

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

1. INTRODUCTION

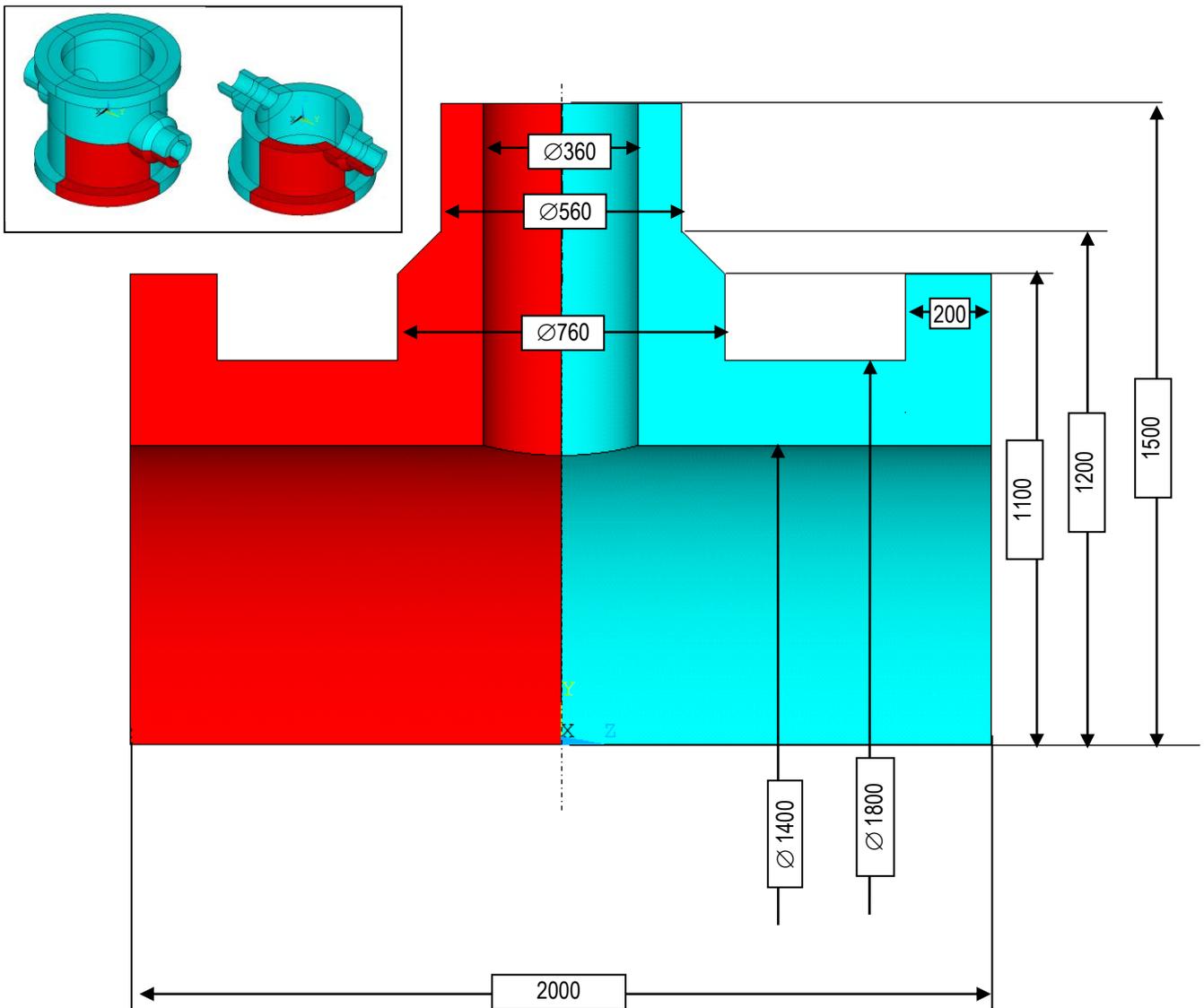
Three-dimensional problem of the theory of elasticity includes an elastic body with defined kinematic or static boundary conditions and the mass forces acting inside. The analytical solution is known only for simple cases. In general, numerical methods are the only way to solve such tasks. Numerical 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:

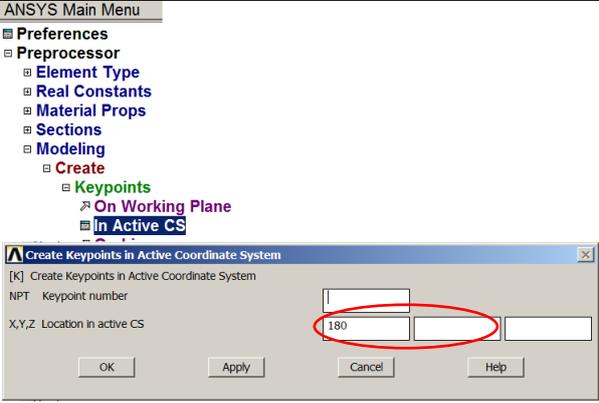


3. TYPICAL COURSE OF NUMERICAL ANALYSIS

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



ANSYS Main Menu
 Preferences
 Preprocessor
 Element Type
 Real Constants
 Material Props
 Sections
 Modeling
 Create
 Keypoints
 On Working Plane
 In Active CS

Create Keypoints in Active Coordinate System
 [K] Create Keypoints in Active Coordinate System
 NPT Keypoint number
 X,Y,Z Location in active CS
 180

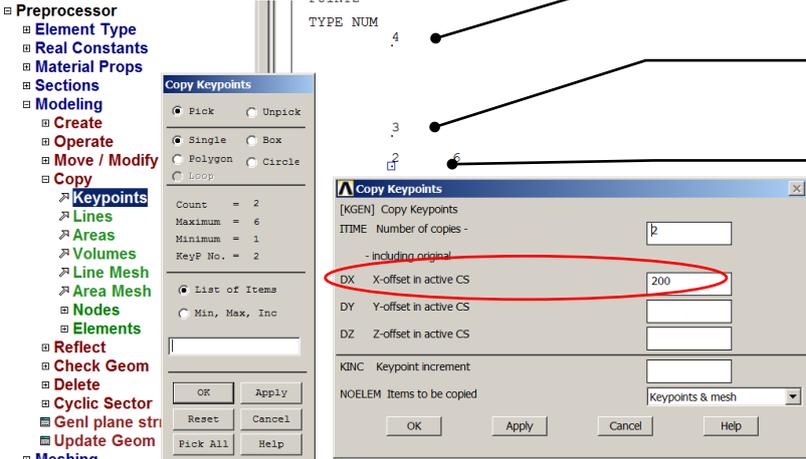
Preprocessor>
 Modeling>Create>Keypoint>
 In Active CS: X=180, Y=1500

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
 Element Type
 Real Constants
 Material Props
 Sections
 Modeling
 Create
 Operate
 Move / Modify
 Copy
 Keypoints
 Lines
 Areas
 Volumes
 Line Mesh
 Area Mesh
 Nodes
 Elements
 Reflect
 Check Geom
 Delete
 Cyclic Sector
 Genl plane str
 Update Geom
 Meshing
 Checking Ctrl
 Numbering Ctrl

Copy Keypoints
 Pick Unpick
 Single Box
 Polygon Circle
 Loop
 Count = 2
 Maximum = 6
 Minimum = 1
 KeyP No. = 2
 List of Items
 Min, Max, Inc

Copy Keypoints
 [KGEN] Copy Keypoints
 ITIME Number of copies -
 2
 - including original
 DX X-offset in active CS
 200
 DY Y-offset in active CS
 DZ Z-offset in active CS
 KINC Keypoint increment
 NOELEM Items to be copied
 Keypoints & mesh

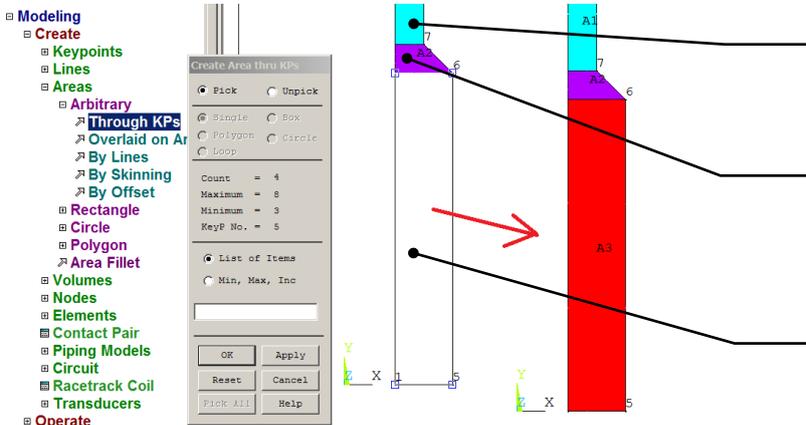
Preprocessor>Modeling>
 Copy>Keypoint No 4: DX=100

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



Modeling
 Create
 Keypoints
 Lines
 Areas
 Arbitrary
 Through KPs
 Overlaid on A
 By Lines
 By Skinning
 By Offset
 Rectangle
 Circle
 Polygon
 Area Fillet
 Volumes
 Nodes
 Elements
 Contact Pair
 Piping Models
 Circuit
 Racetrack Coil
 Transducers
 Operate

Create Area Thru KPs
 Pick Unpick
 Single Box
 Polygon Circle
 Loop
 Count = 4
 Maximum = 8
 Minimum = 3
 KeyP No. = 5
 List of Items
 Min, Max, Inc

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)

ANSYS Main Menu
 Preferences
 Preprocessor
 Element Type
 Real Constants
 Material Props
 Sections
 Modeling
 Create
 Operate
 Extrude
 Elem Ext Opt
 Areas
 Along Normal
 By XYZ Offse
 About Axis
 Along Lines
 Lines
 Keypoints
 Extend Line
 Booleans
 Scale
 Calc Geom Items
 Move / Modify
 Copy
 Reflect
 Check Geom
 Delete
 Cyclic Sector
 Genl plane strn
 Update Geom

1 AREAS
 AREA NUM

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

Sweep Areas about Axis
 Pick Unpick
 Single Box
 Polygon Circle
 Loop
 Count = 2
 Maximum = 2
 Minimum = 2
 KeyP No. = 10
 List of Items
 Min, Max, Inc

Sweep Areas about Axis
 [VROTAT] Sweep Areas about Axis
 ARC Arc length in degrees 90
 NSEG No. of volume segments

6. Create 1/4 of cylinder:

ANSYS Main Menu
 Preferences
 Preprocessor
 Element Type
 Real Constants
 Material Props
 Sections
 Modeling
 Create
 Keypoints
 Lines
 Areas
 Volumes
 Arbitrary
 Block
 Cylinder
 Solid Cylinder
 Hollow Cylinder
 Partial Cylinder
 By End Pts & Z
 By Dimensions
 Prism
 Sphere
 Cone
 Torus
 Nodes
 Elements
 Contact P
 Piping Mo
 Circuit
 Racetrack
 Transduc
 Operate

1 VOLUMES
 VOLU NUM

Create Cylinder by Dimensions
 [CYLIND] Create Cylinder by Dimensions
 RAD1 Outer radius 800
 RAD2 Optional inner radius 700
 Z1,Z2 Z-coordinates -1000 0
 THETA1 Starting angle (degrees) 0
 THETA2 Ending angle (degrees) 90

7. Create 1/4 of cylinder for flange:

ANSYS Main Menu
 references
 reprocessor
 Element Type
 Real Constants
 Material Props
 Sections
 Modeling
 Create
 Keypoints
 Lines
 Areas
 Volumes
 Arbitrary
 Block
 Cylinder
 Solid Cylinder
 Hollow Cylinder
 Partial Cylinder
 By End Pts & Z
 By Dimensions
 Prism
 Sphere
 Cone
 Torus
 Nodes
 Elements
 Contact Pair

1 VOLUMES
 VOLU NUM

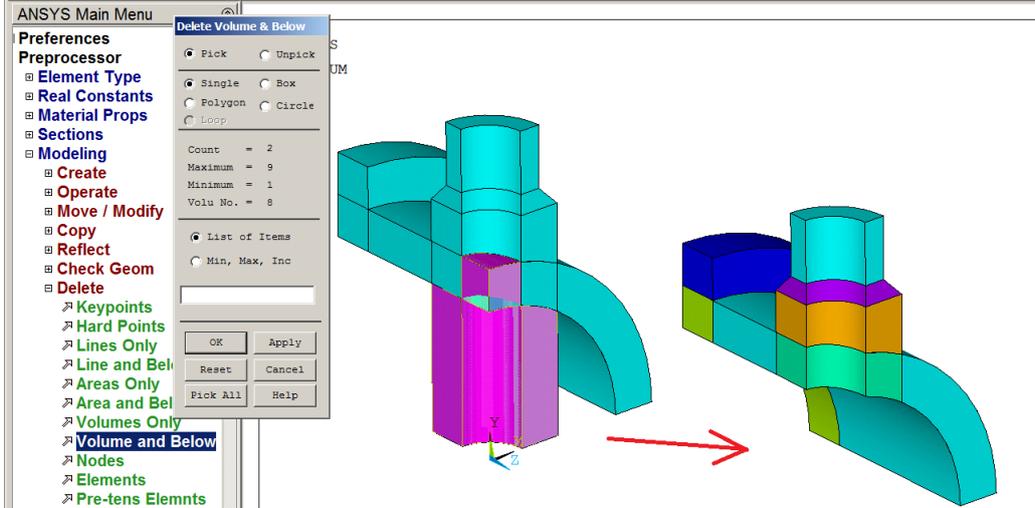
Create Cylinder by Dimensions
 [CYLIND] Create Cylinder by Dimensions
 RAD1 Outer radius 1100
 RAD2 Optional inner radius 700
 Z1,Z2 Z-coordinates -1000 -1000+200
 THETA1 Starting angle (degrees) 0
 THETA2 Ending angle (degrees) 90

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

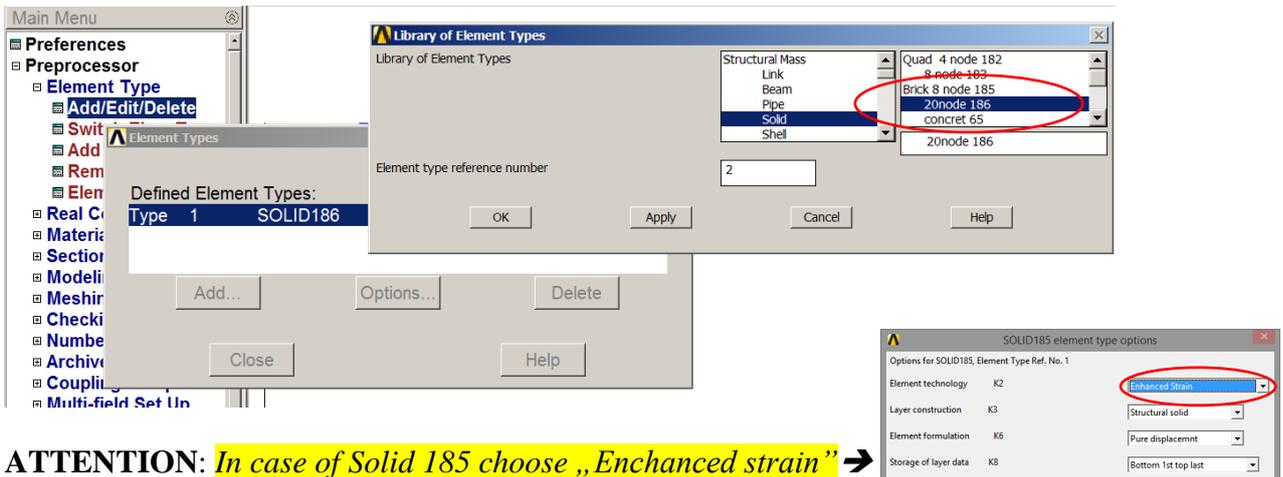
ANSYS Main Menu
 references
 reprocessor
 Element Type
 Real Constants
 Material Props
 Sections
 Modeling
 Create
 Operate
 Extrude
 Extend Line
 Booleans
 Intersect
 Add
 Subtract
 Divide
 Glue
 Overlap
 Volumes
 Areas
 Lines
 Partition
 Settings
 Show Degeneracy
 Scale
 Calc. Geom Items

Overlap Volumes
 Pick Unpick
 Single Box
 Polygon Circle
 Loop
 Count = 0
 Maximum = 5
 Minimum = 2
 Volu No. =
 List of Items
 Min, Max, Inc

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

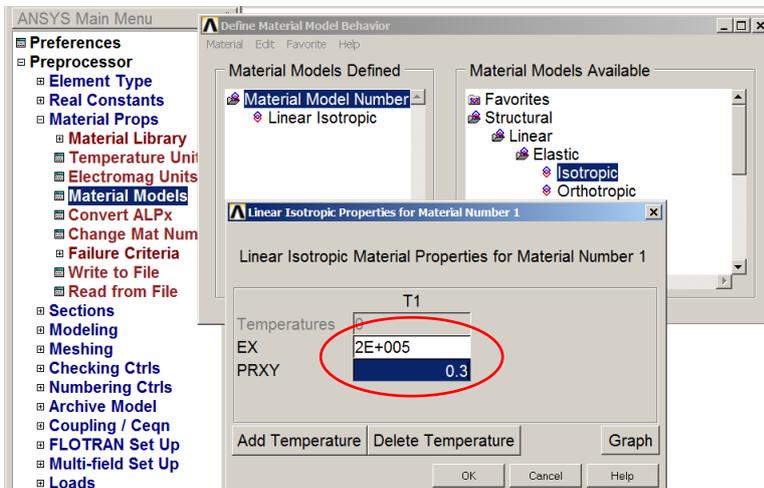


10. Select Element types: **Preprocessor>Element Type>Add> (SOLID186 or SOLID185)**



ATTENTION: In case of Solid 185 choose „Enhanced strain” →

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

The screenshot shows the ANSYS Main Menu on the left with the 'Meshing' path expanded. The 'Global Element Sizes' dialog box is open, showing 'Global' element size set to 50. The 'Mesh' button is circled in red. A secondary 'Volume Sweeping' dialog box is also open, with 'Pick All' circled in red. The background shows a 3D model of a mechanical part with a mesh.

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

3.2. Solution

Define boundary conditions:

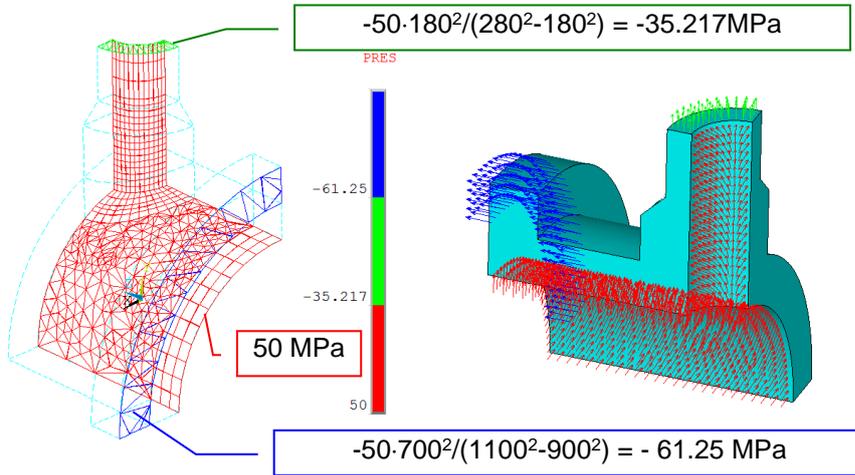
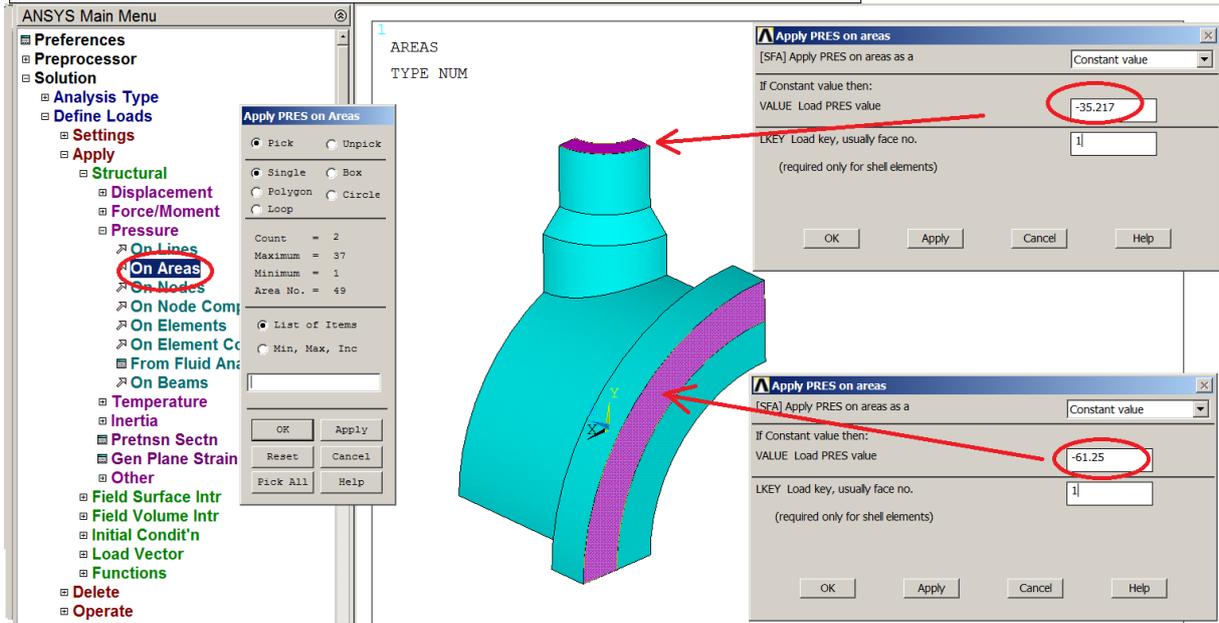
14. Define Symmetry B.C. on Areas:
Solution>Define Loads> Apply>Structural>Displacement> Symmetry BC>On Areas

The screenshot shows the 'Apply SYMM on Areas' dialog box. The 'On Areas' option is selected and circled in red. The background shows a 3D model of the mechanical part with symmetry boundaries highlighted in purple.

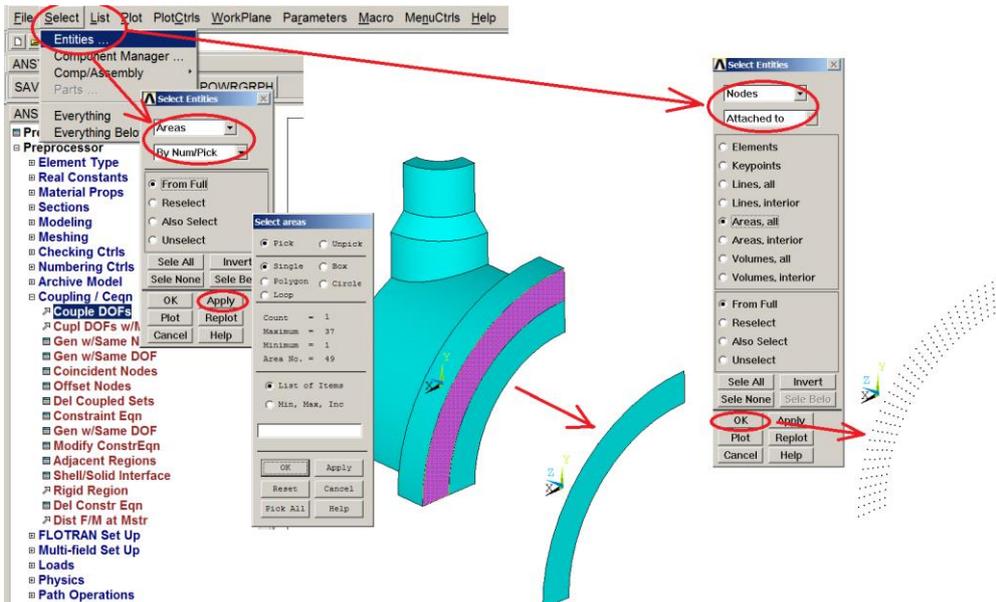
15. Define pressure on internal Areas:
Solution>Define Loads> Apply>Structural>Pressure>On Areas

The screenshot shows the 'Apply PRES on Areas' dialog box. The 'On Areas' option is selected and circled in red. The 'Constant value' is set to 50. The background shows a 3D model of the mechanical part with internal pressure applied to a surface, highlighted in pink.

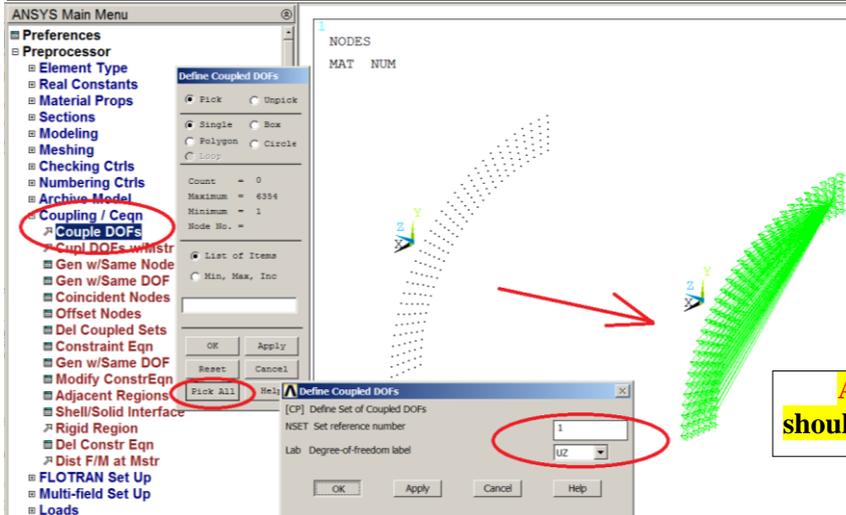
16. Define negative pressure on nozzle and flange areas:
Solution>Define Loads> Apply>Structural>Pressure>On Areas



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

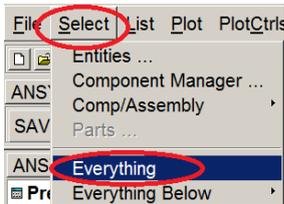


**18. Couple DOFs (UZ) on the sticking surface of the flange:
Preprocessor>Coupling / Ceqn> Couple DOFs**



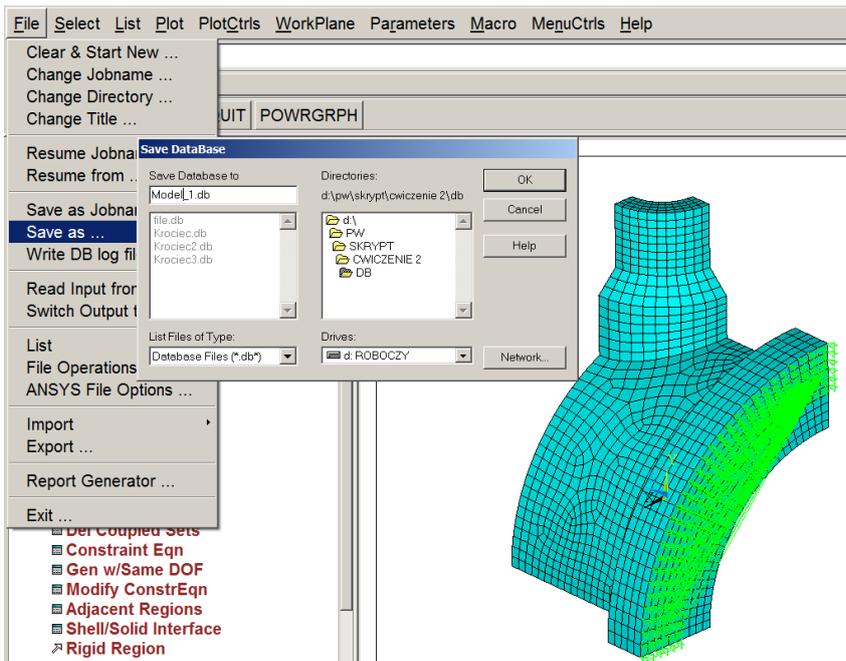
ATTENTION: Steps 17 and 18 should be performed for each model !!!

19. Select all entities:



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

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

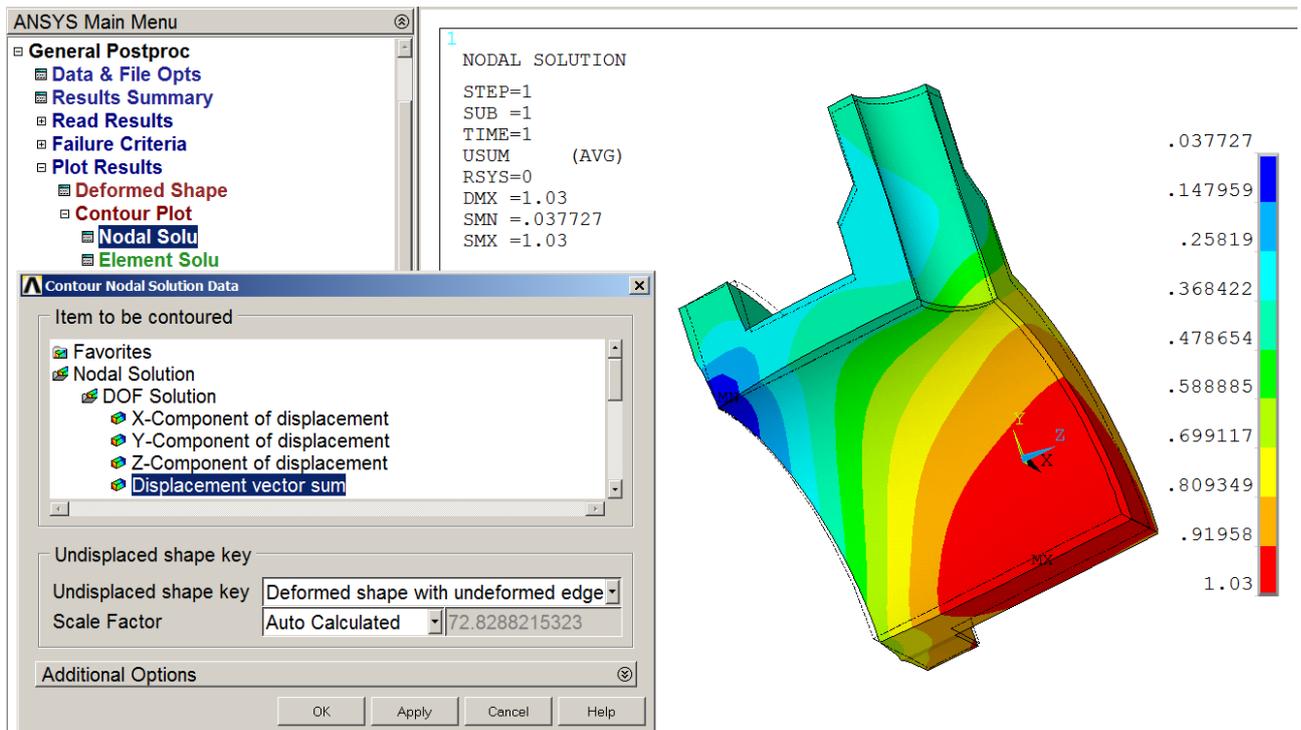


3.3. General postprocessor

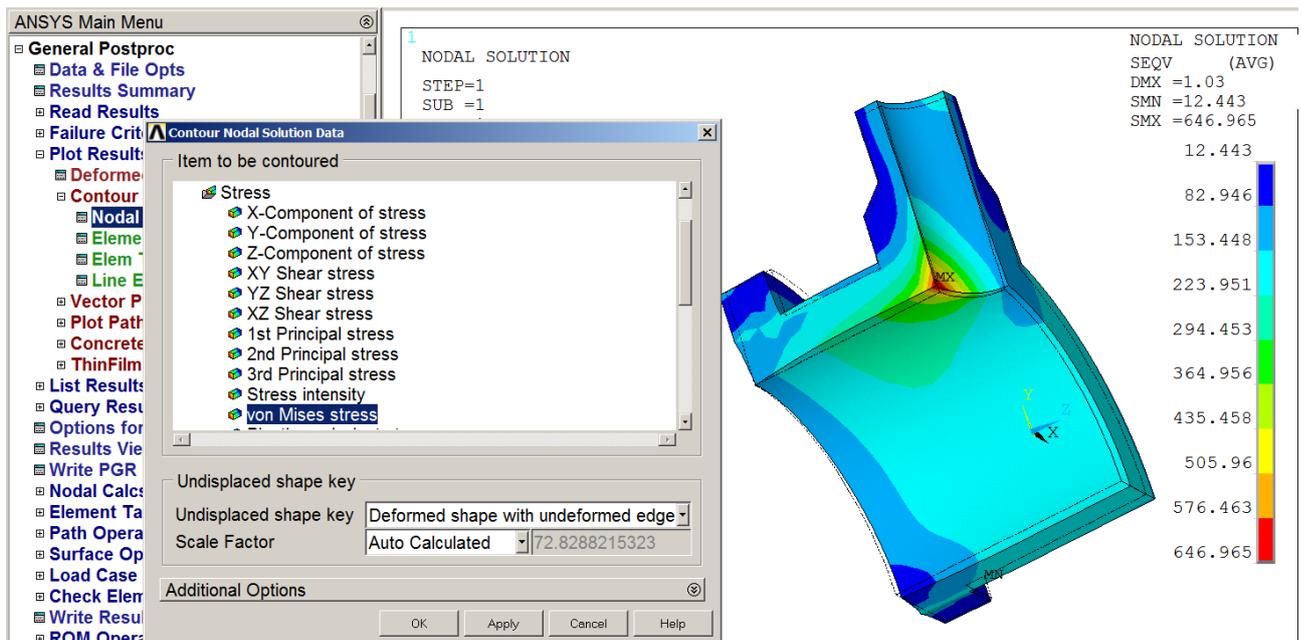
Show the results as contour maps:

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

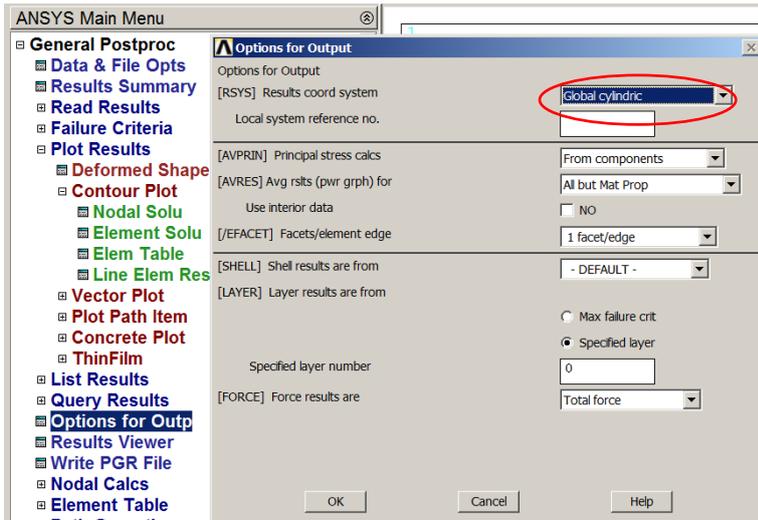
22. Plot Total displacements (USUM)



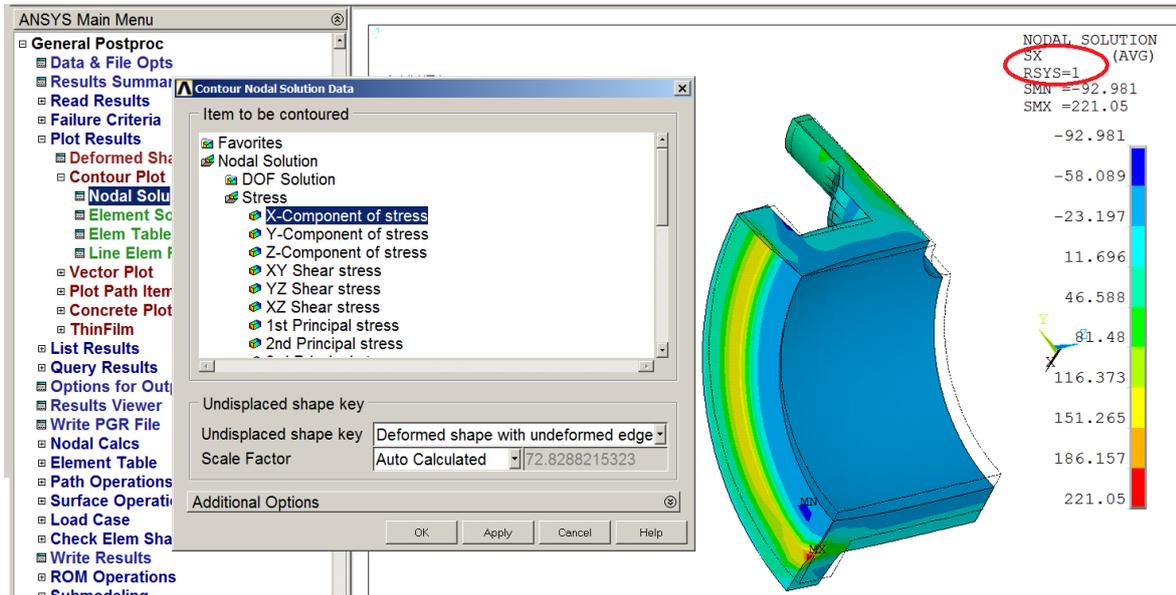
23. Plot Von Mises stress (SEQV)



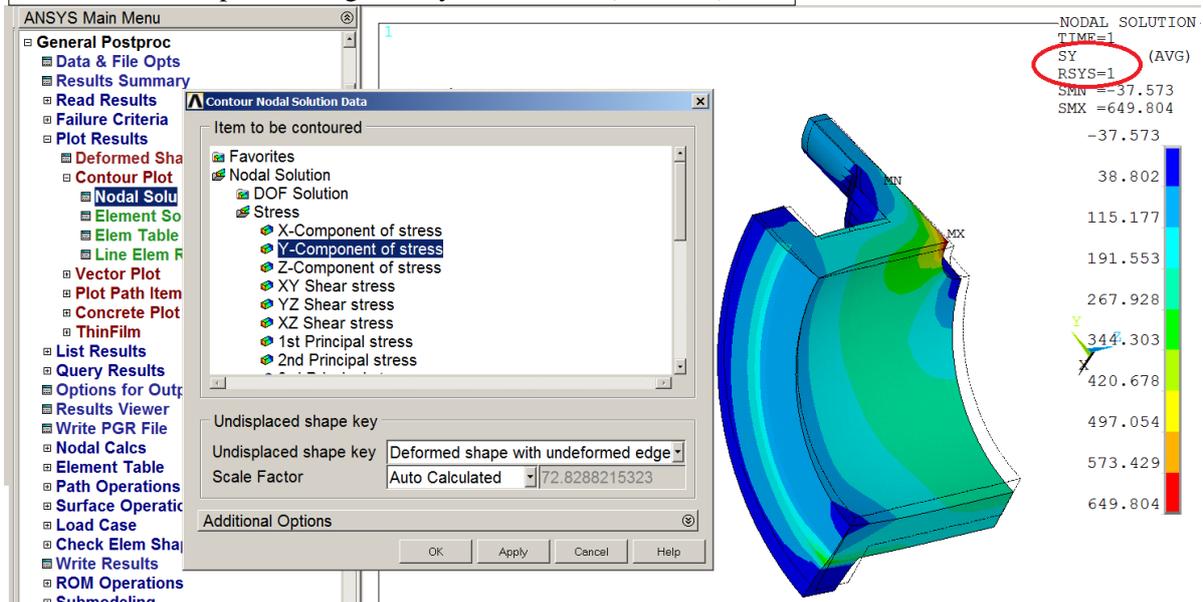
24. Select global cylindrical CS for results presentation:



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



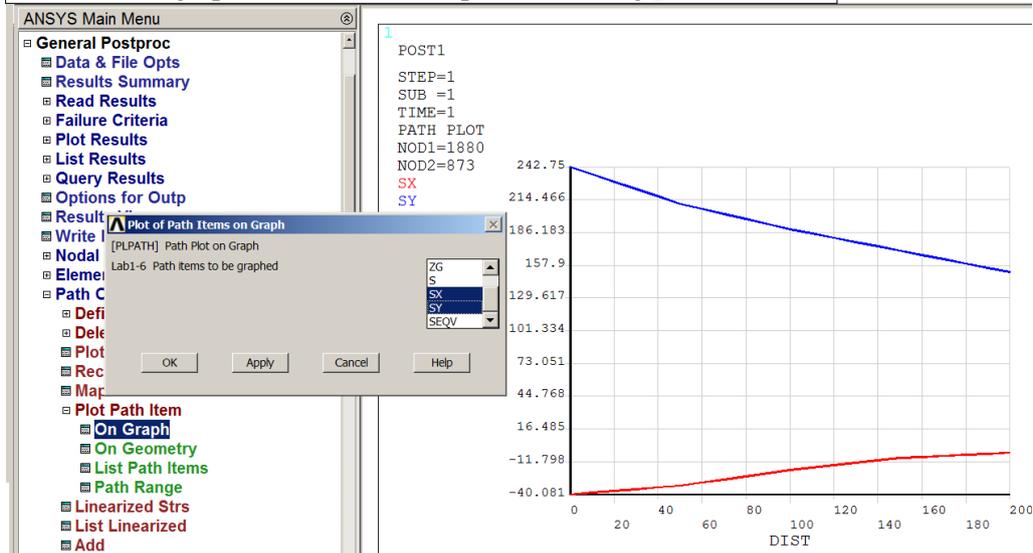
26. Plot hoop stress in global cylindrical CS ($RSYS=1$)



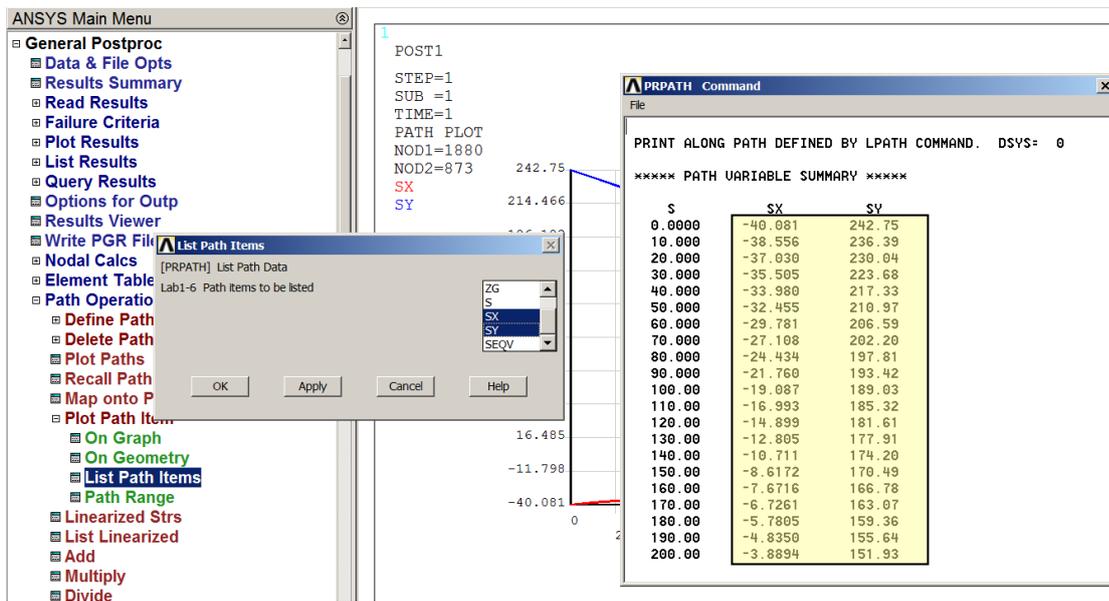
27. Define path AB along wall thickness and map radial and hoop stresses on it:

The screenshot shows the ANSYS Main Menu on the left. Under 'Path Operations', 'Define Path' is expanded, and 'By Nodes' is selected. The 'By Nodes' dialog box is open, showing 'Name: PATH1', 'nSets: 30', and 'nDiv: 20'. The 'Map Result Items onto Path' dialog is also open, showing 'Item,Comp: Stress' and 'Item to be mapped: Strain-total, Energy, Strain-elastic, Strain-thermal'. The 'X-direction' is set to 'SX' and 'Y-direction' to 'SY'. A red arrow points from the 'Map onto Path' option in the Main Menu to the 'Map Result Items onto Path' dialog.

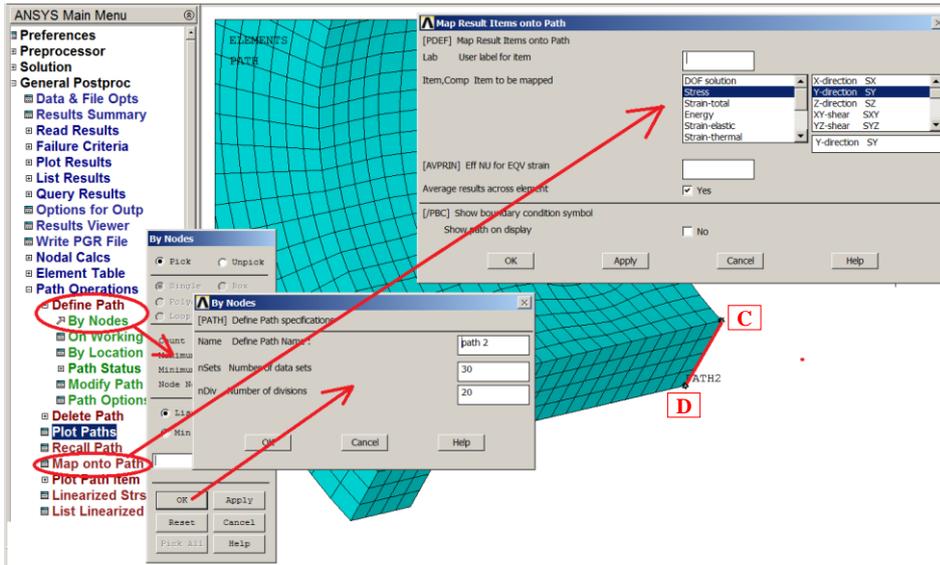
28. Plot graphs of radial and hoop stresses along path AB:



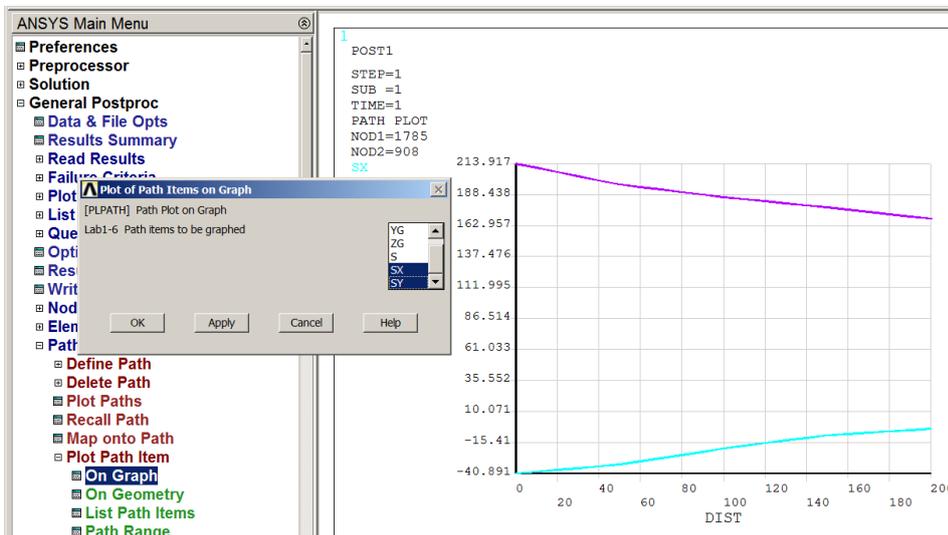
29. List radial and hoop stresses along path AB:



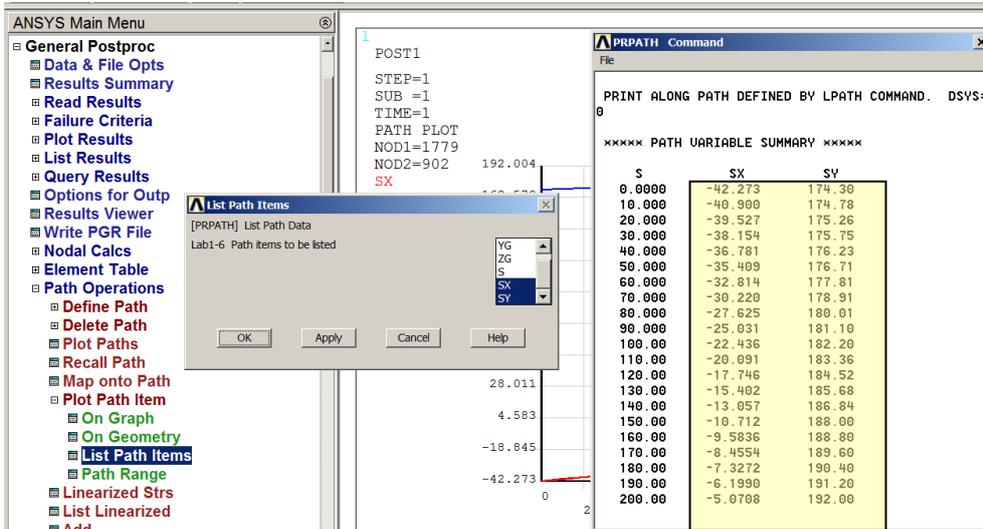
30. Define path CD along wall thickness and map radial and hoop stresses on it:

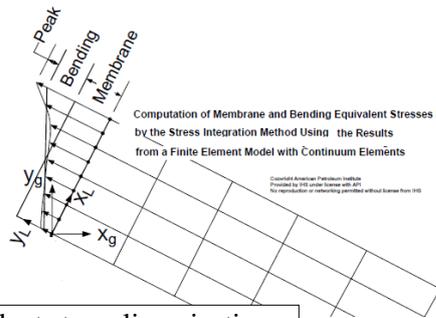
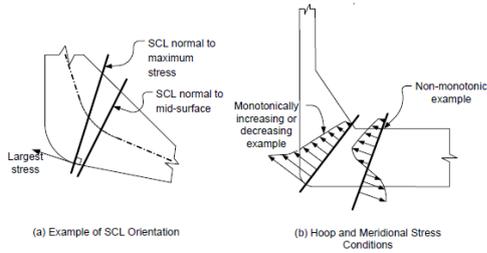


31. Plot graphs of radial and hoop stresses along path CD:



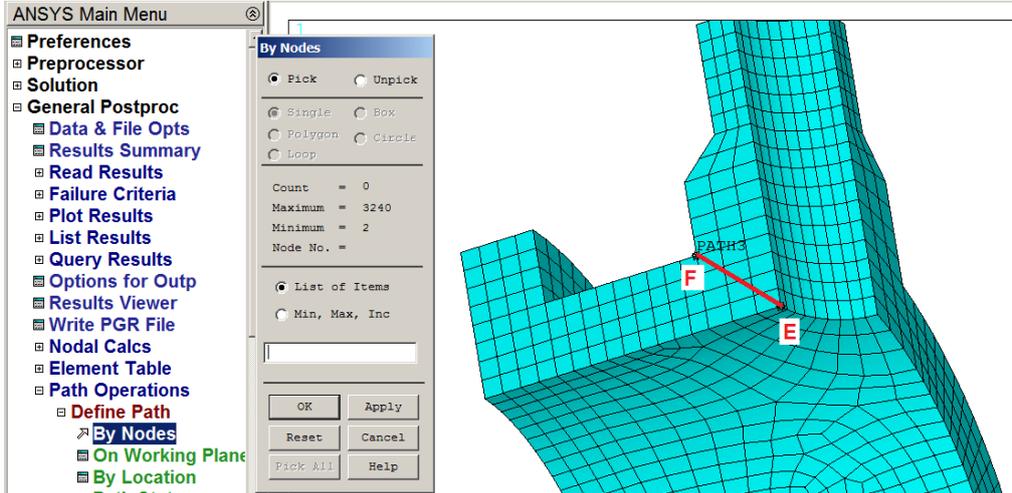
32. List radial and hoop stresses along path CD:



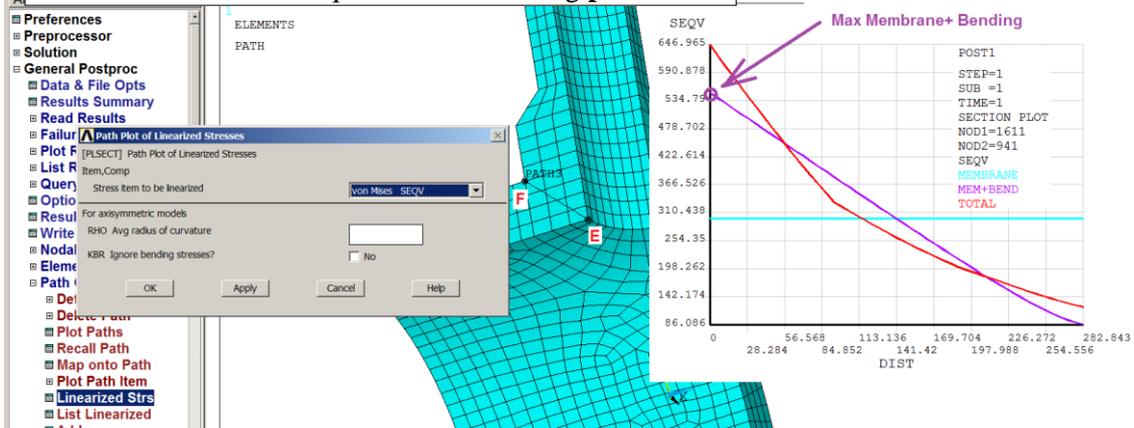


Stress Classification Line Orientation and Validity Guidelines

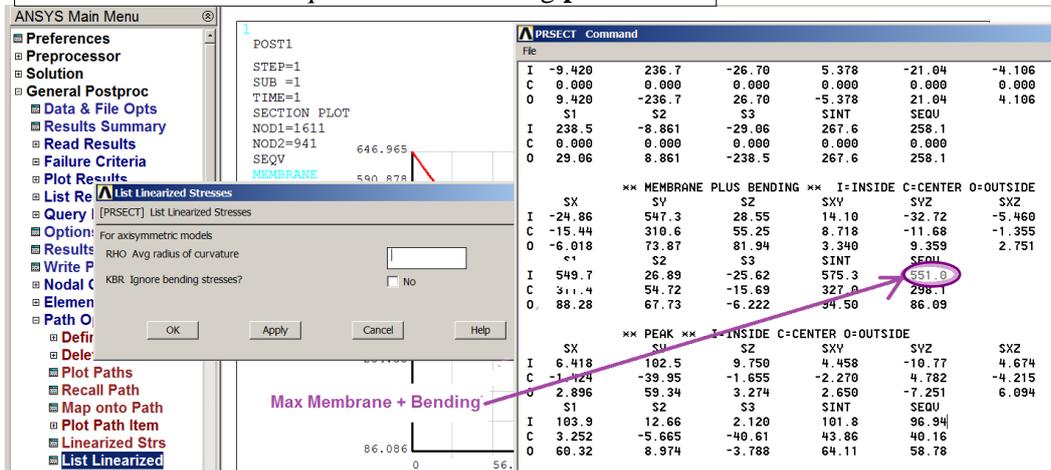
33. Define path EF along wall thickness for equivalent stress linearization:



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:

- 20-noded elements (Solid1865) using ‘sweeping’ HEX/WEDGE option (**Model 1**),
- 8-noded elements (Solid185) using ‘sweeping’ HEX/WEDGE option (**Model 2**),
- 8-noded elements (Solid185) 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 Solid186 Hex/Wed	Model 2 Solid185 Hex/Wed	Model 3 Solid185 Free	
No. of nodes				Plots needed (should be archived during program session for each model) : 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 Report finaly: Final report: 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				
from Lamé theorem (for inside pressure): $\sigma_r = \frac{p_a \cdot a^2}{b^2 - a^2} \cdot \left(1 - \frac{b^2}{r^2}\right) \quad \sigma_t = \frac{p_a \cdot a^2}{b^2 - a^2} \cdot \left(1 + \frac{b^2}{r^2}\right)$				
$\sigma_r(a) =$				
$\sigma_t(a) =$				
$\sigma_r(b) =$				
$\sigma_t(b) =$				