

THREE-DIMENSIONAL PROBLEM OF THE THEORY OF ELASTICITY STRESS IN A THICK-WALLED PRESSURE VESSEL

1. INTRODUCTION

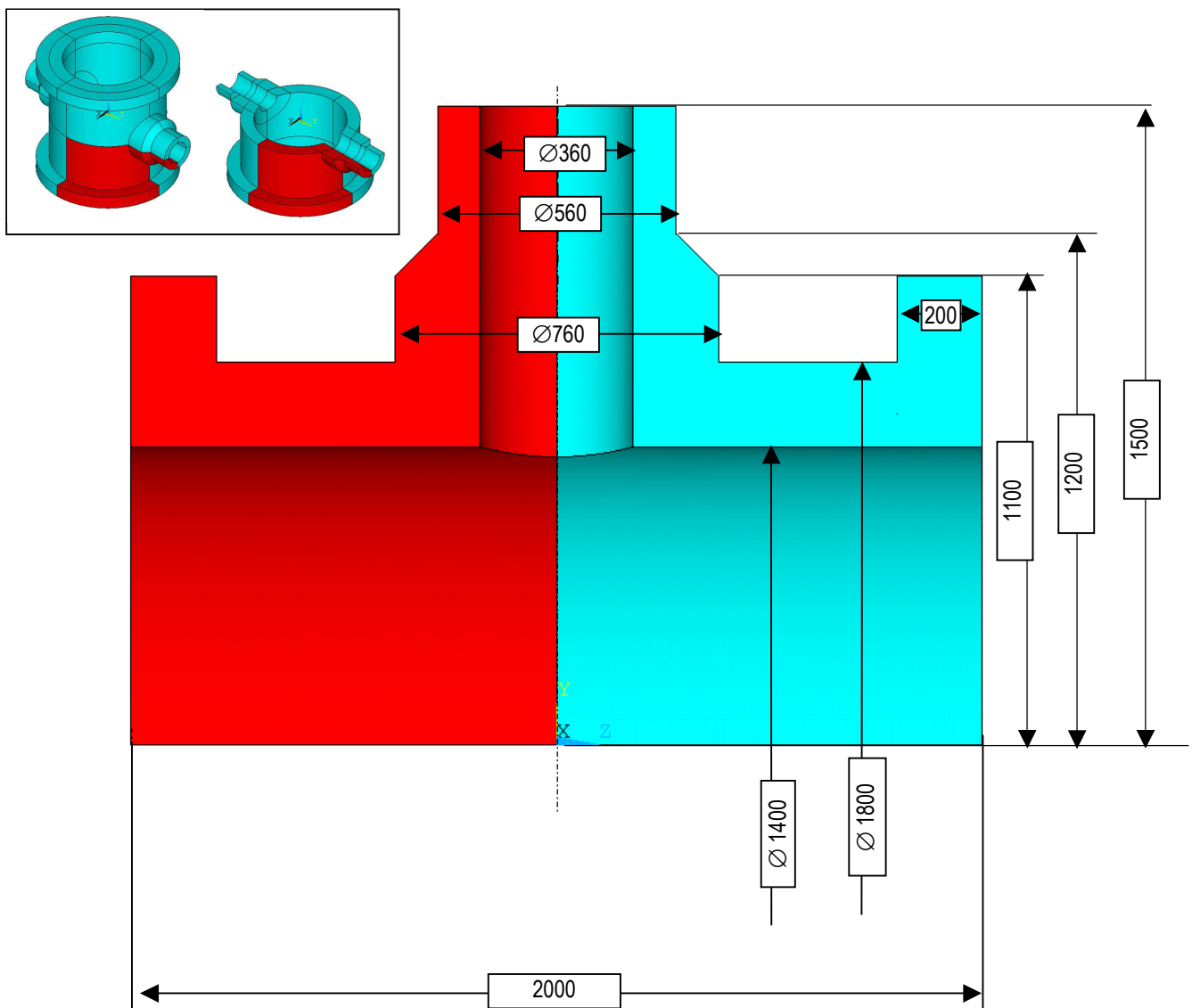
Three-dimensional problem of the theory of elasticity includes an elastic body with defined kinematic or static boundary conditions and the mass forces acting inside. The analytical solution is known only for simple cases. In general, numerical methods are the only way to solve such tasks. Numerical solution of the problem by using FEM requires a three-dimensional spatial discretization with a solid three-dimensional finite elements.

2. PROBLEM DESCRIPTION

The goal of analysis is to determine stress distribution inside a pressure vessel made of steel which is a part of hydraulic installation. The vessel is loaded with internal pressure p . The vessel is attached by two flanges. The other two nozzles are free of displacements.

Data: $p=50\text{MPa}$, $E=2\cdot 10^5\text{MPa}$, $\nu=0.3$

Geometric data (in millimeters) are presented below:

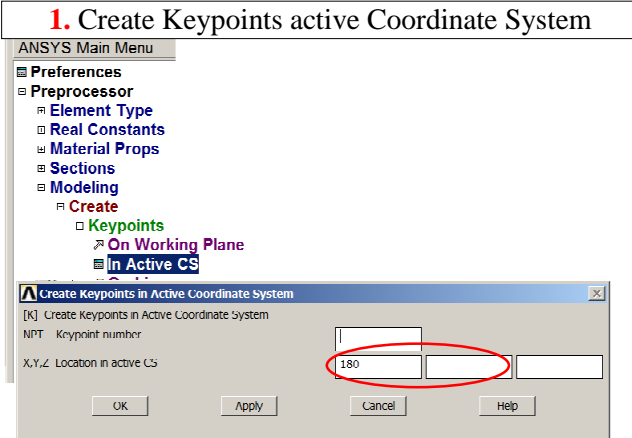


3. TYPICAL COURSE OF NUMERICAL ANALYSIS

Taking into consideration the triple symmetry (xz, yz and zx planes), the model includes only 1/8 part of the vessel. Convenient units are: *mm, N and MPa*.

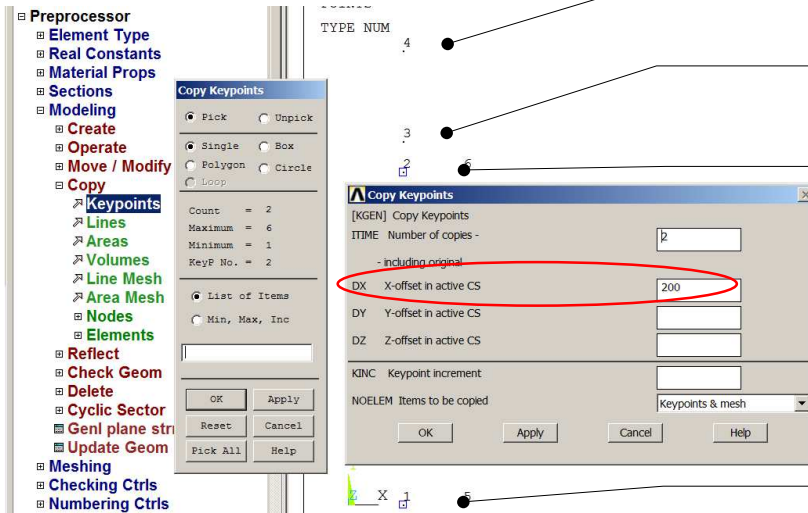
3.1. Preprocessor

1. Create Keypoints active Coordinate System



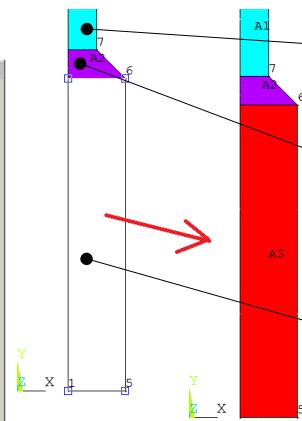
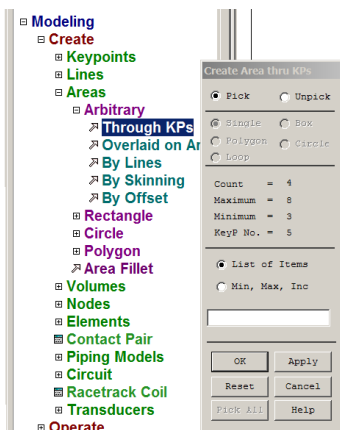
- 4 Preprocessor> Modeling>Create>Keypoint> In Active CS: X=180, Y=1500
- 3 Preprocessor> Modeling>Create>Keypoint> In Active CS: X=180, Y=1200
- Preprocessor> Modeling>Create>Keypoint> In Active CS: X=180, Y=1100
- Preprocessor> Modeling>Create>Keypoint> In Active CS: X=180, Y=0

2. Copy Keypoints on X direction (right)



- Preprocessor>Modeling> Copy>Keypoint No 4: DX=100
- Preprocessor>Modeling> Copy>Keypoint No 3: DX=100
- Preprocessor>Modeling> Copy>Keypoint No 2: DX=200
- Preprocessor>Modeling> Copy>Keypoint No 1: DX=200

3. Create areas through Keypoints



- Preprocessor>Modeling> Create>Areas>Arbitrary> Through KPs: 3,4,8,7
- Preprocessor>Modeling> Create>Areas>Arbitrary> Through KPs: 2,3,7,6
- Preprocessor>Modeling> Create>Areas>Arbitrary> Through KPs: 1,2,6,5

4. Create Keypoints in active Coordinate System (on axis of revolution - Y)

Preprocessor> Modeling>Create>Keypoint> In Active CS: X=0, Y=200

Preprocessor> Modeling>Create>Keypoint> In Active CS: X=0, Y=0

5. Extrude areas about axis defined by two Keypoints:
Preprocessor>Modeling>Operate> Extrude>Areas>About Axis

The image shows the ANSYS Main Menu on the left with the 'About Axis' option highlighted under 'Extrude' > 'Areas'. A 3D model of a red vertical shaft with a blue cylindrical top is shown. A coordinate system (X, Y, Z) is visible at the base. Two red circles with numbers '1' and '2' indicate the locations of keypoint creation on the Y-axis. A dialog box titled 'Sweep Areas about Axis' is open, showing 'ARC' set to 90 degrees and 'NSCG' set to 1. A red arrow points from the shaft to the resulting 3D model of a cylinder with a flange.

6. Create 1/4 of cylinder:

The image shows the ANSYS Main Menu with 'Cylinder' selected under 'Volumes'. A 3D model of a red shaft with a blue cylindrical top and a blue quarter-cylinder flange is shown. A dialog box titled 'Create Cylinder by Dimensions' is open, showing 'RAD1' set to 900, 'RAD2' set to 700, 'Z1,Z2' set to -1000 and 0, 'THETA1' set to 0, and 'THETA2' set to 90. Red circles highlight these input fields.

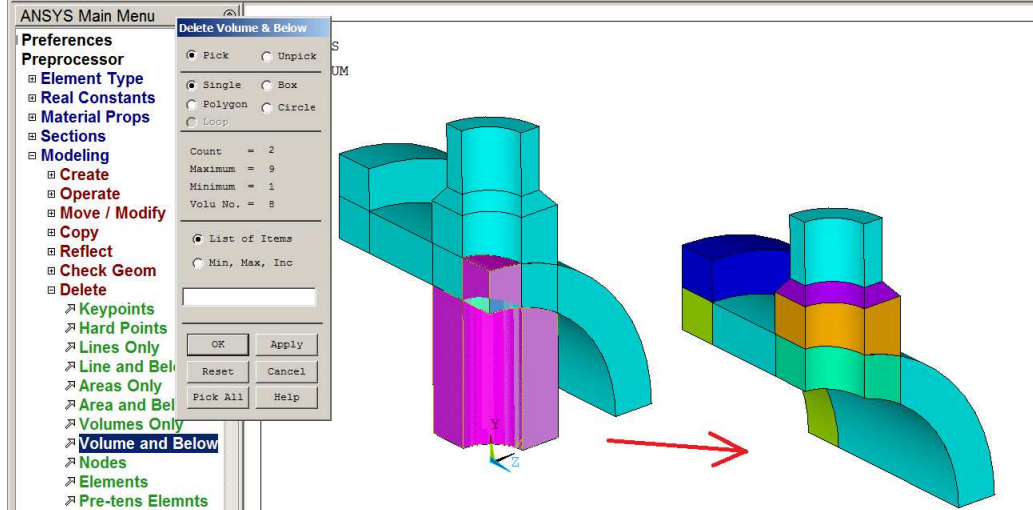
7. Create 1/4 of cylinder for flange:

The image shows the ANSYS Main Menu with 'Cylinder' selected under 'Volumes'. A 3D model of a red shaft with a blue cylindrical top and a blue quarter-cylinder flange is shown. A dialog box titled 'Create Cylinder by Dimensions' is open, showing 'RAD1' set to 1100, 'RAD2' set to 700, 'Z1,Z2' set to -1000 and -1000+200, 'THETA1' set to 0, and 'THETA2' set to 90. Red circles highlight these input fields.

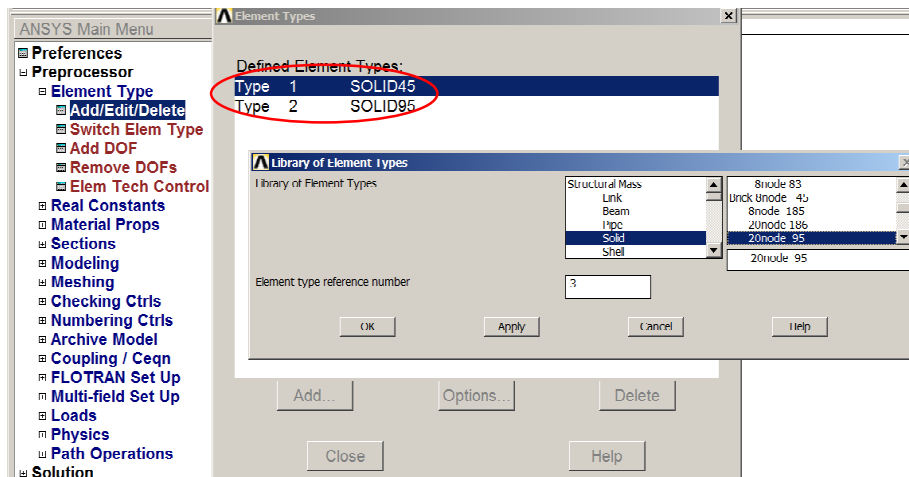
8. Overlap Volumes: Preprocessor>Modeling>Operate> Booleans>Overlap>Volumes: All

The image shows the ANSYS Main Menu with 'Overlap Volumes' selected under 'Booleans'. A 3D model of a red shaft with a blue cylindrical top and a blue quarter-cylinder flange is shown. A dialog box titled 'Overlap Volumes' is open, showing 'Pick All' highlighted with a red circle. A red arrow points from the initial model to the resulting model where the volumes are overlapped.

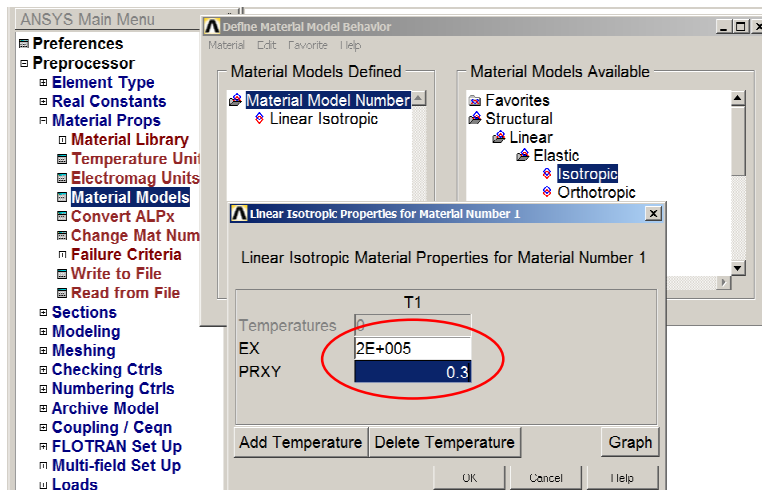
9. Delete unnecessary Volumes: **Preprocessor>Modeling>Delete> Volumes and Below**



10. Select Element types: **Preprocessor>Element Type>Add> (SOLID45 and SOLID95)**



11. Define Material Properties: **Preprocessor>Material Props>Material Models: Structural/Linear/Elastic/Isotropic: $EX=2e5MPa$, $PRXY=0.3$**



12. Define global element size:
Preprocessor>Meshing> Meshing Tool> Size Controls>Global

13. Mesh Volumes: Preprocessor>Meshing> Meshing Tool> Mesh>Volumes/Hex/Sweep

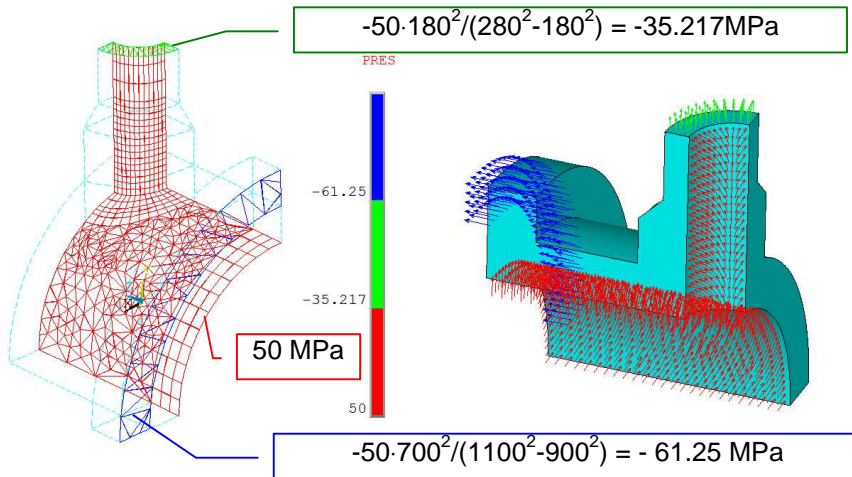
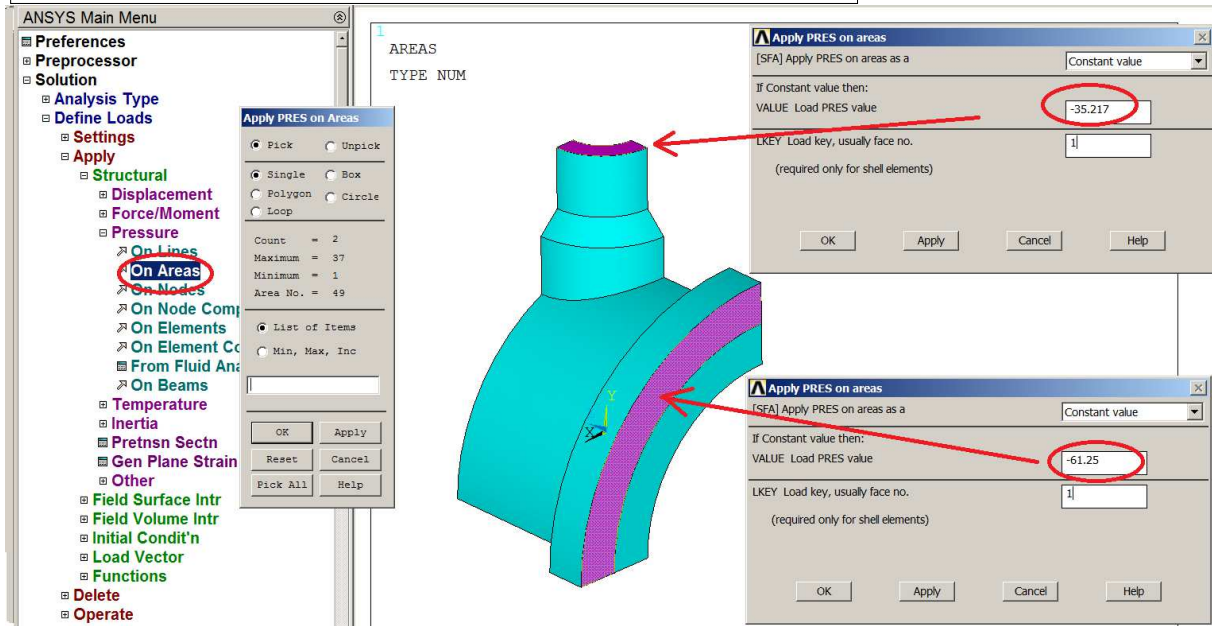
3.2. Solution

Define boundary conditions:

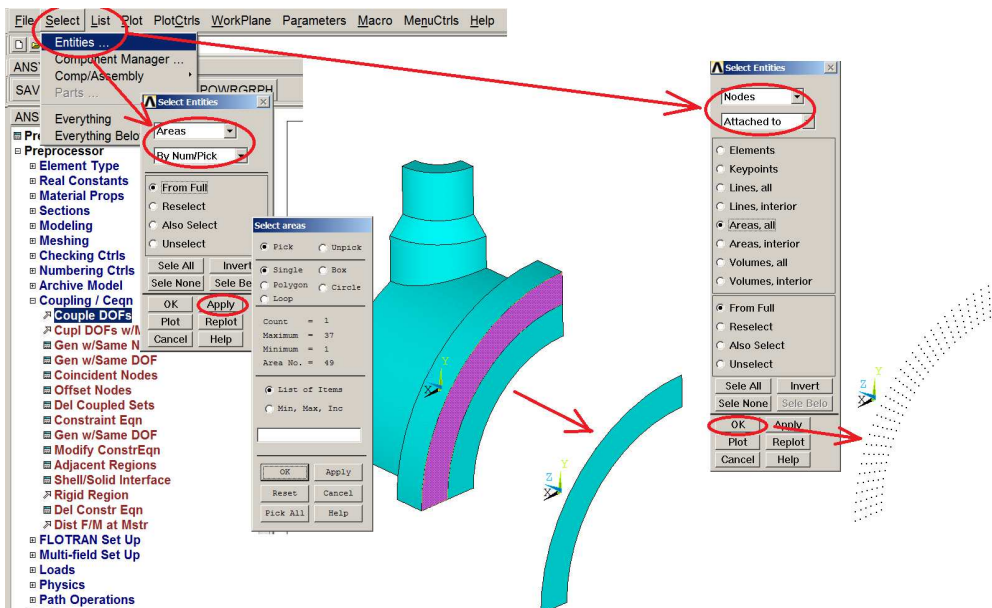
14. Define Symmetry B.C. on Areas:
Solution>Define Loads> Apply>Structural>Displacement> Symmetry BC>On Areas

15. Define pressure on internal Areas:
Solution>Define Loads> Apply>Structural>Pressure>On Areas

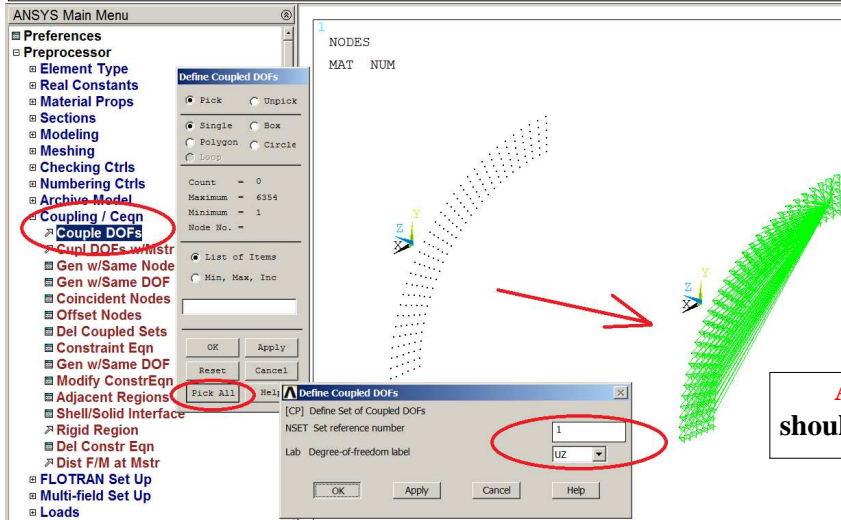
16. Define negative pressure on nozzle and flange areas:
Solution>Define Loads> Apply>Structural>Pressure>On Areas



17. Select nodes on the sticking surface of the flange:

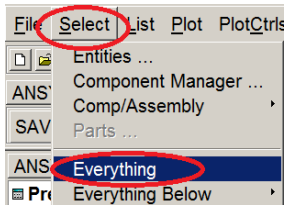


**18. Couple DOFs (UZ) on the sticking surface of the flange:
Preprocessor>Coupling / Ceqn> Couple DOFs**



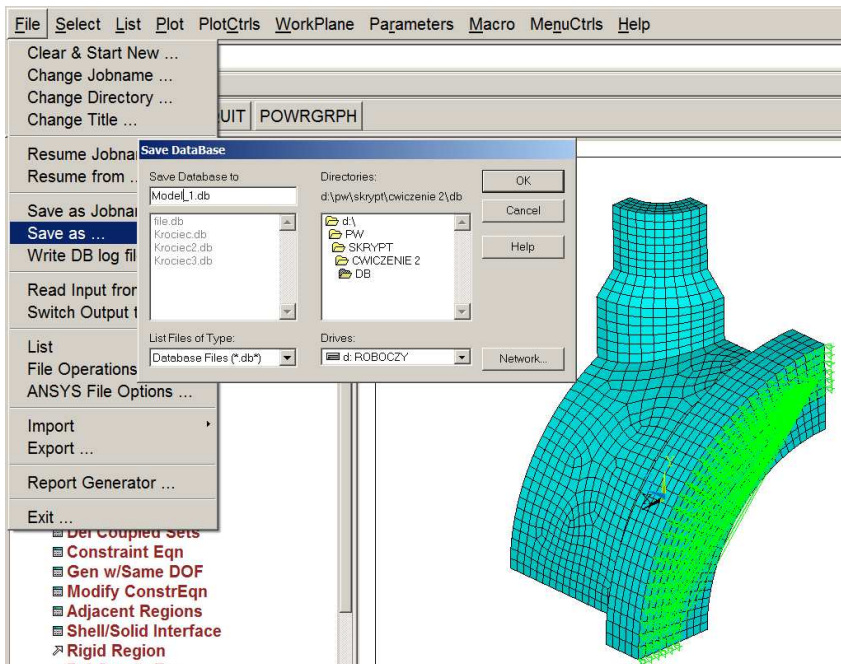
ATTENTION: Steps 17 and 18 should be performed for each model !!!

19. Select all entities:



20. Solve linear problem: Solution>Solve>Current LS

21. Save database with a unique name: *Model_1.db*

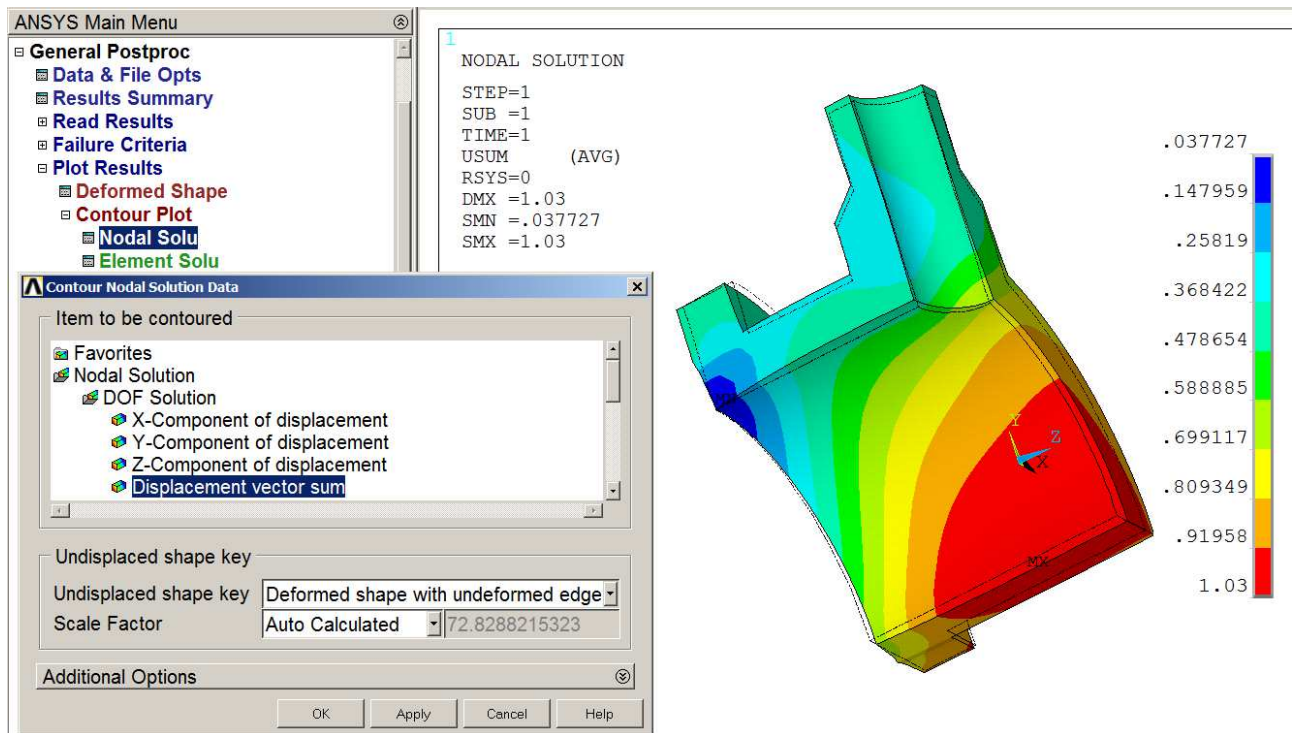


3.3. General postprocessor

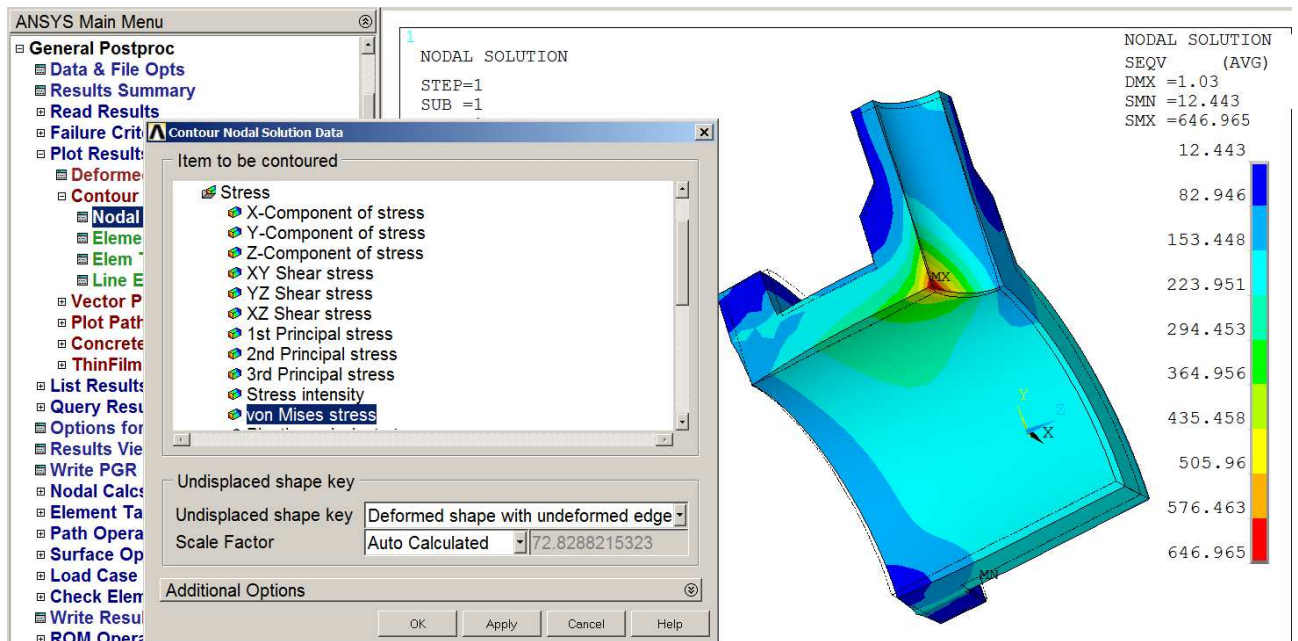
Show the results as contour maps:

Show total displacements (USUM), Von Mises stress (SEQV) and stress components (SX, SY) in global cylindrical system related to cylindrical part of the model.

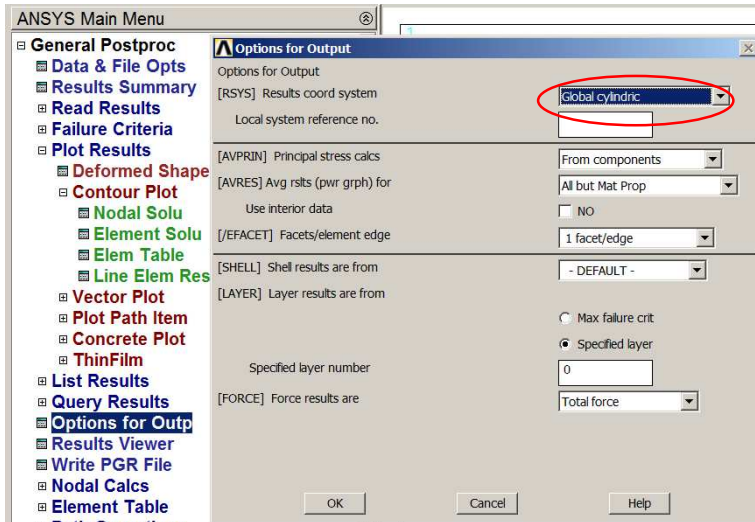
22. Plot Total displacements (USUM)



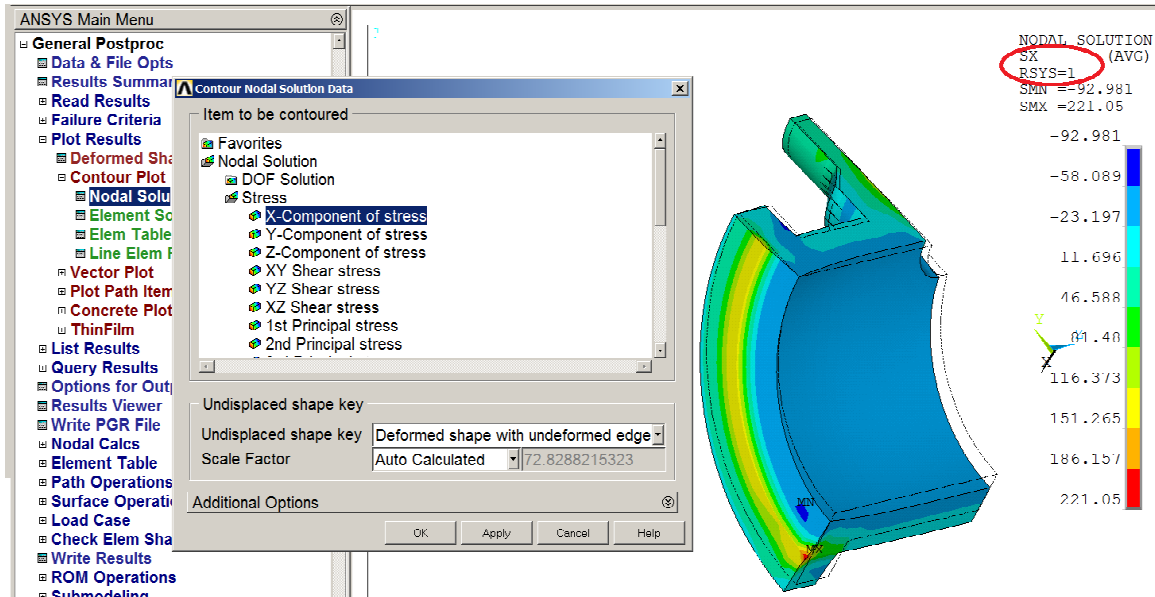
23. Plot Von Mises stress (SEQV)



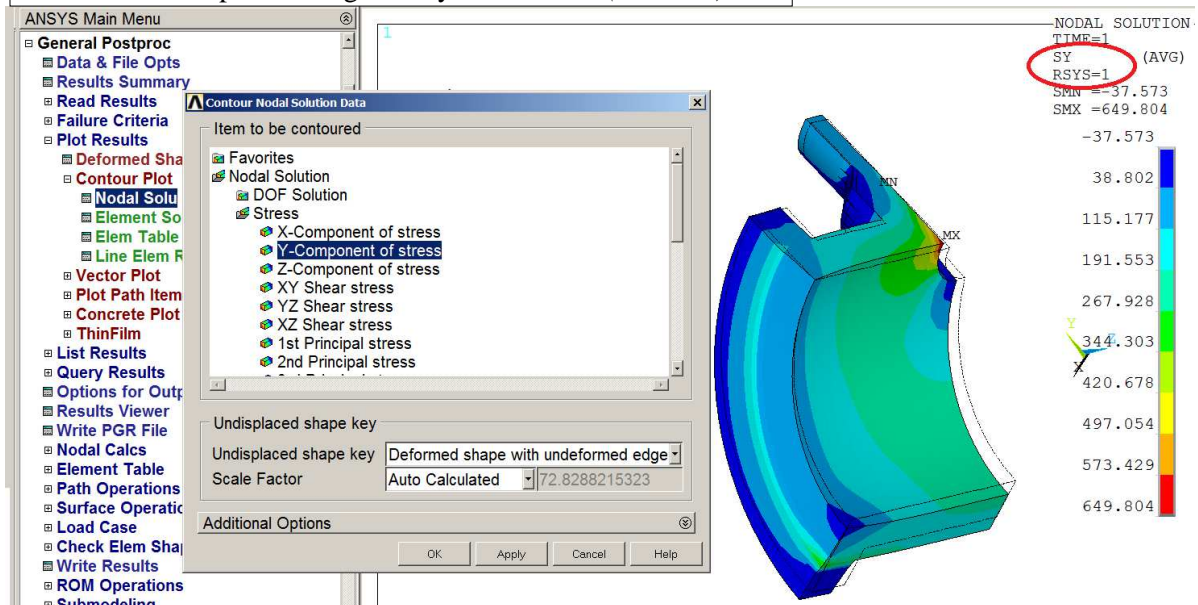
24. Select global cylindrical CS for results presentation:



25. Plot radial stresses in global cylindrical CS ($RSYS=1$)



26. Plot hoop stress in global cylindrical CS ($RSYS=1$)



27. Define path AB along wall thickness and map radial and hoop stresses on it:

ANSYS Main Menu

- General Postproc
 - Data & File Opts
 - Results Summary
 - Read Results
 - Failure Criteria
 - Plot Results
 - List Results
 - Query Results
 - Options for Outp
 - Results Viewer
 - Write PGR File
 - Nodal Calcs
 - Element Table
 - Path Operations
 - Define Path
 - By Nodes
 - On Working Plane
 - By Location
 - Path Status
 - Modify Path
 - Path Options
 - Delete Path
 - Plot Paths
 - Recall Path
 - Map onto Path
 - Plot Path Item
 - Linearized Strs

Map Result Items onto Path

Item,Comp Item to be mapped

Stress

X-direction SX

Y-direction SY

Z-direction SZ

XY-shear SXY

XZ-shear SXZ

X-direction SX

OK Apply Cancel Help

28. Plot graphs of radial and hoop stresses along path AB:

ANSYS Main Menu

- General Postproc
 - Data & File Opts
 - Results Summary
 - Read Results
 - Failure Criteria
 - Plot Results
 - List Results
 - Query Results
 - Options for Outp
 - Results Viewer
 - Write PGR File
 - Nodal Calcs
 - Element Table
 - Path Operations
 - Define Path
 - Delete Path
 - Plot Paths
 - Recall Path
 - Map onto Path
 - Plot Path Item
 - On Graph
 - On Geometry
 - List Path Items
 - Path Range
 - Linearized Strs
 - List Linearized
 - Add

Plot of Path Items on Graph

List Path Items

SX

SY

SEQV

OK Apply Cancel Help

POST1

STEP=1

SUB =1

TIME=1

PATH PLOT

NOD1=1880

NOD2=073

SX

SY

242.75

214.466

186.183

157.9

129.617

101.334

73.051

44.766

16.405

-11.790

-40.001

0 20 40 60 80 100 120 140 160 180 200

DTST

29. List radial and hoop stresses along path AB:

ANSYS Main Menu

- General Postproc
 - Data & File Opts
 - Results Summary
 - Read Results
 - Failure Criteria
 - Plot Results
 - List Results
 - Query Results
 - Options for Outp
 - Results Viewer
 - Write PGR File
 - Nodal Calcs
 - Element Table
 - Path Operations
 - Define Path
 - Delete Path
 - Plot Paths
 - Recall Path
 - Map onto Path
 - Plot Path Item
 - List Path Items
 - Path Range
 - Linearized Strs
 - List Linearized
 - Add
 - Multiply
 - Divide

PRPATH Command

File

PRINT ALONG PATH DEFINED BY LPATH COMMAND. DSVS= 0

***** PATH VARIABLE SUMMARY *****

S	SX	SY
0.0000	0.000000	0.000000
10.0000	186.183000	214.466000
20.0000	186.183000	214.466000
30.0000	186.183000	214.466000
40.0000	186.183000	214.466000
50.0000	186.183000	214.466000
60.0000	186.183000	214.466000
70.0000	186.183000	214.466000
80.0000	186.183000	214.466000
90.0000	186.183000	214.466000
100.0000	186.183000	214.466000
110.0000	186.183000	214.466000
120.0000	186.183000	214.466000
130.0000	186.183000	214.466000
140.0000	186.183000	214.466000
150.0000	186.183000	214.466000
160.0000	186.183000	214.466000
170.0000	186.183000	214.466000
180.0000	186.183000	214.466000
190.0000	186.183000	214.466000
200.0000	186.183000	214.466000

POST1

STEP=1

SUB =1

TIME=1

PATH PLOT

NOD1=1880

NOD2=073

SX

SY

242.75

214.466

186.183

157.9

129.617

101.334

73.051

44.766

16.405

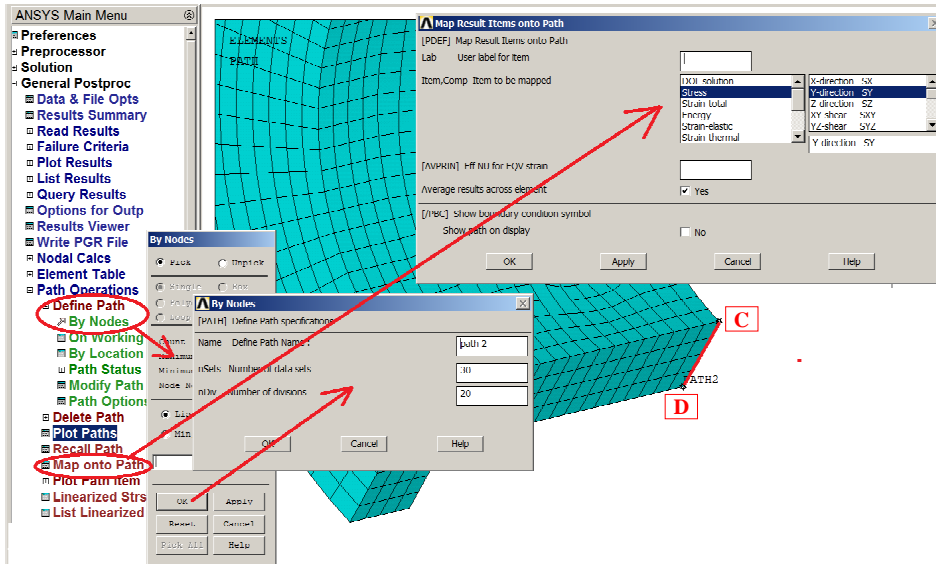
-11.790

-40.001

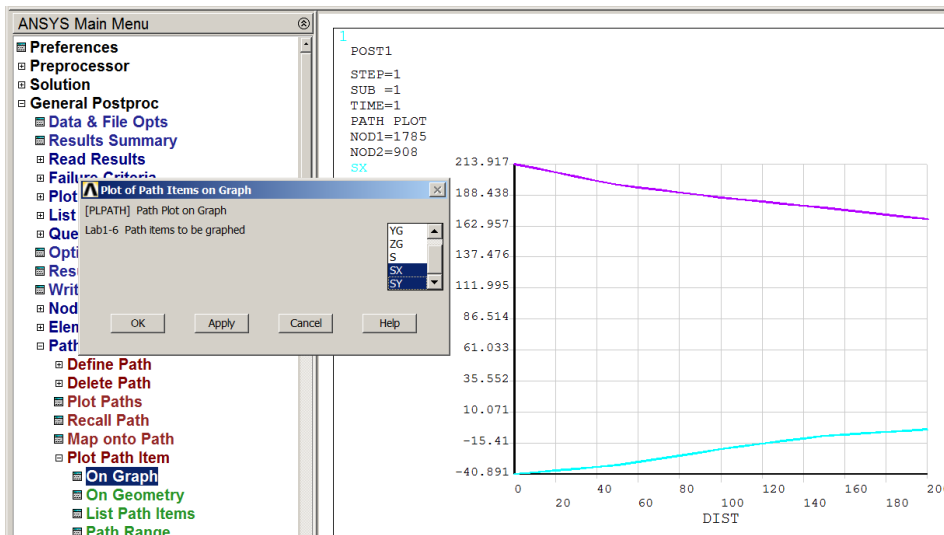
0 20 40 60 80 100 120 140 160 180 200

DTST

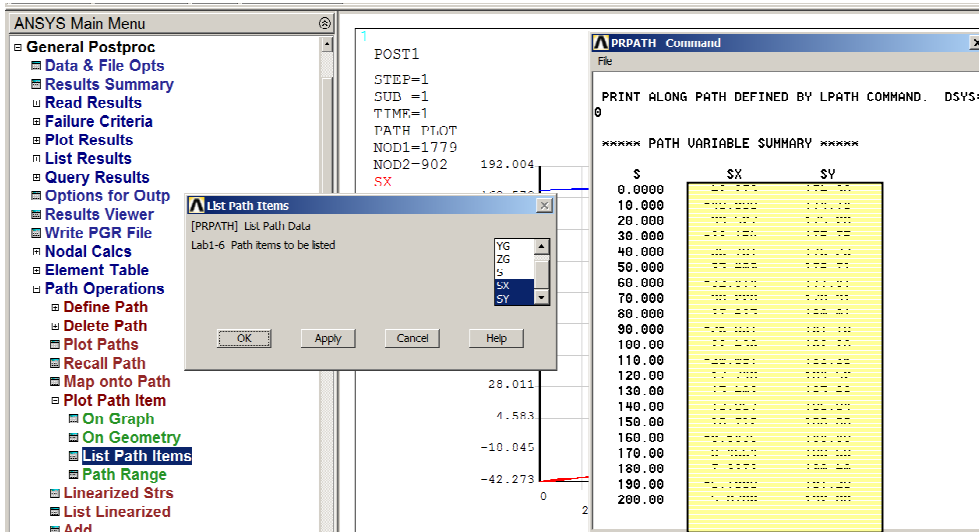
30. Define path CD along wall thickness and map radial and hoop stresses on it:

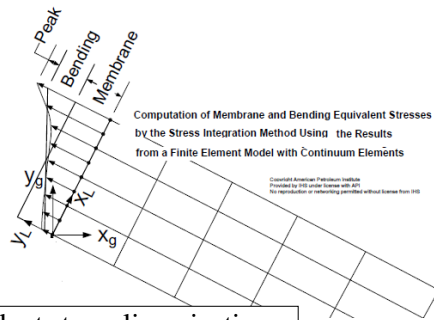
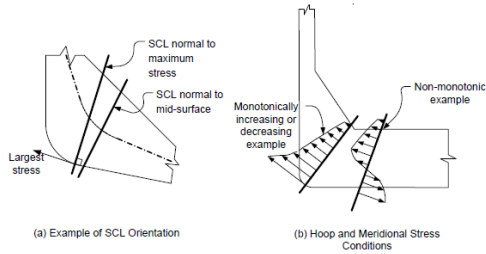


31. Plot graphs of radial and hoop stresses along path CD:



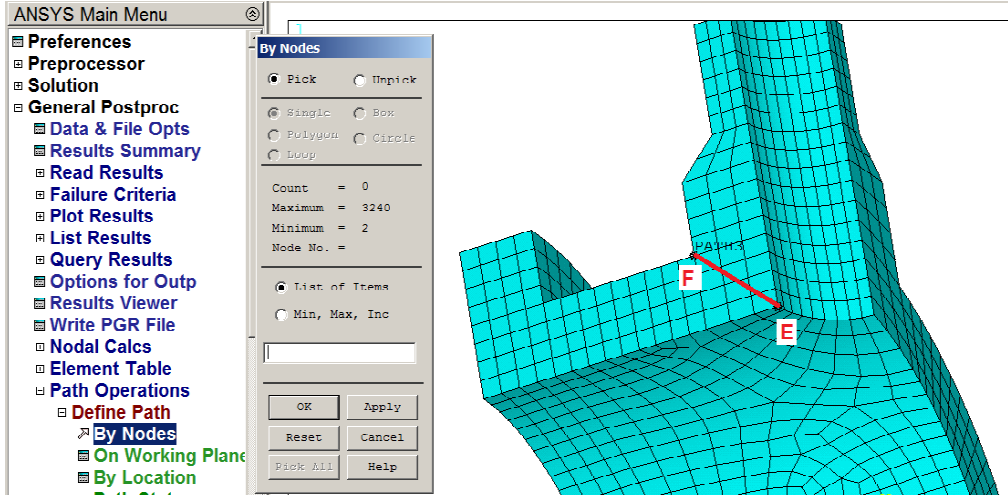
32. List radial and hoop stresses along path CD:



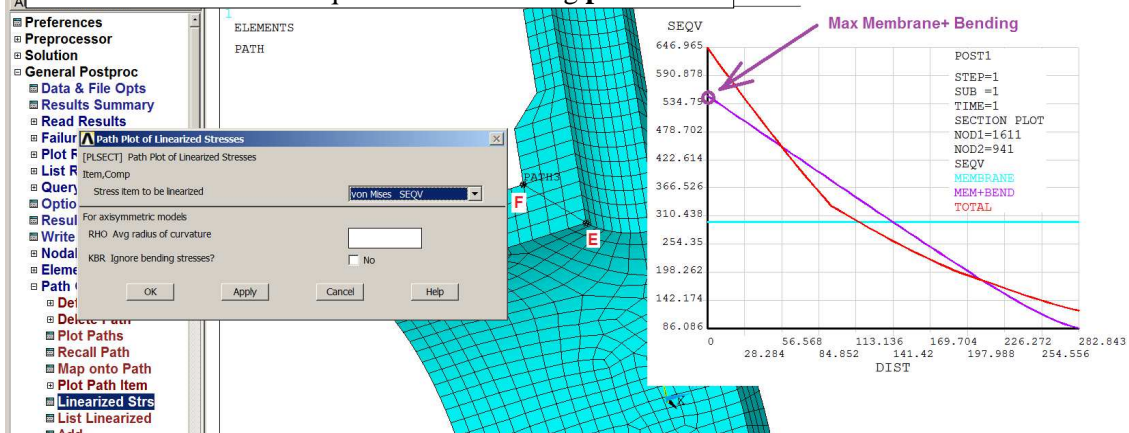


Stress Classification Line Orientation and Validity Guidelines

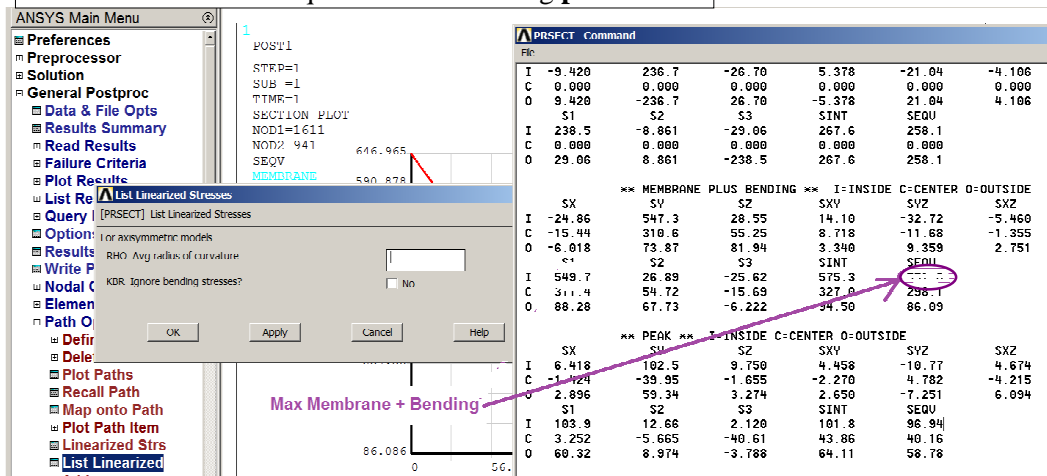
33. Define path EF along wall thickness for equivalent stress linearization:



34. Linearization of equivalent stress along path EF



35. List linearized equivalent stress along path EF



4. INTERPRETATION OF THE RESULTS. TASKS TO BE DONE

Compare results of the models built with the same mesh density (ESIZE parameter see p.12) using:

- 8-noded elements (Solid45) using 'sweeping' HEX/WEDGE option (**Model 1**),
- 20-noded elements (Solid95) using 'sweeping' HEX/WEDGE option (**Model 2**),
- 8-noded elements (Solid45) using 'free meshing' TETRA option (**Model 3**).

Put the results in the **table** for each model:

No. of nodes, No. of elements, $USUM_{max}$, $SEQV_{max}$, $SX_{RSYS=1}$, $SY_{RSYS=1}$ for points: A,B,C i D and maximum Membrane and Bending SEQV stress on path EF (step 35).

Discuss the results.

	Model 1 Solid 45 Hex/Wed	Model 2 Solid 95 Hex/Wed	Model 3 Solid 45 Free	
No. of nodes				<p>Plots needed (should be archived during program session for each model) :</p> <ol style="list-style-type: none"> 1) FE mesh 2) USUM(x,y) 3) SEQV(x,y) 4) SX(x,y)_{RSYS=1} 5) SY(x,y)_{RSYS=1} 6) Graph: SX(x,y)_{RSYS=1} i SY(x,y)_{RSYS=1} on path AB 7) Graph: SX(x,y)_{RSYS=1} i SY(x,y)_{RSYS=1} on path CD 8) Graph of linearized SEQV on path EF <p>Report finalny:</p> <p>Final report:</p> <ol style="list-style-type: none"> 1) Introduction 2) Assumptions for the modeling 3) model description (solid model, mesh, boundary cond. and loads) 4) Results 5) Results in the Table 6) Discursion 7) Conclusion
No. of elements				
$USUM_{max}$				
$SEQV_{max}$				
$SX^A_{RSYS=1}$				
$SY^A_{RSYS=1}$				
$SX^B_{RSYS=1}$				
$SY^B_{RSYS=1}$				
$SX^C_{RSYS=1}$				
$SY^C_{RSYS=1}$				
$SX^D_{RSYS=1}$				
$SY^D_{RSYS=1}$				
Max Membrane + Bending stress				
<p>from Lamé theorem (for inside pressure):</p> $\sigma_r = \frac{p_a \cdot a^2}{b^2 - a^2} \cdot \left(1 - \frac{b^2}{r^2}\right) \quad \sigma_t = \frac{p_a \cdot a^2}{b^2 - a^2} \cdot \left(1 + \frac{b^2}{r^2}\right)$				
$\sigma_r(a) =$				
$\sigma_t(a) =$				
$\sigma_r(b) =$				
$\sigma_t(b) =$				