note down the principal stresses in middle section of the pillar formed by mining height of 3, 6, 12, 18 and 24m in stages.

NM – 5: Modelling of mechanical behaviour of pillars under different geo-mining conditions

Aim: Perform FE analysis of a 24m wide rib pillars at a depth of 200m between two horizontal panels of 100m wide each. Study effect of height of the pillar on its strength for following height of extraction: 3m, 6m, 12, 18m, 24m for the property set -1 and set-2 separately.

Property Set-1

Pillar rock

Uniaxial compressive strength, $C_0 = 5 \text{ MPa}$

Coefficient of friction, $\Phi = 30^\circ$

(Coulomb failure criteria)

Modulus of elasticity, $E = 2 \text{ GPa}$

Poisson's ratio $= 0.15$

Unit Weight, $\rho = 1400 \text{ kg/m}^3$

Roof and floor rock

Uniaxial compressive strength, $C_0 = 10 \text{ MPa}$

Coefficient of friction $\Phi = 35^\circ$

Modulus of elasticity, $E = 10 \text{ GPa}$

Poisson's ratio $= 0.25$

Unit Weight, $\rho = 2200 \text{ kg/m}^3$

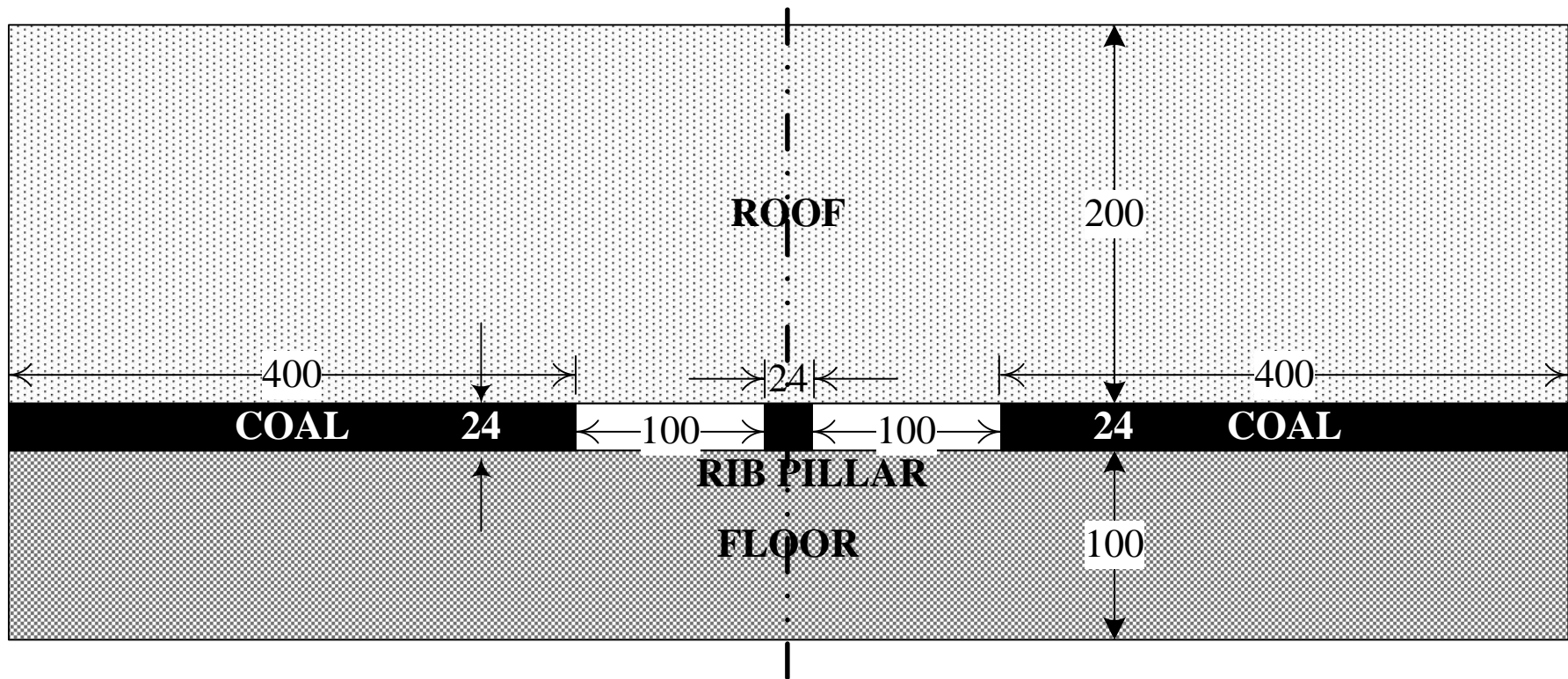
Property Set-2:

Property Set-1 with 2m thick soft clay of $E = 0.2 \text{ GPa}$, $\nu = 0.25$ $\rho = 1700 \text{ kg/m}^3$ in the floor below the ore body.

In-situ Stress: gravity load: (9.8 m/sec^2 acceleration)

Topics Covered

- Generate FEM mesh
- Groups
- Stages
- Boundary conditions
- Material properties
- Post processing



Observation table**For property set -1**

Table 1: Pillar height = 3m

Distance from Centre of the pillar	σ_1 (1)	σ_2 or σ_3 (2)	Strength (3)	Safety Factor (Strength / σ_1)
0				
12m				
		Weighted SF		

Table 2: Pillar height = 6m

Distance from Centre of the pillar	σ_1 (1)	σ_2 or σ_3 (2)	Strength (3)	Safety Factor (Strength / σ_1)
0				
12m				
		Weighted SF		

Table 3: Pillar height = 12m

Distance from Centre of the pillar	σ_1 (1)	σ_2 or σ_3 (2)	Strength (3)	Safety Factor (Strength / σ_1)
0				
12m				
		Weighted SF		

Table 4: Pillar height = 18m

Distance from Centre of the pillar	σ_1 (1)	σ_2 or σ_3 (2)	Strength (3)	Safety Factor (Strength / σ_1)
0				
12m				
		Weighted SF		

Table 5: Pillar height = 24m

Distance from Centre of the pillar	σ_1 (1)	σ_2 or σ_3 (2)	Strength (3)	Safety Factor (Strength / σ_1)
0				
12m				
		Weighted SF		

For property set -1 and property set-2

Table 1: Pillar height = 3m

Distance from Centre of the pillar	σ_1 (1)	σ_2 or σ_3 (2)	Strength (3)	Safety Factor (Strength / σ_1)
0				
12m				
		Weighted SF		

Table 2: Pillar height = 6m

Distance from Centre of the pillar	σ_1 (1)	σ_2 or σ_3 (2)	Strength (3)	Safety Factor (Strength / σ_1)
0				
12m				
		Weighted SF		

Table 3: Pillar height = 12m

Distance from Centre of the pillar	σ_1 (1)	σ_2 or σ_3 (2)	Strength (3)	Safety Factor (Strength / σ_1)
0				
12m				
		Weighted SF		

Table 4: Pillar height = 18m

Distance from Centre of the pillar	σ_1 (1)	σ_2 or σ_3 (2)	Strength (3)	Safety Factor (Strength / σ_1)
0				
12m				
		Weighted SF		

Table 5: Pillar height = 24m

Distance from Centre of the pillar	σ_1 (1)	σ_2 or σ_3 (2)	Strength (3)	Safety Factor (Strength / σ_1)
0				
12m				
		Weighted SF		

Mohr-Coulomb shear failure criterion:

Strength = $C_0 + q\sigma_3$; Safety Factor = strength/ σ_1
where,

$$q = \tan^2 \left(\frac{f}{4} + \frac{w}{2} \right)$$

σ_1 and σ_3 are major and minor principal stresses in the pillar.

INSTRUCTION SHEET

FEM Software:

Steps

- Make a proper sketch of the model showing symmetry if any and the boundary conditions. Write down coordinates which define the geometry. Divide geometry into appropriate blocks (plates). Draw the FE mesh by hand. Note down number of sub-divisions in each plate (block).
- Generate **coarse FEM mesh** by extrusion:
 - g) create node 0,0
 - h) extrude the node 1 by increment x direction to form beams
 - i) Extrude beams in y direction to form plates for floor, coal seam and roof.
- Use **subdivide** and **sub grade** options of the menu **tool** proper grading of the mesh i.e. finer mesh in the region of interest and coarser away from the region of interest
- Simulation of mining of the seam in stages of 3 m using tool **global group**
 - a) Open tool **global group**
 - i) Select a new group and rename it slice-1
 - j) Select bottom 3 m slice in the panel
 - k) Assign selected slice to the selected group
 - l) Repeat b, c and d till groups are made of all slices.
 - b) After formation of the group open tool **global stage**
 - a) Select a new group and rename it virgin
 - b) select a new sub-group of virgin and rename it **pillar height=3m and click slice-1**
 - c) select a new sub-group of virgin again and rename it **pillar height=6m and click slice-1 and slice -2**
 - d) Repeat c till stages are defined for all slices.
- Thus, generate 1m x 1m mesh size in pillar and larger size elements in the region away from the pillar. Assign boundary conditions.
- Give material properties for coal seam and sandstone
- For solving the problem, select nonlinear static solver.
 - a) Click **load increment**
 - b) Select **staged analysis**
 - c) **Solve**
- Open the **result file**. In **contour** option uncheck **interpolation to nodes**.
- Check the correctness of the stress analysis result: check deformation and stresses at various places and judge their correctness.
- Note down the principal stresses in middle section of the pillar formed by mining height of 3, 6, 12, 18 and 24m in stages.
- Calculate strength and safety factor of the pillar and note down in the observation table.
- Plot graphs of pillar width-height ratio vis-à-vis strength / safety factor.

NM - 6: Modelling of a hydroelectric cavern and gas oil storage cavern

Aim: Using Finite Element stress analysis of the oil storage cavern, and perform its stability analysis & support requirement of the oil storage tunnel

A cross section through a oil storage cavern is shown in Figure 1. Cross sectional dimnsions and excavation sequences of the storage cavern, construction tunnel and water tunnels are shown in Figure 2.

Rock propertires are: density (ρ) = 2650 kg/m³, Young's modulus (E) = 50 GPa, poisson's ratio = 0.22, Uniaxial compressive strength (UCS) = 3.5 MPa and angle of internal friction of rock = 40°.

Insitu horizontal stress along x-axis is 2 times the vertical stress.

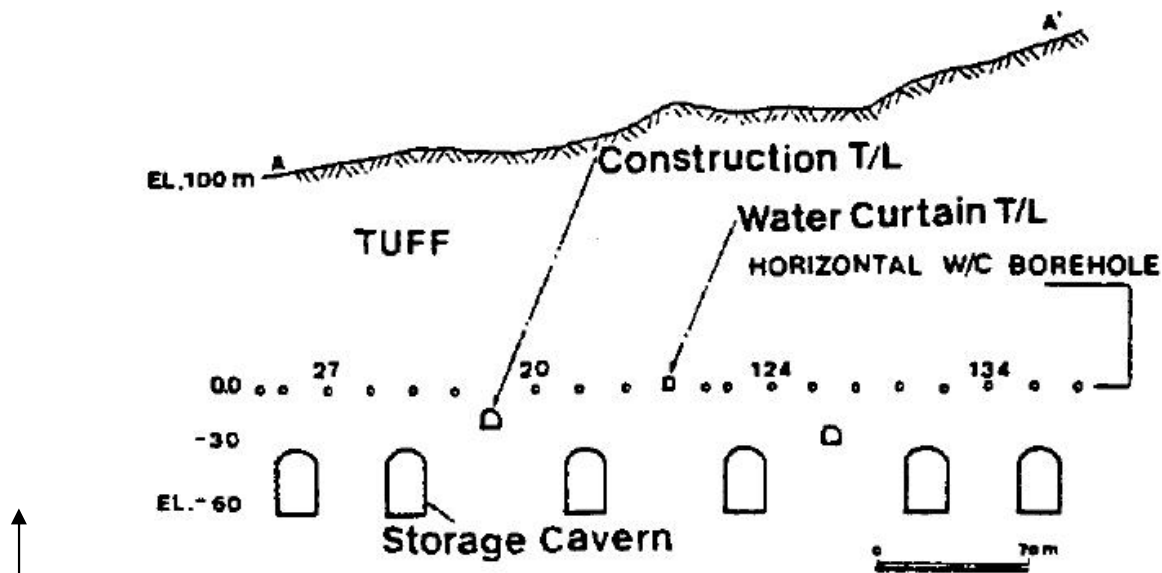


Figure -1: X-section through oil storage caverns

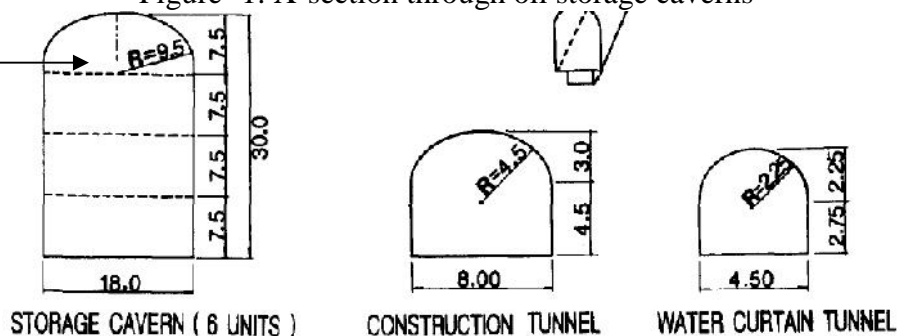


Figure -2: Sequence of excavation of the cavern and tunnel.

Model: Prepare the model using pre-processing modules of the FEM package.

Observation Table

Table 1: σ_{xx} & σ_{yy} at the nodes along x-axis
(size of the elements at the boundary =)

Distance from center of the cavern, m	Insitu results (1)		FEA results (2)		Difference (1 – 2)		Remarks
	σ_{xx}	σ_{yy}	σ_{xx}	σ_{yy}	σ_{xx}	σ_{yy}	

Table 2: σ_{xx} & σ_{yy} at the nodes along y-axis
(size of the elements at the boundary =)

Distance from center of the cavern, m	Insitu results (1)		FEA results (2)		Difference (1 – 2)		Remarks
	σ_{xx}	σ_{yy}	σ_{xx}	σ_{yy}	σ_{xx}	σ_{yy}	

Topics Covered

- Generate FEM mesh
- Groups
- Stages
- Boundary conditions
- Material properties
- Post processing

INSTRUCTION SHEET

2D- FEM modelling of stress around different caverns / excavations using Strand7

STEPS TO BE FOLLOWED:

13. Click on the icon of Strand7 server mode on the desktop
14. Click on menu **file – new**
15. Screen will be displayed for selecting model unit as displayed below
16. Choose SI unit and change the Modulus/ Stress to MPA, Force to kN, let the other units as it is. Click on OK button

CREATING FEM MESH

31. Go to the menu **Create – node**. The first node will have coordinate (0,0,0) in (X, Y, Z). Press apply button. The second node will have the coordinate (18, 0, 0). Again press apply button. Similarly, create nodes at (0, 22.5, 0), (18, 22.5, 0), (9, 30, 0).
32. Press **F3** button in keyboard to view all nodes.
33. Go to **Tools > Points and Lines**. Select Find Circle Centre. Select the top 3 nodes using cursor as 1, 2, 3 points. Click Apply. Circle centre is obtained.
34. Select Two Point Circle in the dialog box. Using cursor, Select circle centre as P1 and the two points on each side of the circle center as P2 and P3. Select Create Beams. Enter 30 in Steps. Click Apply. An arc is created.
35. Go to the menu **Create – element**. A dialog box appears at the left side of screen. Choose the **Beam element** from the box. Click on the first node. A rubber band will appear. With the rubber band move the mouse to the second node and select by clicking the left button of the mouse. The rubber band will vanish. Finally, close the dialog box. Geometry of cavern/excavation is created.
36. Select the whole tunnel. Go to **Tools > Copy > by Increment**. A dialog box appears. Enter (18+31.5, 0, 0) in (x, y, z). Click Apply. Press F3. Select second tunnel. Enter (18+63, 0, 0) in (x, y, z). Click Apply. Repeat the above for all other cavern/excavation. Press F3 button in keyboard to view all cavern/excavation.
37. Create the domain of the problem
38. Select 'Toggle Beam Select'. Select All. Go to **Tools > Geometry Tools> Face from Beam Polygon**. Enter 5 in 'Edge Tol'. Click Apply. Close the dialog box. Right click on **(hide entity)** in vertical toolbar on the left side window. A dialog box appears. Select 'Show Wireframes'. Click Apply. Repeat the above for all individual cavern/excavation and for external boundary also.
39. Go to **Tools > Clean > Geometry**. Select 'Leave One Face'. Click Apply.
40. Go to **Global > Groups**. A dialog box appears. Create group under 'Model' using 'New'. Rename them as cavern 1, cavern -2 etc.
41. Go to **Global > Stages**. A dialog box appears. Create 2 stages using 'New'. Rename them as Virgin, cavern -1, and cavern -2 etc.
42. **Meshing:** Go to **Attributes > Vertex > Mesh Size**. Enter 1 in 'Value'. Click Apply. Go to **Tools > Automeshing> Surface Mesh**. A dialog box appears. Enter 10 in 'Maximum Edge Length'. Click Mesh.
43. Go to menu **Tool – align – plate axis** and select all elements to be aligned with the global axis.

DEFINING LOAD CASES

4. Go to menu **Global – load and Freedom cases**. Select freedom cases and check **2D Plane** in auto set. Select **Primary load cases** and **edit case name** to **Load case1: Gravity**. Select new load case and edit it to **Load case 2:Sh = 2×Sv**
5. Go to menu **Attribute – plate – pre- (load) stress** and apply pre-stresses separately. (You have to select the appropriate load cases).
6. Select 'Toggle Plate Select'. Select All. Go to **Tools > Align > Plate Axes**. Select 'x' and 'X'. Click Apply.

DEFINING BOUDBARY CONDITIONS

3. Go to the menu **attribute - node - restrain** and fix the boundary of the model by restricting the translation either along the X, and/or Y and/or Z direction depending on the selection as per boundary conditions. For example, select the nodes of the lower horizontal boundary, select **Fix** from **Node Attribute** menu and press **apply**.

DEFINE MATERIAL PROPERTY

2. Now go to the menu **property - plate** and for **plate property 1** select **2-D plain strain** condition and enter the following properties of plate
Select the material type as **Isotropic** and enter: Modulus = 50×10^3 MPa,
Density = 2650 kg/m^3 , Poisson's ratio = 0.22
Edit the name of the plate.

SOLVE

4. Now go to menu **solver – nonlinear static** and select **calculate – plate stress**
 - a. Click **load increment**
 - b. Select **staged analysis**
 - c. **Solve**
5. Open the **result file**. In **contour** option uncheck **interpolation to nodes**.
6. Check the correctness of the stress analysis result: check deformation and stresses at various places and judge their correctness.

Some Viva Questions

1. What is discretization?
2. Steps in FEM
3. Theories of failure.
4. What is meant by plane stress?
5. What is meant by plane strain?
6. What is factor of safety?
7. What are isotropic and orthotropic materials?
8. What is the difference between static analysis and dynamic analysis?
9. What are Global coordinates?
10. What are natural coordinates?
11. What is a CST element?
12. Define shape function.
13. What are the characteristics of shape function?
14. Why polynomials are generally used as shape function?
15. State the properties of a stiffness matrix.
16. What is a sub parametric element?
17. What is a super parametric element?
18. What is meant by isoparametric element?
19. What is the purpose of isoparametric element?