|  |  |
| --- | --- |
| **EX.NO:01** | **STRESS ANALYSIS OF A CANTILEVER BEAM WITH POINT LOAD** |
| **DATE:** |

Analyze the given Cantilever beam using ANSYS software

F

b

b

L

E = 210000 N/mm2

L = 100 mm

b = 10 mm

F = 100 N

v = 0.3

Note: (Refer PSG Design data page no.6.1)

Izz = Ixx + Iyy

= b4/12+b4/12 =b4/6 =104/6 =1666.667 mm4

A = b2  = 102  = 100 mm2

Where, E – Modulus of Elasticity

V– Poisson Ratio

Izz– Polar moment of Inertia

A – Ar

**AIM:**

To analyze the given cantilever beam using ANSYS software.

**SOFTWARE USED:**

ANSYS APDL 19

**PROCEDURE:**

**# PREPROCESSOR**

**Step 1:**

Main menu > Preferences

Click on structural and give ok.

**Step 2:**

1. Main Menu > Preprocessor > Element Type> Add/Edit/Delete
2. Add an element type.
3. Structural family of beam elements.
4. Choose 2D elastic 3 ( beam3 )
5. OK to apply the element type and close the dialog box

**Step 3:**

1. Main Menu> Preprocessor> Real Constants> Add/Edit/Delete
2. Enter 100 for area
3. Enter 1666.667 for Izz
4. Enter 10 for height and give OK

**Step 4:**

1. Main Menu> Preprocessor> Material Props> Material Models
2. Double-click on Structural, Linear, Elastic, And Isotropic.
3. Enter 210000 for EX
4. Enter 0.3 for PRXY
5. OK to define material property set and close the dialog box.
6. Material> Exit

Tool bar>save\_DB

**Step 5:**

Main Menu> Preprocessor> Modeling> create>Key point>In active CS

Create Key point 1 at 0,0,0 then enter Apply and create key point 2 at 100,0,0 then enter ok.

**Step 6:**

Main Menu> Modeling> create> line> line> St Line

Create line through key point 1, 2

**Step 7:**

Main Menu> Preprocessor> Meshing>Mesh tool

Mesh tool window will appear, in that make a tick mark in the small box (Smart size) and

Made the cursor into 1 (fine size) and give, mesh. Pick all for the area to be meshed.

**# SOLUTION**

**Step 8:**

Main Menu> Solution> Define Loads> Apply> Structural>Displacement> on keypoints

Select key point 1 and fix (all DOF Constrained)

**Step 9:**

Main Menu> Solution> Define Loads> Apply> Structural> Force/Moment> on keypoints

Select direction of Force/Moment FY

Force/Moment Value -100 then give ok and save\_DB

Main Menu> Solution> Solve> Current LS –ok

**# POST PROCESSOR**

**Step 10:**

1. Main Menu> General Postproc> Plot Results> Deformed Shape

2. Choose Def + Undeformed.

3. OK.

**RESULT:**

Thus the stress analysis on cantilever beam with point load was done using ANSYS Software.

|  |  |
| --- | --- |
| **EX.NO:02** | **STRESS ANALYSIS OF A FIXED BEAM WITH POINT LOAD** |
| **DATE:** |

Analyze the given Cantilever beam using ANSYS software.

F

b

b

6M 6M

E = 2.1e11 N/m2

b = 1m

F = 1000 N

v = 0.3

Note: (Refer PSG Design data page no.6.1)

Izz = Ixx + Iyy

= b4/12+b4/12 =b4/6 =0.14/6 =0.1667 m4

A = b2 = 12 = 1m2

Where, E – Modulus of Elasticity

V – Poisson Ratio

Izz – Polar moment of Inertia

A – Area

**AIM:**

To analyze the given fixed beam with point load using ANSYS Software.

**SOFTWARE USED:**

ANSYS APDL 19

**PROCEDURE:**

**# PREPROCESSOR**

**Step 1:**

Main menu > Preferences

Click on structural and give OK

**Step 2:**

Main Menu > Preprocessor > Element Type> Add/Edit/Delete Add an element type. Structural family of beam elements. Choose 2D elastic 3 ( beam3 ) OK to apply the element type and close the dialog box

**Step 3:**

Main Menu> Preprocessor> Real Constants> Add/Edit/Delete Enter 100 for area Enter 0.1667 for Izz Enter 1 for height and give OK

**Step 4:**

Main Menu> Preprocessor> Material Props> Material Models Double-click on Structural, Linear, Elastic, And Isotropic. Enter 2.1e11 for EX Enter 0.3 for PRXY OK to define material property set and close the dialog box. Material> Exit Tool bar> save\_DB

**Step 5:**

Main Menu> Preprocessor> Modeling> create> Key point> in active CS

Create Key point 1 at 0,0,0 then enter Apply and create key point 2 at 6,0,0 then enter Apply

And key point 3 at 12,0,0 then enter ok.

**Step 6:**

Main Menu> Modeling> create> line> line> St Line

Create line through key point 1 to 2 and then key point 2 to 3 then enter ok.

**Step 7:**

Main Menu> Preprocessor> Meshing> Mesh tool *Mesh tool window will appear, in that make a tick mark in the small box (Smart size) and Made the cursor into 1 (fine size) and give, mesh. Pick all for the area to be meshed.*

**# SOLUTION**

**Step 8:**

Main Menu> Solution> Define Loads> Apply> Structural>Displacement> on key points

Select key point 1 and key point 3 fix ( all DOF Constrained )

**Step 9:**

Main Menu> Solution> Define Loads> Apply> Structural> Force/Moment> on key points

Select key point 2

Select direction of Force/Moment FY

Force/Moment Value -1000 then give ok and save\_DB

Main Menu> Solution> Solve> Current LS –ok

**# POST PROCESSOR**

**Step 10:**

1. Main Menu> General Postproc> Plot Results> Deformed Shape

2. Choose Def + undeformed.

3. OK.

**RESULT:**

Thus the stress analysis on fixed beam with point load was done using ANSYS Software.

|  |  |
| --- | --- |
| **EX.NO:03** | **STRESS ANALYSIS OF A PLATE WITH HOLE** |
| **DATE:** |

Consider the square plate of uniform thickness with a circular hole with dimensions shown in fig. the thickness of the plate is 1mm. The Young’s Modulus E=1x107 N/mm2 and Poisson ratio is 0.3. A uniform pressure P=1 N/mm2 acts on the boundary of the hole. By assuming plane stress conditions, determine the stress and displacement field using ANSYS.

R7000

20000

20000

NOTE: Instead of doing the analysis for the whole body, it is enough to analyze the quarter portion. Because the plate is symmetric about both the axis.

10000

R7000 3000

20000 10000

R7000

20000

**AIM:**

To analyze the given plate of uniform thickness and circular hole by using ANSYS software.

**SOFTWARE USED:**

ANSYS 10

**PROCEDURE:**

**#PREPROCESSOR**

**Step 1:**

Main menu> Preferences

Click on structural and give OK.

**` Step 2:**

1. Main Menu> Preprocessor> Element Type> Add/Edit/Delete
2. Add an element type.
3. Structural solid family of elements.
4. Choose Quad 4 node 42
5. OK to apply the element type and close the dialog box.

**Step 3:**

1. Main Menu> Preprocessor> Material Props> Material Models
2. Double-click on Structural, Linear, Elastic And Isotropic.
3. Enter 1e7 for EX.
4. Enter 0.3 for PRXY.
5. OK to define material property set and close the dialog box.
6. Material> Exit

Tool bar>save\_DB

**Step 4:**

1. Main Menu> Preprocessor> Modeling> Create> Areas> Rectangle>

By Dimensions enter the following:

X1 = 0 (Note: Press the Tab key between entries)

X2 = 10e3

Y1 = 0

Y2 = 10e3

1. Main Menu> Preprocessor> Modeling> Create> Areas> circle> By Dimensions

Outer radius = 7e+3

Save\_DB

**Step 5:**

1. Main Menu> Preprocessor> Modeling> Operate> Booleans> subtract> Areas
2. Pick the rectangular area give ok and then pick the circular area and give ok.

Toolbar: SAVE\_DB

**Step 6:**

Main menu> Preprocessor> Meshing>Mesh tool

Mesh tool window will appear, in that make a tick mark in the small box (Smart size) and

Make the cursor into 1 (fine size) and give, mesh.

Pick all for the area to be meshed.

**Step 7:**

1. Utility Menu> File> Save ad
2. Enter mesh.db for database file name.
3. OK to save file and close dialog box.

**# SOLUTION**

**Step 8:**

Main Menu> Solution> Define Loads> Apply> Structural> Displacement>symmetry

B.C> On Lines. Select the straight lines corresponding to the left and bottom edges

(which are the line symmetry for this problem then click OK)

**Step 9:**

Main Menu> Solution> Define Loads> Apply> Structural> pressure>on Lines> Value

Load PRES value> enter 1> Select the circular arc and click ok.

**Step 10:**

Main Menu> Solution> Solve> current LS ok

**Step 11:**

1. Main Menu> General Postproc> Plot Results> Deformed Shape
2. Choose Def + undeformed.
3. OK

**# POST PROCESSOR**

**Step 12:**

Plot the Von Mises equivalent stress.

1. Main Menu> general Postproc> Plot Results> Contour Plot> Nodal solution
2. Choose Stress item to be contoured.

Scroll down and choose Von Mises( SEQU )

OK.

1. Utility Menu> Plot Ctrls> Animate> Deformed Results
2. Choose Stress item to be contoured.

Scroll down and choose Von Mises( SEQU )

OK.

1. Make choices in the Animation Controller (not shown), if necessary, then choose close.

**Step 13:**

1. Main Menu> General Postprocessor> List Results> Reaction Solution
2. OK to list all items and close the dialog box.
3. Scroll down and find the total vertical force, FY.

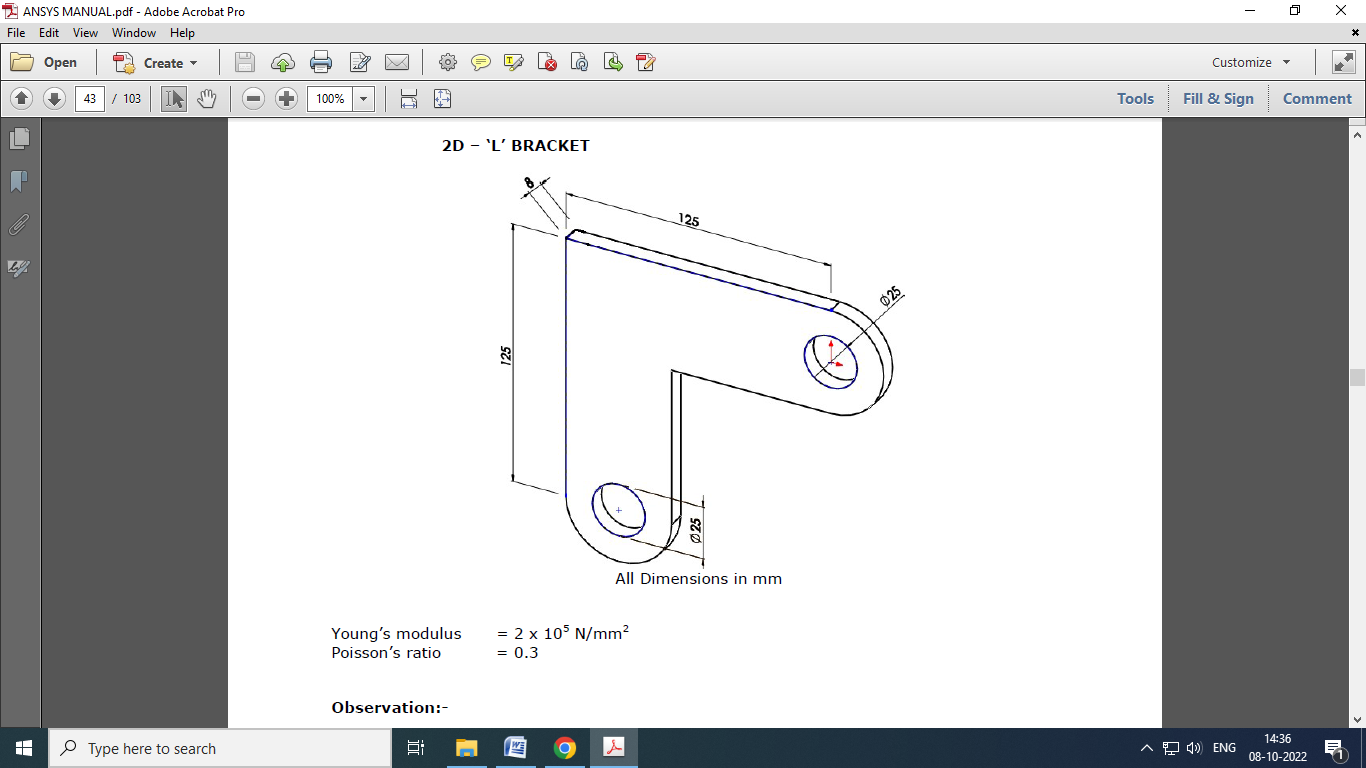
File> Close (Windows), or Close (X11/Motif), to close the window.

**RESULT:**

Thus the given Rectangular plate of uniform thickness with circular hole was statically analyzed and stress distribution was found and plotted.

|  |  |
| --- | --- |
| **EX.NO:04** | **STRESS ANALYSIS OF RECTANGULAR L-BRACKET** |
| **DATE:** |

Consider the corner angle bracket the upper left-hand pin hole is constrained (welded) around its entire circumference and a tapered pressure load is applied to the bottom of the lower tightened pin hole analyze the stresses on the given rectangular L-Bracket using ANSYS Software.



**AIM:**

To analyze the stresses on the given rectangular L-Bracket using ANSYS Software.

**SOFTWARE USED:**

ANSYS 19

**PROCEDURE:**

**#PREPROCESSOR**

**Step 1:**

Main Menu> Preprocessor> Modeling> Create> Areas> Rectangle>By Dimensions Enter the following: X1 = 0 (Note: Press the Tab key between entries) Apply to create the first rectangle.

X2 = 6

Y1 = -1

Y2 = 1

Enter the following:

X1 = 4

X2 = 6

Y1 = -1

Y2 = -3

OK to create the second rectangle and close the dialog box.

**Step 2:** **Change plot controls and re plot**

1. Utility Menu> Plot Ctrls> Numbering
2. Turn on area numbers.
3. OK to change controls, close the dialog box, and re plot.

**Step 3:** **Change working plane to polar and create first circle**

1. Utility Menu>Plot Ctrls> Pan, Zoom, Rotate
2. Click on small dot once to zoom out.
3. Close dialog box.
4. Utility Menu>Work plane> Display Working Plane ( toggle on )
5. Utility Menu>Work plane> WP Settings
6. Click on Polar.
7. Click on Grid and Triad.
8. Enter 0.1 for snap increment.
9. OK to define settings and close the dialog box.
10. Main Menu> Preprocessor> Modeling> Create> Areas> Circle> Solid Circle be sure to read prompt before picking
11. Pick center point at:

WP X = 0

WP Y = 0

1. Move mouse to radius of 1 and click left button to create circle.
2. OK to close picking menu.
3. Toolbar: SAVE\_DB.

**Step 4: Move working plane and create second circle**

1. Utility Menu> Work Plane>Offset WP to>Key points
2. Pick key point at lower left corner of rectangle.
3. Pick key point at lower right of rectangle.
4. OK to close picking menu.
5. Main Menu> Preprocessor> Modeling> Create> Areas> Circle> Solid Circle
6. Pick center point at: WP X = 0, WP Y = 0, Move mouse to radius of 1 and click left button to create circle
7. OK to close picking menu.
8. Toolbar: SAVE\_DB.

**Step 5:** **Add areas.**

1. Main Menu> Preprocessor> Modeling> Operate> Booleans> Add> Areas
2. Pick all for all areas to be added.
3. Toolbar: SAVE\_DB

**Step 6:** **Create line fillet.**

1. Utility Menu>Plot Ctrls> Numbering
2. Turn on line numbering.
3. OK to change controls, close the dialog box, and automatically replot.
4. Utility Menu>Work Plane> Display Working Plane ( toggle off )
5. Main Menu> Preprocessor> Modeling> Create> Lines> Line fillet
6. Pick lines 17 and 8.
7. OK to finish picking lines ( in picking menu ).
8. Enter 0.4 as the radius.
9. OK to create line fillet and close the dialog box.
10. Utility Menu> Plot> Lines

**Step 7:** **Create fillet area.**

1. Utility Menu> Plot Ctrls> Pan, Zoom, Rotate
2. Click on Zoom button.
3. Move mouse to fillet region, click left button, move mouse out and click again.
4. Main Menu> Preprocessor> Modeling> create> Areas> Arbitrary> By Lines
5. Pick lines 4, 5 and 1.
6. OK to create area and close the picking menu.
7. Utility Menu> Plot Ctrls> Pan, Zoom, Rotate
8. Click on Fit button.
9. Close the Pan, Zoom, Rotate dialog box.
10. Utility Menu> Plot> Areas
11. Toolbar: SAVE\_DB.

**Step 8:** **Add areas together**.

1. Main Menu> Preprocessor> Modeling> Operate> Booleans> Add> Areas
2. Pick all for all areas to be added.
3. Toolbar: SAVE\_DB

**Step 9: Create first pin hole.**

1. Utility Menu>Work Plane> Display Working Plane ( toggle on )
2. Main Menu> Preprocessor> Modeling> Create> Areas> Circle> Solid Circle
3. Pick center point at:

WP X = 0 (in Graphics Window)

WP Y = 0

1. Move mouse to radius of 0.4 and click left button to create circle.
2. OK to close picking menu.

**Step 10: Move working plane and create second pin hole.**

1. Utility Menu>Work Plane> Offset WP to> Global Origin
2. Main Menu> Preprocessor> Modeling> Create> Areas> Circle> Solid Circle
3. Pick center point at: WP X = 0 (in Graphics Window), WP Y = 0, Move mouse to radius of 0.4 and click left mouse button to create circle.
4. OK to close picking menu.
5. Utility Menu>Work Plane> Display Working Plane ( toggle off )
6. Utility Menu> Plot>Re plot
7. Utility Menu> Plot> Lines
8. Toolbar: SAVE\_DB

**Step 11: Subtract pin holes from bracket.**

1. Main Menu> Preprocessor> Modeling> Operate> Booleans> Subtract> Areas
2. Pick bracket as base area from which to subtract.
3. Apply ( in picking menu )
4. Pick both pin holes as areas to be subtracted.
5. OK to subtract holes and close picking menu.

**Step 12: Save the database as model.db.**

1. Utility Menu> File> Save As
2. Enter model.db for the database file name.
3. OK to save and close dialog box.

**Step 13: Set preferences**

1. Main Menu> Preferences
2. Turn on structural filtering.
3. OK to apply filtering and close the dialog box.

**Step 14: Define material properties.**

1. Main Menu> Preprocessor> Material Props> Material Models
2. Double-click on Structural, Linear, Elastic, And Isotropic.
3. Enter 30e6 for EX
4. Enter 0.27 for PRXY.
5. OK to define material property set and close the dialog box.
6. Material> Exit.

**Step 15: Define element types and options.**

1. Main Menu> Preprocessor> Element Type> Add/Edit/Delete
2. Add an element type.
3. Structural solid family of elements.
4. Choose the 8-node quad (PLANE82).
5. OK to apply the element type and close the dialog box.
6. Options for PLANE82 are to be defined.
7. Choose plane stress with thickness option for element behavior.
8. OK to specify options and close the options dialog box.
9. Close the element type dialog box.

**Step 16: Define real constants.**

1. Main Menu> Preprocessor> Real Constants> Add/Edit/Delete.
2. Add a real constant set.
3. OK for PLANE82.
4. Enter 0.5 for THK.
5. OK to define the real constant and close the dialog box.
6. Close the real constant dialog box.

**Step 17: Mesh the area.**

1. Main Menu> Preprocessor> Meshing> Mesh Tool
2. Set Global Size control.
3. Type in 0.5.
4. OK.
5. Choose Area Meshing.
6. Click on Mesh.
7. Pick all for the area to be meshed ( in picking menu ). Close any warning messages that appear.
8. Close the Mesh Tool.

**Step 18:**

1. Save the database as mesh.db.
2. Enter mesh.db for database file name. LK to save file and close dialog box

Utility Menu> File> Save as

**# SOLUTION**

**Step 19: Apply Loads-Apply Displacement.**

1. Utility Menu> Plot Lines
2. Main Menu> Solution> Define Loads> Apply> Structural> Displacement> On Lines
3. Pick the four lines around left-hand hole ( Line numbers 10, 9, 11, 12 ).
4. OK ( in picking menu ).
5. Click on ALL DOF.
6. Enter 0 for zero displacement.
7. OK to apply constraints and close dialog box.
8. Toolbar: SAVE\_DB.

**Step 20: Apply Pressure Load.**

1. Main Menu> Solution> Define Loads> Apply> Structural> Pressure> On Lines
2. Pick line defining bottom left part of the circle (line 6 ).
3. Apply.
4. Enter 50 for VALUE.
5. Enter 500 for optional value.
6. Apply.
7. Pick line defining bottom right part of circle ( line 7 )
8. Apply.
9. Enter 500 for VALUE.
10. Enter 50 for optional value.
11. OK
12. Toolbar: SAVE\_DB.

**Step 21: Solve**

1. Main Menu> Solution> Solve> Current LS
2. Review the information in the status window, then choose File> Close Windows, or Close ( X11/Motif ), to close the window.
3. OK to begin the solution. Choose Yes to any Verify messages that appear.
4. Close the information window when solution is done.

**# POST PROCESSOR**

**Step 22: Enter the general postprocessor and read in the results.**

Main Menu> General Postproc> Read Results> First Set

**Step 23: Plot the deformed shape.**

Main Menu> general Postproc> Plot Results> Deformed Shape

Choose Def + undeformed.

OK

**Step 24: Plot the Von Mises equivalent stress.**

Main Menu> general Postproc> Plot Results> Contour Plot> Nodal solution

Choose Stress item to be contoured.

Scroll down and choose Von Mises( SEQU )

OK.

Utility Menu> Plot Ctrls> Animate> Deformed Results

Choose Stress item to be contoured.

Scroll down and choose Von Mises( SEQU )

OK.

Make choices in the Animation Controller (not shown), if necessary, then choose close.

**Step 25: List reaction solution.**

Main Menu> General Postprocessor> List Results> Reaction Solution

OK to list all items and close the dialog box.

Scroll down and find the total vertical force, FY.

File> Close (Windows), or Close (X11/Motif), to close the window.

**RESULT:**

Thus the given Rectangular L-bracket was statically analyzed and stress distribution was found and plotted.

|  |  |
| --- | --- |
| **EX.NO:05** | **MODAL ANALYSIS OF A CANTILEVER BEAM** |
| **DATE:** |

Analyze the frequency of a Cantilever Beam using ANSYS Software

b

b

L

E = 206800e6 N/mm2

L = 1 m

b = 0.1 mm

p = 7830 Kg/m3

v = 0.3

Note: (Refer PSG Design data page no.6.1)

Izz = Ixx + Iyy

= b4/12+b4/12 =b4/6 =0.14/6 =1.667\*10-9 m4

A = b2  = 0.0112  = 0.0001 m2

Where, E – Modulus of Elasticity

V– Poisson Ratio

Izz– Polar moment of Inertia

P - Density of Material

**AIM:**

To analyze the frequency response of a cantilever beam using ANSYS software

**SOFTWARE USED:**

ANSYS 19

**PROCEDURE:**

**# PREPROCESSOR**

**Step 1:**

Main menu > Preferences

Click on structural and give ok.

**Step 2:**

Main Menu > Preprocessor > Element Type> Add/Edit/Delete Add an element type Structural family of beam elements. Choose 2D elastic 3 ( beam3 ) OK to apply the element type and close the dialog box

**Step 3:**

Main Menu> Preprocessor> Real Constants> Add/Edit/Delete Enter 0.0001 for area Enter 1.667e-9 for Izz Enter 0.01 for height and give OK

**Step 4:**

Main Menu> Preprocessor> Material Props> Material Models Double-click on Structural, Linear, Elastic, And Isotropic. Enter 206800e6 for EX Enter 0.3 for PRXY Enter density = 7830 OK to define material property set and close the dialog box Material> Exit Tool bar>save\_DB

**Step 5:**

Main Menu> Preprocessor> Modeling> create>Key point>In active CS Create Key point 1 at 0,0,0 then enter Apply and create key point 2 at 1,0,0 then enter ok.

**Step 6:**

Main Menu> Modeling> create> line> line> St Line Create line through key point 1, 2

**Step 7:**

Main Menu> Preprocessor> Meshing>Mesh tool *Mesh tool window will appear, in that make a tick mark in the small box (Smart size) and*  *Made the cursor into 1 (fine size) and give, mesh. Pick all for the area to be meshed.*

**# SOLUTION**

**Step 8:**

Main Menu> Solution> Define Loads> Apply> Structural>Displacement> on key points

Select key point 1 and fix (all DOF Constrained)

**Step 9:**

1. Solution > Analysis type > New analysis > Modal
2. Solution > Analysis type > Analysis options

A window will appear

Select subspace method and enter 5 in the no of nodes to extract

Check the box beside ‘expand mode shape’ and enter 5 in the no of modes to expand

Click OK

Then a window will appear just give OK

**Step 10:**

Main Menu> Solution> Solve> Current LS –ok

**# POST PROCESSOR**

**Step 11:**

1. General postproc > result summary
2. General posproc > read result > first se
3. Main menu > General postproc > Plot results > Deformed shape
4. Choose Def + un deformed edge OK
5. General postproc > read result > next set
6. Main menu > general postproc > plot results > deformed shape
7. Choose Def + un deformes edg OK

**RESULT:**

Thus the frequency response analysis of a cantilever beam was done using ANSYS Software.

**RESULT SUMMARY**

\*\*\*\*\* INDEX OF DATA SETS ON RESULTS FILE \*\*\*\*\*

SET TIME/FREQ LOAD STEP SUBSTEP CUMULATIVE

1 11.742 1 1 1

2 73.806 1 2 2

3 208.47 1 3 3

4 469.15 1 4 4

|  |  |
| --- | --- |
| **EX.NO:06** | **MODAL ANALYSIS OF A SIMPLY SUPPORTED BEAM** |
| **DATE:** |

Analyze the frequency response of a simply supported Beam using ANSYS Software

b

b

L

E = 206800e6 N/mm2

L = 1 m

b = 0.01 mm

p = 7830 Kg/m3

v = 0.27

Note: (Refer PSG Design data page no.6.1)

Izz = Ixx + Iyy

= b4/12+b4/12 =b4/6 =0.14/6 =1.667\*10-9 m4

A = b2  = 0.0112  = 0.0001 m2

Where, E – Modulus of Elasticity

V– Poisson Ratio

Izz– Polar moment of Inertia

P - Density of Material

**AIM:**

To analyze the frequency response of a simply supported beam using ANSYS software

**SOFTWARE USED:**

ANSYS 19

**PROCEDURE:**

**# PREPROCESSOR**

**Step 1:**

Main menu > Preferences

Click on structural and give ok.

**Step 2:**

Main Menu > Preprocessor > Element Type> Add/Edit/Delete Add an element type. Structural family of beam elements. Choose 2D elastic 3 ( beam3 ) OK to apply the element type and close the dialog box

**Step 3:**

Main Menu> Preprocessor> Real Constants> Add/Edit/Delete Enter 0.0001 for area Enter 1.667e-9 for Izz Enter 0.01 for height and give OK

**Step 4:**

Main Menu> Preprocessor> Material Props> Material Models Double-click on Structural, Linear, Elastic, And Isotropic. Enter 206800e6 for EX Enter 0.27 for PRXY Enter density = 7830 OK to define material property set and close the dialog box. Material> Exit Tool bar>save\_DB

**Step 5:**

Main Menu> Preprocessor> Modeling> create>Key point>In active CS

Create Key point 1 at 0,0,0 then enter Apply and create key point 2 at 1,0,0 then enter ok.

**Step 6:**

Main Menu> Modeling> create> line> line> St Line

Create line through key point 1, 2

**Step 7:**

Main Menu> Preprocessor> Meshing>Mesh tool

*Mesh tool window will appear, in that make a tick mark in the small box (Smart size) and*

*Made the cursor into 1 (fine size) and give, mesh. Pick all for the area to be meshed.*

**# SOLUTION**

**Step 8:**

Main Menu> Solution> Define Loads> Apply> Structural>Displacement> on key points

Select key point 1 and key point 2 fix (UX and UY Constrained)

**Step 9:**

Solution > Analysis type > New analysis > Modal

Solution > Analysis type > Analysis options

A window will appear

Select subspace method and enter 5 in the no of nodes to extract

Check the box beside ‘expand mode shape’ and enter 5 in the no of modes to expand

Click OK

Then a window will appear just give OK

**Step 10:**

Main Menu> Solution> Solve> Current LS –ok

**# POST PROCESSOR**

**Step 11:**

General postproc > result summary

General postproc > read result > first se

Main menu > General postproc > Plot results > Deformed shape

Choose Def + un deformed edge OK

General postproc > read result > next set

Main menu > general postproc > plot results > deformed shape

Choose Def + un deformes edg OK

**Step 12:**

Utility menu > plot ctrl > animate > mode shape A window will appear give OK

**RESULT:**

Thus the frequency response analysis of a simply supported beam was done using ANSYS Software.

**RESULT SUMMARY**

\*\*\*\*\* INDEX OF DATA SETS ON RESULTS FILE \*\*\*\*\*

SET TIME/FREQ LOADSTEP SUBSTEP CUMULATIVE

1 32.984 1 1 1

2 133.35 1 2 2

3 329.00 1 3 3

4 611.37 1 4 4

|  |  |
| --- | --- |
| **EX.NO:07** | **MODAL ANALYSIS OF A FIXED BEAM** |
| **DATE:** |

Analyze the frequency response of a fixed Beam using ANSYS Software

b

b

L

E = 206800e6 N/mm2

L = 1 m

b = 0.01 mm

p = 7830 Kg/m3

v = 0.27

Note: (Refer PSG Design data page no.6.1)

Izz = Ixx + Iyy

= b4/12+b4/12 =b4/6 =0.14/6 =1.667\*10-9 m4

A = b2  = 0.0112  = 0.0001 m2

Where, E – Modulus of Elasticity

V– Poisson Ratio

Izz– Polar moment of Inertia

P - Density of Material

**AIM:**

To analyze the frequency response of a fixed beam using ANSYS software

**SOFTWARE USED:**

ANSYS 19

**PROCEDURE:**

**# PREPROCESSOR**

**Step 1:**

Main menu > Preferences

Click on structural and give ok.

**Step 2:**

Main Menu > Preprocessor > Element Type> Add/Edit/Delete Add an element type. Structural family of beam elements. Choose 2D elastic 3 (beam3) OK to apply the element type and close the dialog box

**Step 3:**

Main Menu> Preprocessor> Real Constants> Add/Edit/Delete Enter 0.0001 for area Enter 1.667e-9 for Izz Enter 0.01 for height and give OK

**Step 4:**

Main Menu> Preprocessor> Material Props> Material Models Double-click on Structural, Linear, Elastic, And Isotropic. Enter 206800e6 for EX Enter 0.27 for PRXY Enter density = 7830 OK to define material property set and close the dialog box. Material> Exit Tool bar>save\_DB

**Step 5:**

Main Menu> Preprocessor> Modeling> create>Key point>In active CS

Create Key point 1 at 0,0,0 then enter Apply and create key point 2 at 1,0,0 then enter ok.

**Step 6:**

Main Menu> Modeling> create> line> line> St Line

Create line through key point 1, 2

**Step 7:**

Main Menu> Preprocessor> Meshing>Mesh tool

*Mesh tool window will appear, in that make a tick mark in the small box (Smart size) and*

*Made the cursor into 1 (fine size) and give, mesh. Pick all for the area to be meshed.*

**# SOLUTION**

**Step 8:**

Main Menu> Solution> Define Loads> Apply> Structural>Displacement> on key points

Select key point 1 and key point 2 fix (UX and UY Constrained)

**Step 9:**

Solution > Analysis type > New analysis > Modal Solution > Analysis type > Analysis options A window will appear Select subspace method and enter 5 in the no of nodes to extract Check the box beside ‘expand mode shape’ and enter 5 in the no of modes to expand Click OK Then a window will appear just give OK

**Step 10:**

Main Menu> Solution> Solve> Current LS –ok

**# POST PROCESSOR**

**Step 11:**

General postproc > result summary

General posproc > read result > first se

Main menu > General postproc > Plot results > Deformed shape

Choose Def + un deformed edge OK

General postproc > read result > next set

Main menu > general postproc > plot results > deformed shape

Choose Def + un deformes edg OK

**Step 12:**

Utility menu > plot ctrl > animate > mode shape A window will appear give OK

**RESULT:**

Thus the frequency response analysis of a cantilever beam was done using ANSYS Software.

**RESULT SUMMARY**

\*\*\*\*\* INDEX OF DATA SETS ON RESULTS FILE \*\*\*\*\*

SET TIME/FREQ LOADSTEP SUBSTEP CUMULATIVE

1 75.014 1 1 1

2 209.98 1 2 2

3 488.13 1 3 3

4 969.16 1 4 4

5 2688.0 1 5 5

|  |  |
| --- | --- |
| **EX.NO:08** | **MODAL ANALYSIS OF A 2D PLATE** |
| **DATE:** |

Analyze the frequency response of a 2D plate using ANSYS Software

10

10

E = 30e6 N/mm2

p = 0.786 x 10-5  Kg/mm3

v = 0.27

**AIM**

To analyze the frequency response of a 2D plate using ANSYS software

**SOFTWARE USED:**

ANSYS 19

**PROCEDURE:**

# **PREPROCESSOR**

**Step 1:**

Main Menu > Preferences Click on Structural and give OK

**Step 2:**

1. Main Menu> Preprocessor> Element Type> Add/Edit/Delete
2. Add an element type.
3. Structural solid family of elements.
4. Choose the 8-node quad (PLANE82).
5. OK to apply the element type and close the dialog box.
6. Options for PLANE82 are to be defined.
7. Choose plane stress with thickness option for element behavior.
8. OK to specify options and close the options dialog box.
9. Close the element type dialog box.

**Step 3:**

1. Real Constant > Add/Edit/Delete 2. Add an Real Constant 3. Type 1 PLANE 82 > OK > Thickness > 1 > OK 4. Close the Real Constant dialog box

**Step 4:**

1. Main Menu> Preprocessor> Material Props> Material Model

2. Double-click on Structural, Linear, Elastic, And Isotropic.

3. Enter 30e6 for EX

4. Enter 0.27 for PRXY

5. Enter density = 0.786e-5

6. OK to define material property set and close the dialog box.

7. Material> Exit

Tool bar> save\_DB

**Step 5:**

1. Main Menu > Preprocessor > Modeling > Create > Areas > Rectangles >By Dimensions2. Enter the Following X1 = 0 (Note: Press the TAB key between circles)X2 = 10Y1 = 0 Y2 = 2.5

**Step 6:**

Main Menu > Preprocessor > Meshing > Meshtool *Meshtool window will appear in that make a tick mark in the small box (smart size) and make the cursor into 11 (fine size) and give mesh*Pick all for the area to be mesh

**#SOLUTION**

**Step 7:** Main Menu > Solution > Define Loads > Apply > Structural > Displacement > on line Select the left vertical edge and constrain in all DOF Save\_DB

**Step 8:** 1. Solution > Analysis Type > New Analysis > Modal 2. Solution > Analysis Type > Analysis Options A window will appear Select subspace method and enter 5 in the no.of nodes to Extract Check the Box beside ‘expand mode shape’ and enter 5 in the no of modes to expand Click OK Then a window will appear just give OK

**#POSTPROCESSOR**

**Step 10:** 1. General Postproc > Result summary 2. General Postproc > Read Result > First Set This selects the results for the first mode shape 3. Main Menu > General Postproc > Plot Results > Deformed Shape Choose Def + un deformed edg OK 4. The first mode shape window will appear in the graphics window 5. General Postproc > Read Result > Next Set 6. Main Menu > General Postproc > Plot Results > Deformed Shape 7. Choose Def + undeformed edg OK

**Step 11:** Utility Menu > Plot Control > Animate > Mode Shape

**RESULT:**

Thus the frequency response analysis of a 2D plate was done using ANSYS Software

**RESULT SUMMARY**

\*\*\*\*\* INDEX OF DATA SETS ON RESULTS FILE \*\*\*\*\*

SET TIME/FREQ LOADSTEP SUBSTEP CUMULATIVE

1 7554.3 1 1 1

2 38479. 1 2 2

3 48944. 1 3 3

4 88573. 1 4 4

5 0.14324E+06 1 5 5

|  |  |
| --- | --- |
| **EX.NO:09** | **HARMONIC ANALYSIS OF A CANTILEVER BEAM** |
| **DATE:** |

Analyze the given Cantilever beam under harmonic condition using ANSYS software.

F

b

b

L

E = 206800 N/mm2

L = 1000 mm

b = 10 mm

F = 100 N

p = 0.783 x 10-5 kg/mm2

v = 0.3

Note: (Refer PSG Design data page no.6.1)

Izz = Ixx + Iyy

= b4/12+b4/12 =b4/6 =104/6 =1666.667 mm4

A = b2  = 102  = 100 mm2

Where, E – Modulus of Elasticity

V– Poisson Ratio

Izz– Polar moment of Inertia

A – Area

**AIM:**

To analyze the fixed beam under frequency using ANSYS software

**SOFTWARE USED:**

ANSYS 19

**PROCEDURE:**

**# PREPROCESSOR**

**Step 1:**

Main menu > Preferences Click on structural and give ok.

**Step 2:**

Main Menu > Preprocessor > Element Type> Add/Edit/Delete Add an element type. Structural family of beam elements. Choose 2D elastic 3 ( beam3 ) OK to apply the element type and close the dialog box

**Step 3:**

Main Menu> Preprocessor> Real Constants> Add/Edit/Delete Enter 100 for area Enter 1666.667 for Izz Enter 10 for height and give OK

**Step 4:**

Main Menu> Preprocessor> Material Props> Material Models Double-click on Structural, Linear, Elastic, And Isotropic. Enter 206800 for EX Enter 0.3 for PRXY Enter density = 0.783e-5 OK to define material property set and close the dialog box. Material> Exit Tool bar>save\_DB

**Step 5:**

Main Menu> Preprocessor> Modeling> create>Key point>In active CS Create Key point 1 at 0,0,0 then enter Apply and create key point 2 at 1000,0,0 then enter ok.

**Step 6:**

Main Menu> Modeling> create> line> line> St Line Create line through key point 1, 2

**Step 7:**

Main Menu> Preprocessor> Meshing>Mesh tool *Mesh tool window will appear, in that make a tick mark in the small box (Smart size) and*  *Made the cursor into 1 (fine size) and give, mesh. Pick all for the area to be meshed.*

**# SOLUTION**

**Step 8:**

Main Menu> Solution> Define Loads> Apply> Structural>Displacement> on keypoints Select key point 1 and fix (all DOF Constrained)

**Step 9:**

Main Menu> Solution> Define Loads> Apply> Structural> Force/Moment> on keypoints Select direction of Force/Moment FY Force/Moment Value -100 then give OK and save\_DB Main Menu> Solution> Solve> Current LS –OK

**Step 10:**

Solution > Analysis type > New Analysis > Harmonic

**Step 11:**

Solution > load step opts > Time/Frequency > Freq & sub steps A window will appear specify a Harmonic frequency range of 0 -100 Hz, 100 No. of steps and stepped B.C

**Step 12:**

Main Menu > Solution > Current LS-OK

**Step 13:**

1. Open the time history postprocessor a window will appear by default variable its assigned either time or frequency n our case it is assigned frequency we want to see the displacement UY to the node at x = 1 which is node number 2

2. Time history variable window will appear then go for add(+) again a window will appear there go for nodal solution > DOF solution >Y-component of displacement then click OK

3. In the ‘time history variables’ window click the ‘list’ button(4th buton to the right of add)

4. In the ‘time history variables’ window click the ‘graph data’ button (2nd button to the right of add)

**RESULT:**

Thus the stress analysis on cantilever beam frequency was done using ANSYS Software.

|  |  |
| --- | --- |
| **EX.NO:10** | **TEMPERATURE DISTRIBUTION ANALYSIS OF A RECTANGULAR PLATE UNDER THERMAL MIXED BOUNDARY** |
| **DATE:** |

Analyze the rectangular plate under thermal mixed boundary conditions using ANSYS software.

internal heat generation T=500

internal heat generation condution

T convection

L

Insulated portion

Primary Data L = 1 m k = 10 W/moc Film coefficient = 10 Bulk Temperature = 100

Secondary Data h = 10 W/m2o c T∞ = 30 oc

Where

k- Thermal Conductivity of Plate

h- Heat Transfer coefficient at outer side

T∞-  Atmospheric air Temperature

**AIM:**

To analyze the rectangular plate under thermal boundary condition using ANSYS software

**SOFTWARE USED:**

ANSYS 19

**PROCEDURE:**

**# PREPROCESSOR**

**Step 1:**

Main menu > Preferences Click on Thermal and give OK.

**Step 2:**

Main Menu > Preprocessor > Element Type> Add/Edit/Delete Add an element type. Thermal mass solid family of elements. Choose the Quad 4 node 55 OK to apply the element type and close the dialog box

**Step 3:**

Main Menu> Preprocessor> Material Props> Material Models > Thermal > conductivity >Isotropic KXX=10 OK to define material property set and close the dialog box.

**Step 4:**

Main Menu > Preprocessor > Modeling > create >Area > Rectangle > By 2 corners > X=0,Y=0,Height=1,Width=1

**Step 5:**

Main Menu> Preprocessor> Meshing>Mesh tool *Mesh tool window will appear, in that make a tick mark in the small box (Smart size) and*  *Made the cursor into 1 (fine size) and give, mesh.* Pick all for the area to be meshed*.*

**# SOLUTION**

**Step 6:**

Main Menu> Solution> Define Loads> Apply> Thermal > Temperature > On Lines Select the top line of the block and constraint it to a constant value of 500C Using the same method constrain the left side of the block to a constant value of 100C

**Step 7:**

Main Menu> Solution> Define Loads> Apply> Thermal> Convection> on Lines Select the right side of the block A window will appear Give film coefficient = 10 Bulk Temperature =100 and give OK

**Step 8:**

Solution > Define load > apply > Thermal > Convection > On Lines Select the bottom of the Block Enter a Constant film coefficient of 0. This will eliminate convection through the side thereby modeling an insulated wall

**Step 9:**

Main Menu > Solution > Current LS-OK

**#POST PROCESSOR**

**Step 10:**

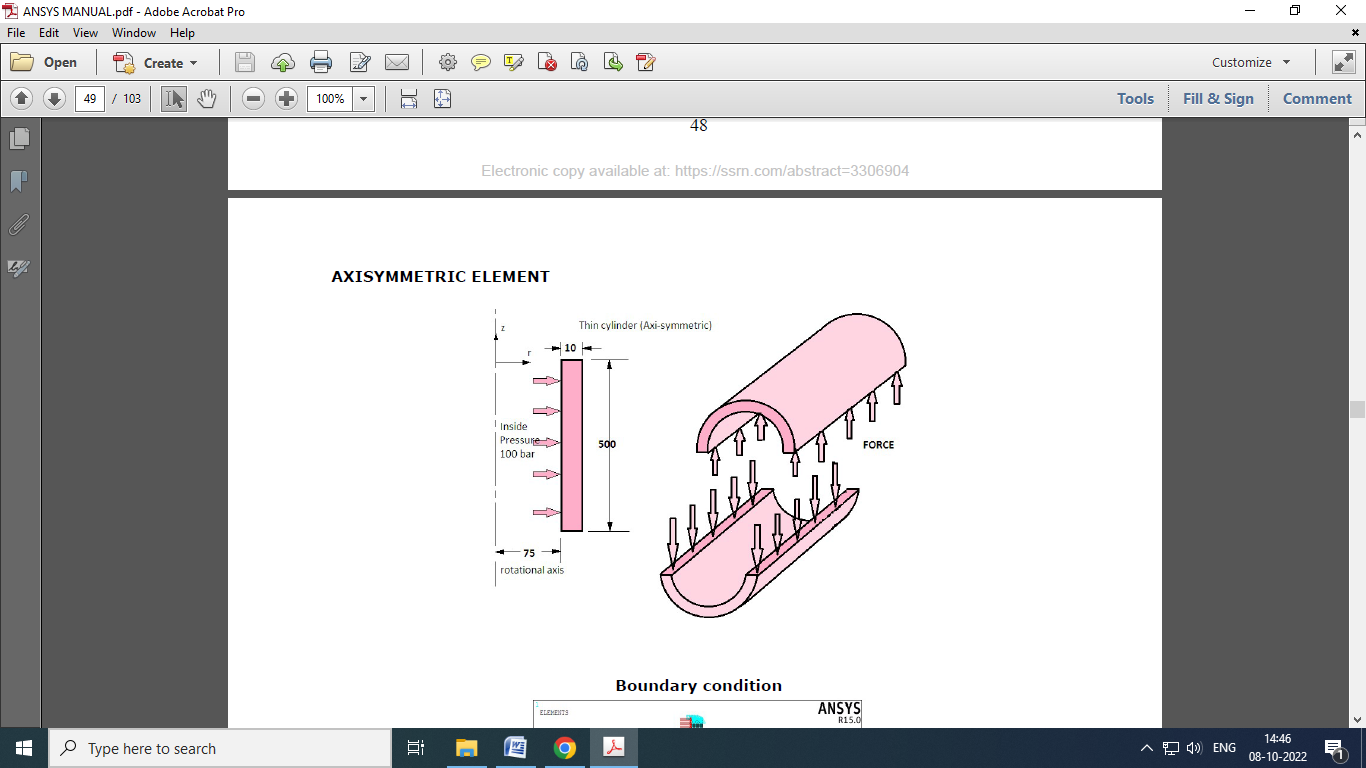
General postproc > plot result > contour plot > nodal solu > DOF solution > Nodal temperature

**RESULT:**

Thus the analysis of a rectangular plate under thermal mixed boundary was done using ANSYS Software.

|  |  |
| --- | --- |
| **EX.NO:11** | **STRESS ANALYSIS OF AXI-SYMMETRIC COMPONENTS** |
| **DATE:** |

Analyze the closed hollow circular shaft under point load using ANSYS software.



Outer Diameter = 170 mm E = 200000 N/mm2 Inner Diameter = 150 mm F = 100 bar Outer Height = 500 mm v = 0.3 Inner Height = 480 mm

**NOTE:** Axisymmetric solids are pictured as numbered of rectangular element as shown in below figure. But these shapes are actually cross-sectional of closed hollow circular shaft.

**AIM:**

To analyze the closed hollow circular shaft under point load using ANSYS software

**SOFTWARE USED:**

ANSYS 19

**PROCEDURE:**

**# PREPROCESSOR**

**Step 1:**

Main menu > Preferences Click on structural and give OK

**Step 2:**

Main Menu > Preprocessor > Element Type> Add/Edit/Delete Add an element type. Structural solid family of elements. Choose Triangle 6 node 2 OK to apply the element type and close the dialog box. In element type window click options button and select Axisymmetric (in element behavior K3)

**Step 3:**

Main Menu> Preprocessor> Material Props> Material Models Double-click on Structural, Linear, Elastic, And Isotropic. Enter 200000 for EX Enter 0.3 for PRXY OK to define material property set and close the dialog box. Material> Exit

**Step 4:**

1. Main Menu> Preprocessor> Modeling> create>Areas>Rectangle>By Dimensions

**Step 5:**

Main Menu> Preprocessor> Modeling> Operate> Booleans> Add> Areas Click Pick all button to create a single area

**Step 6:**

Main Menu> Preprocessor> Meshing>Mesh tool *Mesh tool window will appear, in that make a tick mark in the small box (Smart size) and*  *Made the cursor into 1 (fine size) and give, mesh.* Pick all for the area to be meshed*.*

**# SOLUTION**

**Step 7:**

Main Menu> Solution> Define Loads> Apply> Structural> Force/Moment> on keypoints Pick the top left corner of the area and click OK apply load of **-100** in the FY direction Main Menu> Solution> Define Loads > Apply> Structural > Force/Moment> on keypoints Pick the bottom left corner of the area and click OK apply load of **100** in the FY direction

**Step 8:**

Main Menu> Solution> Solve> Current LS –OK Close the warning message

**#POST PROCESSOR**

**Step 9:**

Main Menu> General Postproc> Plot Results> Deformed Shape Choose Def + Undeformed. OK.

**Step 12:**

Plot the Von Mises equivalent stress. Main Menu> general Postproc> Plot Results> Contour Plot> Nodal solution 1. Choose Stress item to be contoured. 2. Scroll down and choose Von Mises ( SEQU ) OK. 3. Utility Menu> Plot Ctrls> Animate> Deformed Results 4. Choose Stress item to be contoured. 5. Scroll down and choose Von Mises ( SEQU ) OK. 6. Make choices in the Animation Controller (not shown), if necessary, then choose close.

**RESULT:**

Thus the analysis of a closed hollow circular shaft under point load was done using ANSYS Software.

|  |  |
| --- | --- |
| **EX.NO:12** | **COUPLED FIELD ANALYSIS OF A 2D COMPONENT** |
| **DATE:** |

Analyze the square plate under coupled field using ANSYS software.

Internal heat generation T=500

Internal heat generation conduction

T convection

L

L

Primary Data L = 1 m k = 10 W/moc Film coefficient = 10 Bulk Temperature = 100 E = 200000 N/mm2 v = 0.3 Co-efficient of thermal expansion(i.e)Secant coefficient =1.66 x 10-5

Secondary Data h = 10 W/m2o c T∞ = 30 oc

Where

k- Thermal Conductivity of Plate h- Heat Transfer coefficient at outer side T∞-  Atmospheric air Temperature

**AIM:**

To analyze the square plate under coupled field using ANSYS software

**SOFTWARE USED:**

ANSYS 10

**PROCEDURE:**

**# PREPROCESSOR 1**

Create file using change job name in utility menu keep the name as Thermal 1

**Step 1:**

Don’t give Preferences

**Step 2:**

Main Menu > Preprocessor > Element Type> Add/Edit/Delete Add an element type. Thermal mass solid family of elements. Choose the Quad 4 node 55 OK to apply the element type and close the dialog box

**Step 3:**

Main Menu> Preprocessor> Material Props> Material Models > Thermal > conductivity >Isotropic KXX=10 OK to define material property set and close the dialog box.

**Step 4:**

Main Menu > Preprocessor > Modeling > create >Area > Rectangle > By 2 corners > X=0,Y=0,Height=1,Width=1

**Step 5:**

Main Menu> Preprocessor> Meshing>Mesh tool *Mesh tool window will appear, in that make a tick mark in the small box (Smart size) and*  *Made the cursor into 1 (fine size) and give, mesh.* Pick all for the area to be meshed

**# SOLUTION 1**

**Step 6:**

Main Menu> Solution> Define Loads> Apply> Thermal > Temperature > On Lines Select the top line of the block and constraint it to a constant value of 500C Using the same method constrain the left side of the block to a constant value of 100C

**Step 7:**

Main Menu> Solution> Define Loads> Apply> Thermal> Convection> on Lines Select the right side of the block A window will appear Give film coefficient = 10 Bulk Temperature =100 and give OK

**Step 8:**

Main Menu > Solution > Current LS-OK

**# PREPROCESSOR 2**

**Step 9:**

Go to change job name in utility menu and change the name as thermal 12. After entering the job name close the warning message

**Step 10:**

Preprocessor > Load > Define loads > Delete > All load data > All load & opts and give OK

**Step 11:**

Main menu> Preprocessor > Element type > Switch element type > Thermal to stuc > OK Close the warning message

**Step 12:**

Main menu > Preprocessor > Material props > Material Models Double-click on Structural, Linear, Elastic and Isotropic Enter 200000 for EX Enter 0.3 for PRXY Thermal Expansion > Secant Co-efficient > Isotropic > ALPX > 1.66e-5 OK to define material property set and close the dialog box Material > Exit

**# SOLUTION 2**

**Step 13:**

Main menu > Solution > Define loads > Apply > Structural > displacement > on lines Pick the left side vertical line > all DOF

**Step 14:**

Main menu > Solution > Define loads > Apply > temperature > from thermal analysis Now browse and take the file named thermal.rth open and give OK

**Step 15:**

Main menu > Solution > Solve > Current LS-OK

**#POST PROCESSOR**

**Step 16:**

Main Menu> General Postproc> Plot Results> Deformed Shape Choose Def + Undeformed. OK.

**Step 17:**

Plot the Von Mises equivalent stress. Menu> general Postproc> Main Plot Results> Contour Plot> Nodal solution Choose Stress item to be contoured. Scroll down and choose Von Mises ( SEQU ) OK Utility Menu> Plot Ctrls> Animate> Deformed Results Choose Stress item to be contoured Utility Menu> Plot Ctrls> Animate> Deformed Results Scroll down and choose Von Mises ( SEQU ) OK. Make choices in the Animation Controller (not shown), if necessary, then choose close.

**RESULT:**

Thus the analysis of a square plate under coupled field was done using ANSYS Software.

|  |  |
| --- | --- |
| **EX.NO:13** | **CONDUCTION AND CONVECTIVE HEAT TRANSFER ANALYSIS OF A 2D COMPONENT** |
| **DATE:** |

Analyze the rectangular plate under conductive and convective conditions using ANSYS software.

internal heat generation T=500

internal heat generation condution

T convection

L

L

Primary Data L = 1 m k = 10 W/moc Film coefficient = 10 Bulk Temperature = 100

Secondary Data h = 10 W/m2o c T∞ = 30 oc

Where

k- Thermal Conductivity of Plate

h- Heat Transfer coefficient at outer side

T∞-  Atmospheric air Temperature

**AIM:**

To analyze the rectangular plate under conduction and convection using ANSYS software

**SOFTWARE USED:**

ANSYS 19

**PROCEDURE:**

**# PREPROCESSOR**

**Step 1:**

Main menu > Preferences Click on Thermal and give OK.

**Step 2:**

Main Menu > Preprocessor > Element Type> Add/Edit/Delete Add an element type. Thermal mass solid family of elements. Choose the Quad 4 node 55 OK to apply the element type and close the dialog box

**Step 3:**

Main Menu> Preprocessor> Material Props> Material Models > Thermal > conductivity >Isotropic KXX=10 OK to define material property set and close the dialog box.

**Step 4:**

Main Menu > Preprocessor > Modeling > create >Area > Rectangle > By 2 corners > X=0,Y=0,Height=1,Width=1

**Step 5:**

Main Menu> Preprocessor> Meshing>Mesh tool *Mesh tool window will appear, in that make a tick mark in the small box (Smart size) and*  *Made the cursor into 1 (fine size) and give, mesh.* Pick all for the area to be meshed*.*

**# SOLUTION**

**Step 6:**

Main Menu> Solution> Define Loads> Apply> Thermal > Temperature > On Lines Select the left side of the block and constraint it to a constant value of 100C

**Step 7:**

Main Menu> Solution> Define Loads> Apply> Thermal> Convection> on Lines Select the right side of the block A window will appear Give film coefficient = 10 Bulk Temperature =100 and give OK

**Step 8:**

Solution > Define load > apply > Thermal > Convection > On Lines Select the bottom of the Block Enter a Constant film coefficient of 0 and Bulk Temperature of 0. This will eliminate convection through the side thereby modeling an insulated wall

**Step 9:**

Main Menu > Solution > Current LS-OK

**#POST PROCESSOR**

**Step 10:**

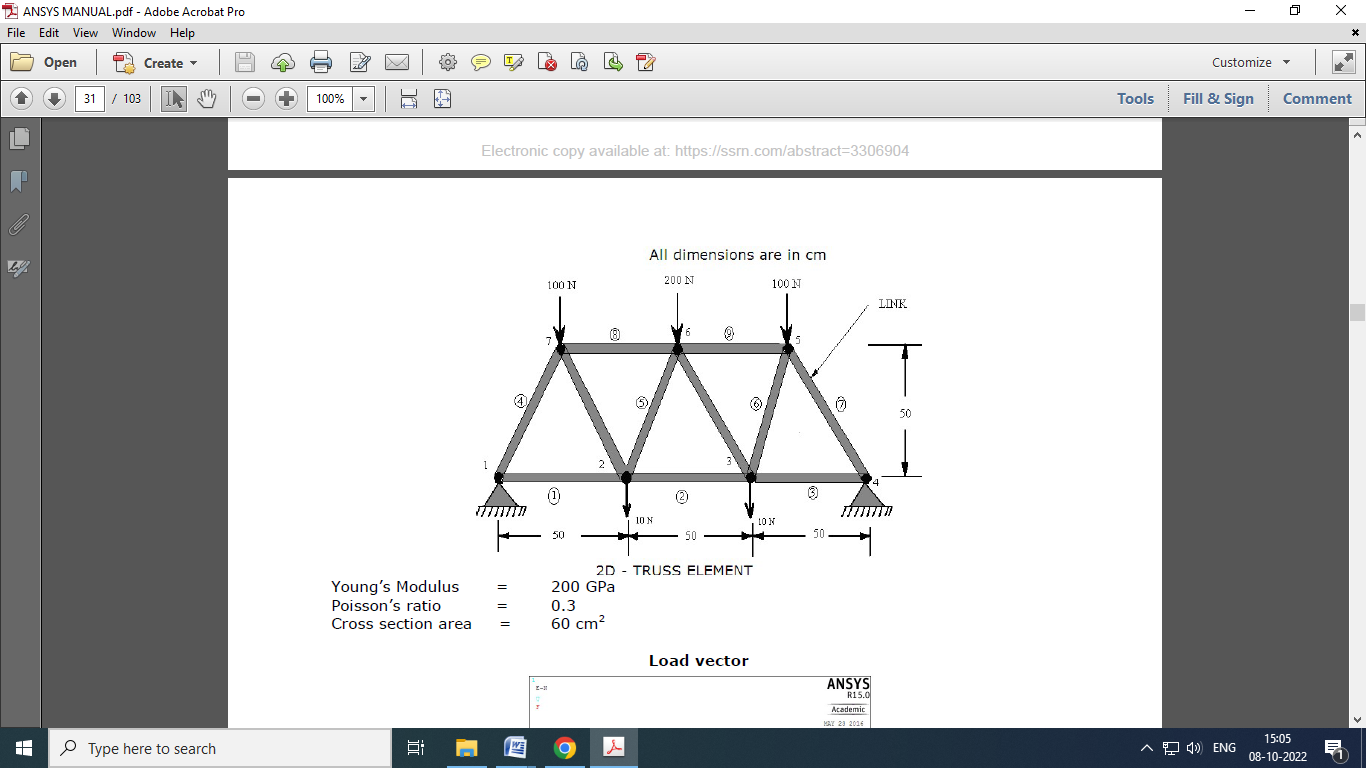
General postproc > plot result > contour plot > nodal solu > DOF solution > Nodal temperature

**RESULT:**

Thus the analysis of a rectangular plate under thermal mixed boundary was done using ANSYS Software.

|  |  |
| --- | --- |
| **EX.NO:14** | **FORCE AND STRESS ANALYSIS USING LINK ELEMENTS IN TRUSSES.** |
| **DATE:** |

Analyze the TRUSS under given load using ANSYS software.



**AIM:-**

To perform static analysis of given 2D truss element using ANSYS 19 analysis

software.

**SOFTWARE USED:**

ANSYS 19

**PROCEDURE:**

It involves three basic types of operations

1. Pre Processing

2. Solution

3. Post Processing

**Preprocessing:-**

1. Structural / h – method/structural

2. Preprocessor/Element type/Add/edit/delete/Add/ link/ 3D finit stn 180/ok/close

3. Real constants/Add//edit/delete/Add/ok/Enter cross sectional area = 60

4. Material props/material models / structural / linear / elastic / isotropic / Exx=2e9

/PRxy=0.3/ok/close.

5. Modeling/Create/Nodes/ Inactive CS/x=0 y=0 z =0/Apply/x=50 y=0 z=0/ Apply /

x=100 y=0 z=0/ Apply /x=150 y=0 z=0/ Apply /x=125 y=50 z=0/ Apply /x=75 y=50

z=0/ Apply / x=25 y=50 z=0 /ok.

6. Modeling/Create/elements/Auto numbered/Throu nodes/Select keypoint1 and track

the mouse into select key point 2.[similarly join all the points as per given truss

diagram].

Solution:-

7. Solution/Define Loads /Apply/Structural / Displacement /on key points/selectnode1and node 2/Select All DOF /ok.

8. Solution/Define Loads /Apply/Structural / Force/moment/select nodes 5 and 7/enter

the value Fy= -100/Apply/ select node 6/enter the value Fy= -200/ Apply/select nodes

2 and 3/enter the value Fy= -10/ok

9. Solutions/solve/current LS/solution is done.

**Post Processing:-**

10. Plot results for deformed/ Def + unreformed shape/plot controls/capture image.

11. Plot results for Contour plot/Nodal solu/Nodal solution/DOF solution/ X component

of displacement. /plot controls/capture image.

12. Plot results for Contour plot/Nodal solu/Nodal solution/DOF solution/ Y component

of displacement. /plot controls/capture image.

13. Plot results for Contour plot/Element solution/stress/Von mises stress/plot

controls/capture image.

14. Plot results for Contour plot/Element solution/Total mechanical strain/Von mises

strain/plot controls/capture image.

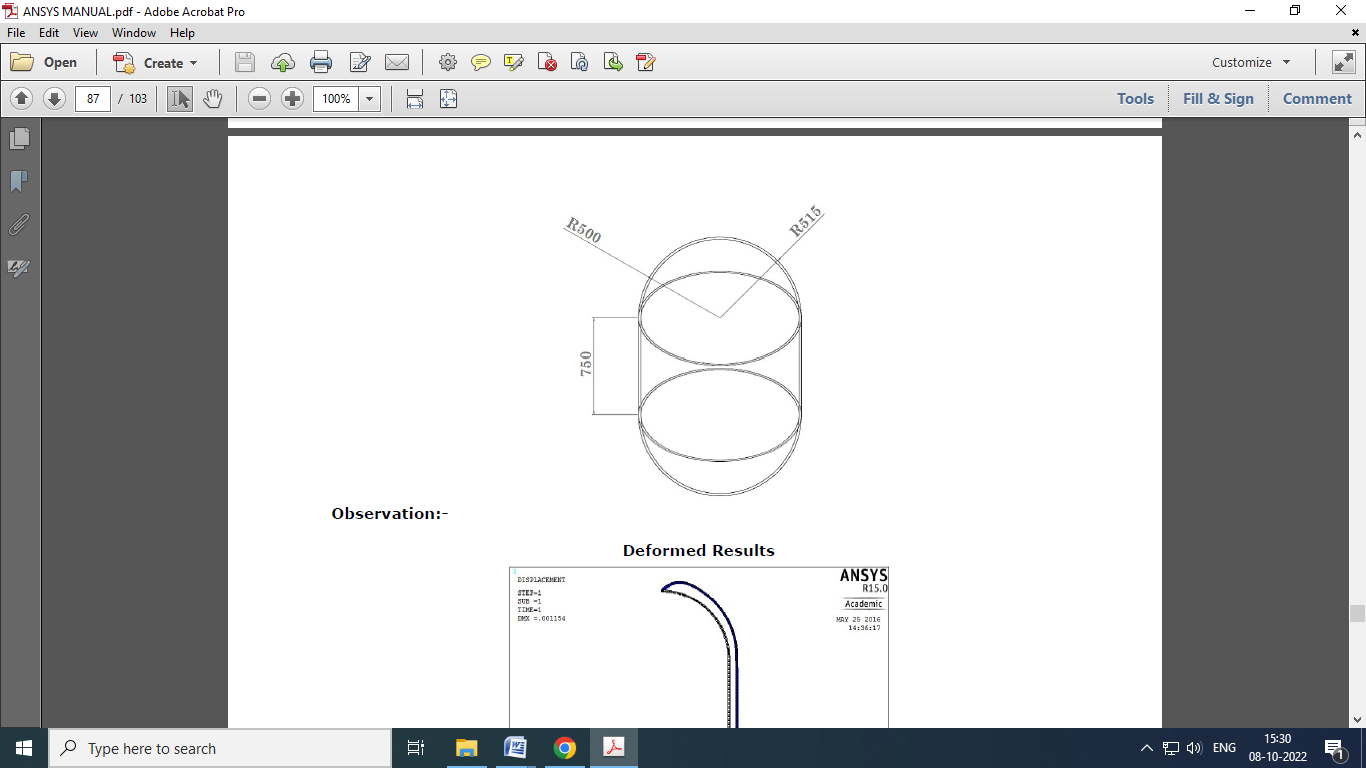
**RESULT:-**

Thus the static analysis of given 2D truss element was performed by using ANSYS APDL 19

Analysis software.

|  |  |
| --- | --- |
| **EX.NO:15** | **THERMAL STRESS ANALYSIS OF CYLINDRICAL SHELL.** |
| **DATE:** |

Analyze the Thermal Stress on given **CYLINDRICAL SHELL** using ANSYS software.



**AIM:-**

To perform static thermal analysis of thin Cylindrical shell (axi-symmetric) by using ANSYS 19

analysis software.

**SOFTWARE USED:**

ANSYS 19

**PROCEDURE:**

It involves three basic types of operations

1. Pre Processing

2. Solution

3. Post Processing

**Preprocessing:**

1. Structural / h – method/structural

2. Preprocessor/Element type/Add/edit/delete/Add/ solid/ Quad 4 node 182 / ok /

options/Axisymmetric/ok/close

3. Material prop/import lib/Browse/C drive/Program files/Ansys inc/v150/ ansys / mat

lib /stl\_AISI-304/ok/close.

4. Modeling/Create/Area/Rectangle/By 2 corner/X = 0.5, Y=0, Width = 0.015 and

Height = 0.75 /ok.

5. Modeling/Create/Area/ circle/ By dimensions/outer radius = 0.515 / inner radius =

0.5 /starting angle = 270 /Ending angle = 360/apply/ outer radius = 0.515 / inner

radius = 0.5 /starting angle = 0/ Ending angle = 90/ok.

6. Modeling/ move/modify/areas/area/Y offset = 0.75/ok

7. Modeling / operate/ Booleans/ Add/ select area 1 and 2/apply/ select area 1 and 3/

ok.

5. Meshing/mesh tool/size controls/area/ set (click)/select area/enter element size =

0.001/ok

6. Meshing/mesh tool/ mesh/ select area/ok.

Solution:-

7. Solution/Define Loads /Apply/Structural / Displacement /on lines/select top and

bottom lines/All dof/ok

8. Solution/Define Loads /Apply/Structural /pressure/on lines/ select vertical inside

line/ok/enter the pressure value = 5 e5/ok

9. Solution/Solve/Current LS/ok/Solution is done.

**Post Processing:-**

10. Plot results for deformed/ Def + unreformed shape/plot controls/capture image.

10. Plot results for deformed/ Def + unreformed shape/plot controls/Style/size and

shape/Display object /on/ok

11. Plot controls/Style/size and shape/Symmetric expansion/2D axi-symmetric/select

the options/ok/plot controls/capture image.

12. Plot results for Contour plot/Nodal solu/Nodal solution/DOF solution/ X component

of displacement. /plot controls/capture image.

13. Plot results for Contour plot/Nodal solu/Nodal solution/DOF solution/ Y component

of displacement. /plot controls/capture image.

14. Plot results for Contour plot/Nodal solution/stress/ Von mises stress /plot

controls/capture image.

15. Plot results for Contour plot/Nodal solution/Total Mechanical strain/von mises

strain/plot controls/capture image.

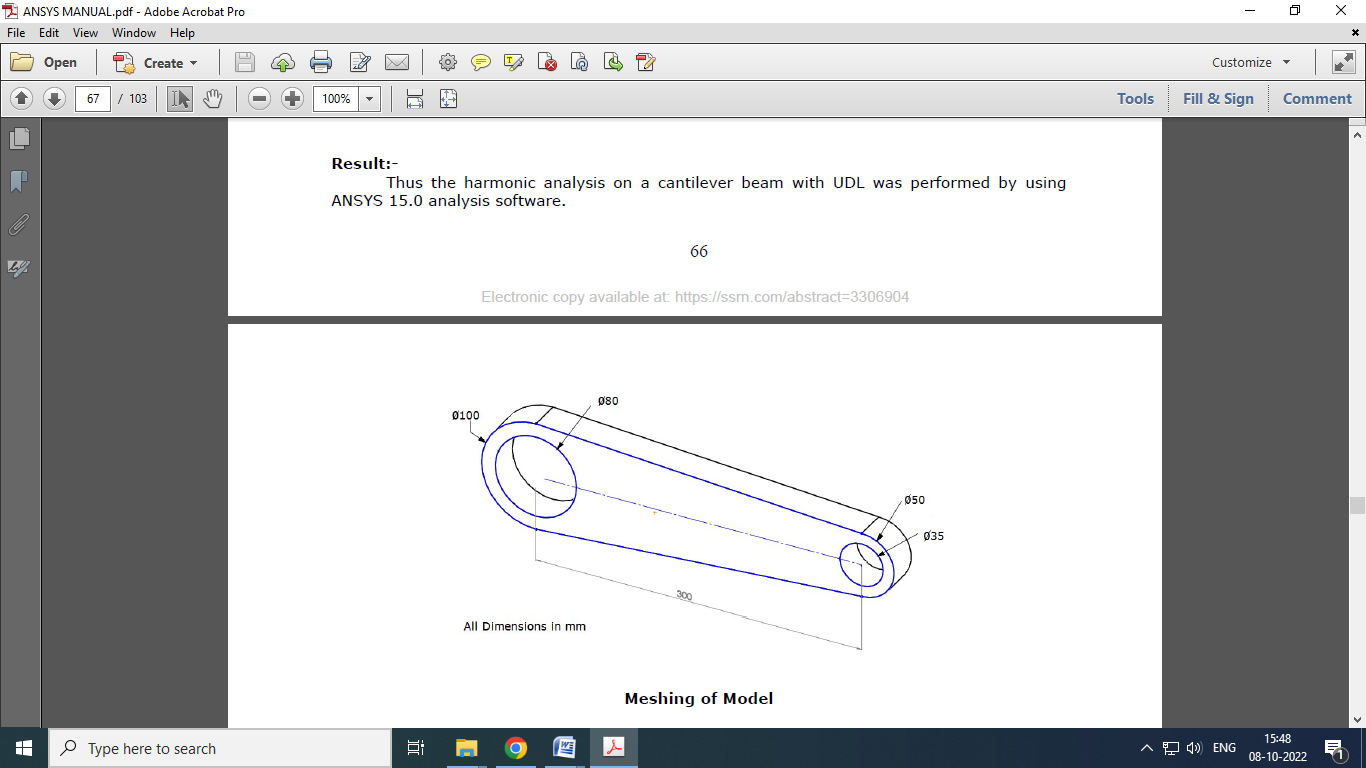
**RESULT:-**

Thus the thermal analysis of given CYLINDRICAL SHELL was performed by using ANSYS

19 analysis software.

|  |  |
| --- | --- |
| **EX.NO:16** | **TRANSIENT ANALYSIS ON A SIMPLE MECHANICAL ELEMENT** |
| **DATE:** |

Perform Conductive heat transfer analysis on a 3D cube by using ANSYS 19 analysis software.



**AIM:**

To perform Transient heat transfer analysis on a 3D cube by using ANSYS 19 analysis software.

**SOFTWARE USED:**

ANSYS 19

**PROCEDURE:**

It involves three basic types of operations

1. Pre Processing

2. Solution

3. Post Processing

**Preprocessing:**

1. Thermal / h – method/structural

2. Element type /Add/edits / delete / add/solid / Quad 4 node 182/ok/options/plane

stress with thk /ok /close

3. Real constants/Add//edit/delete/Add/ok/Enter cross thickness = 0.01

4. Material prop/import lib/Browse/C drive/Program files/Ansys inc /v150/ ansys / mat

lib /stl\_AISI-304/ok/close.

5. Modeling/create / Area /circle/ Solid circle / x=0, y=0, Radius = 0.05/Apply/ x = 0.3,

y = 0 & R = 0.025/ ok.

6. Modeling/create / Area /arbitrary/ throu key points/ select big circle top point first

then select small circle top point/ then small circle bottom point and finally big circle

bottom point/ ok.

7. Modeling/ Operate/ Booleans/ add/ select area1/apply/area3/apply/ select area2/

apply /select area3/ ok.

8. Modeling/create / Area /circle/ Solid circle / x=0, y=0, Radius = 0.04/Apply/ x = 0.3,

y = 0 & R = 0.0175 / ok.

9. Meshing/ mesh tool / mesh attributes/ area/set/select area/ok.

10. Meshing/ mesh tool / size controls/ area/set/select area/ element length = 0.01

/ok.

11. Meshing/ mesh tool /area / mesh/select area/ok.

**Solution:-**

12. Analysis type/ new analysis/ Transient/ok

13. Analysis type/ soln control/ Time at end of load step = 5/ No of sub steps = 5/ Max

no of sub steps = 5/ Min no of sub steps = 1/ok

14. Define the Loads /Apply / structural/ displacement/ on line/select small hole’s inner

lines/ all dof/ ok.

15. Define the Loads /Apply / structural/ Pressure/ on line/ select Big hole’s inner lines/

Pressure value = 5/ ok.

16. Load step option/ write LS file/ file no: 1/ ok.

17. Analysis type/ soln control/ Time at end of load step = 10/ No of sub steps = 5/

Max no of sub steps = 5/ Min no of sub steps = 1/ok

18. Define the Loads /Delete / structural/ displacement/ on line/select Big hole’s inner

lines/ ok.

19. Define the Loads /Apply / structural/ Pressure/ on line/ select Big hole’s inner lines/

Pressure value = 10/ ok.

20. Load step option/ write LS file/ file no: 2/ ok.

17. Analysis type/ soln control/ Time at end of load step = 15/ No of sub steps = 5/

Max no of sub steps = 5/ Min no of sub steps = 1/ok

18. Define the Loads /Delete / structural/ displacement/ on line/select Big hole’s inner

lines/ ok.

19. Define the Loads /Apply / structural/ Pressure/ on line/ select Big hole’s inner lines/

Pressure value = 12/ ok.

20. Load step option/ write LS file/ file no: 3/ ok.

21. Solve/ From LS files/ starting file number =1/ ending file no = 3/ file increment =

1/ok/ solution is done.

**Post Processing:-**

21. Read results / by pick / select first load step/read/ close.

22. Plot results /Nodal solution /DOF solution/ Displacement vector sum/ ok. (Capture

Image)

23. Plot results /Nodal solution /stress/ von mises stress/ ok. (Capture Image)

24. Plot results /Nodal solution / Total mechanical strain/ von mises strain/ ok.

(Capture Image)

25. Read results / by pick / select second load step/read/ close.

26. Plot results /Nodal solution /DOF solution/ Displacement vector sum/ ok. (Capture

Image)

27. Plot results /Nodal solution /stress/ von mises stress/ ok. (Capture Image)

28. Plot results /Nodal solution / Total mechanical strain/ von mises strain/ ok.

(Capture Image)

29. Read results / by pick / select third load step/read/ close.

30. Plot results /Nodal solution /DOF solution/ Displacement vector sum/ ok. (Capture

Image)

31. Plot results /Nodal solution /stress/ von mises stress/ ok. (Capture Image)

32. Plot results /Nodal solution / Total mechanical strain/ von mises strain/ ok.

(Capture Image)

**RESULT:**

Thus the transient analysis on a simple mechanical element was performed using Ansys 19 analysis software.