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

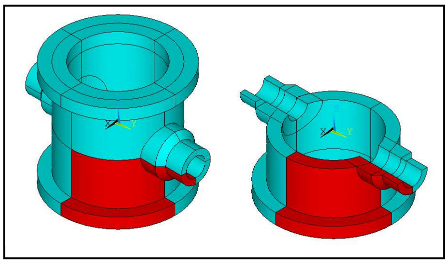
1. INTRODUCTION

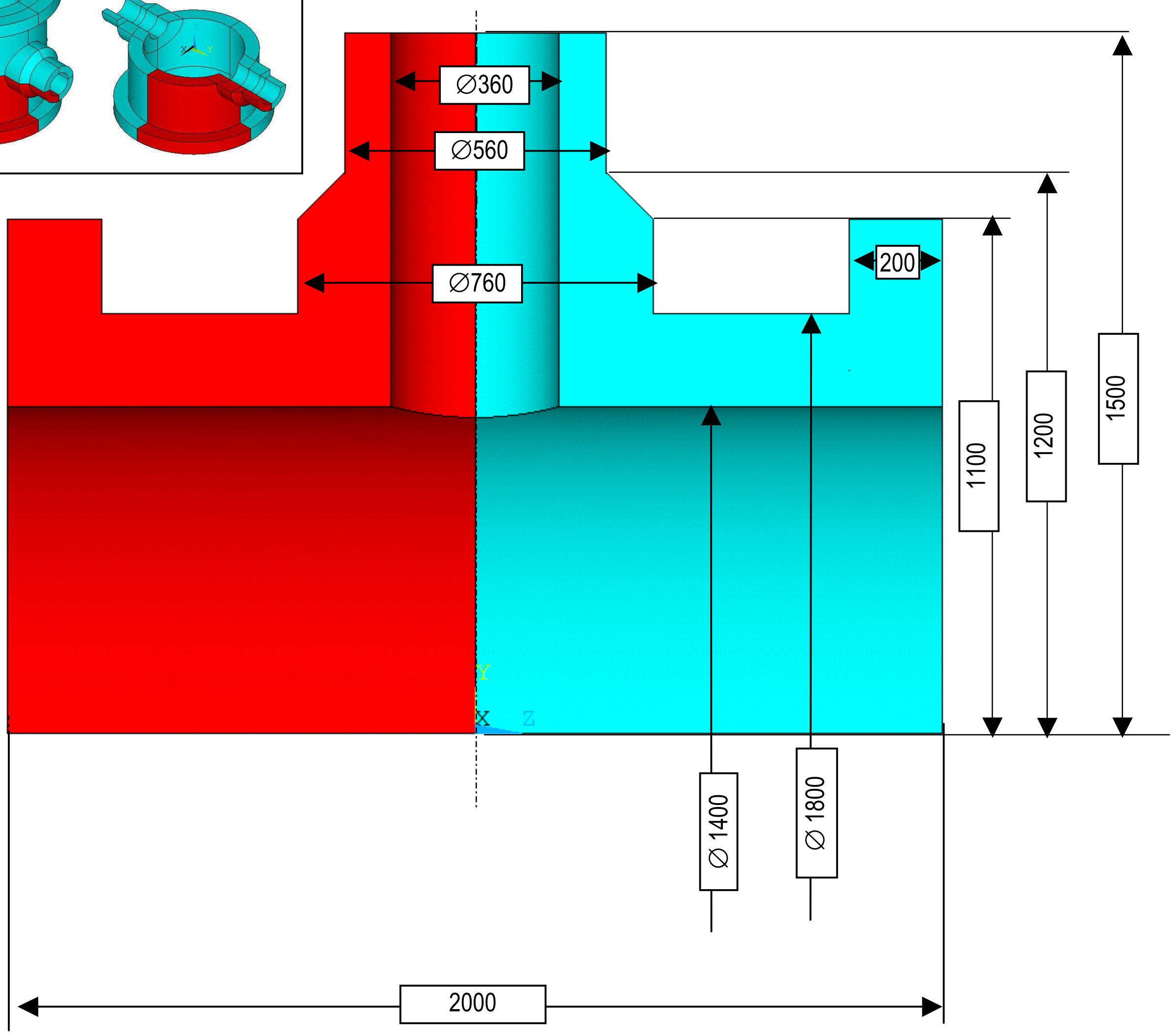
Three-dimensional problem of the theory of elasticity includes an elastic body with defined kinematic or static boundary conditions and the mass forces acting inside. The analytical solution is known only for simple cases. In general, numerical methods are the only way to solve such tasks. Numerical 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: $\boldsymbol{p}=50MPa$, $\boldsymbol{E}=2\cdot10^5\ MPa$, $\boldsymbol{\nu}=0.3$
Geometric data (in millimeters) are presented below:



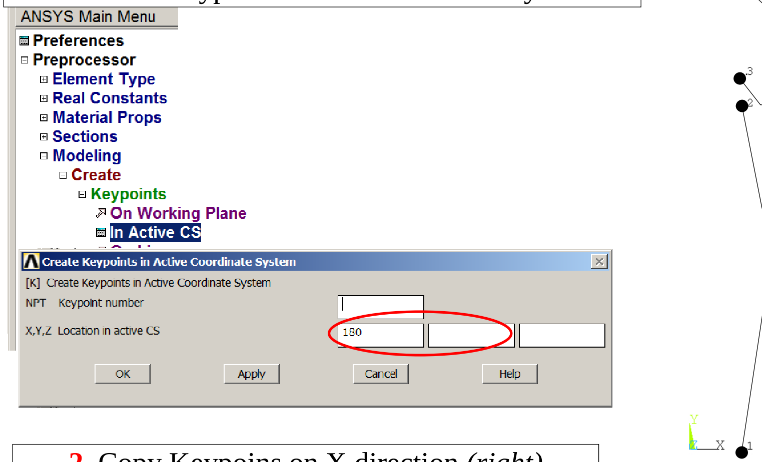


3. TYPICAL COURSE OF NUMERICAL ANALYSIS

Taking into consideration the triple symmetry (xz, yz and zx planes), the model includes only ⅛ part of the vessel. Convenient units are: ***mm, N and MPa***.

3.1. Preprocessor

1. Create Keypoints active Coordinate System



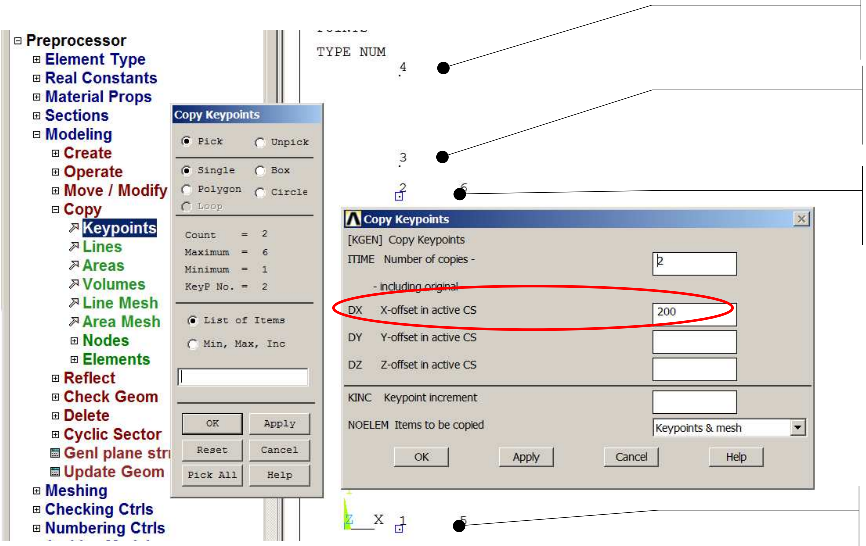
Preprocessor>
Modeling>Create>Keypoint>
In Active CS: X=180, Y=1500

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 Keypoins on X direction *(right)*



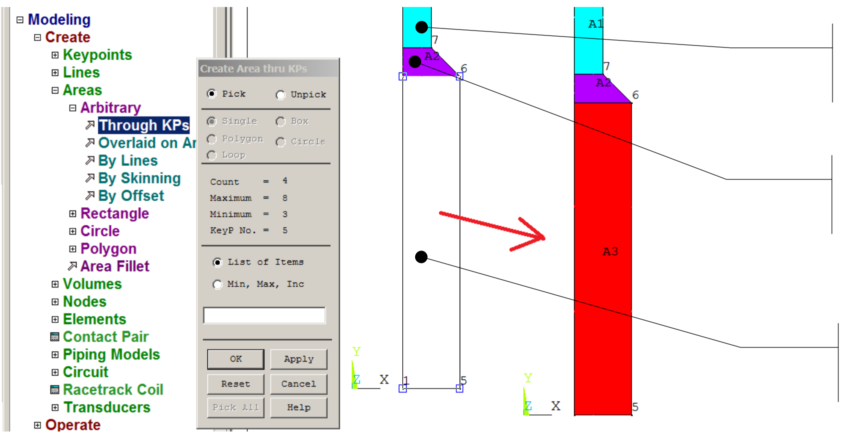
Preprocessor>Modeling>
Copy>Keypoint No 4: DX=100

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

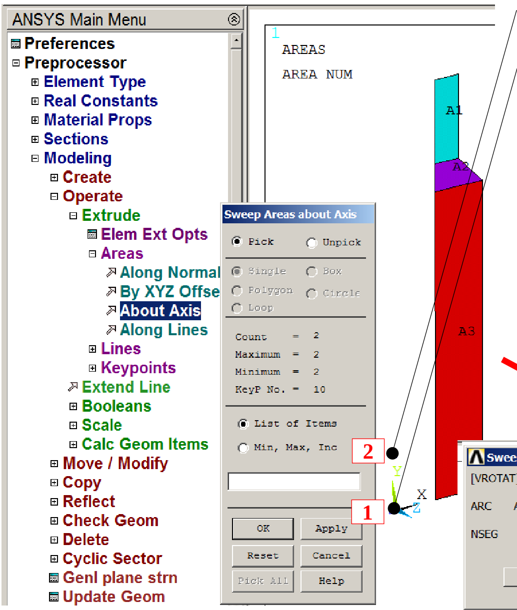
Preprocessor>Modeling>
Create>Areas>Arbitrary>
Through KPs: 2,3,7,6

Preprocessor>Modeling>
Create>Areas>Arbitrary>
Through KPs: 1,2,6,5

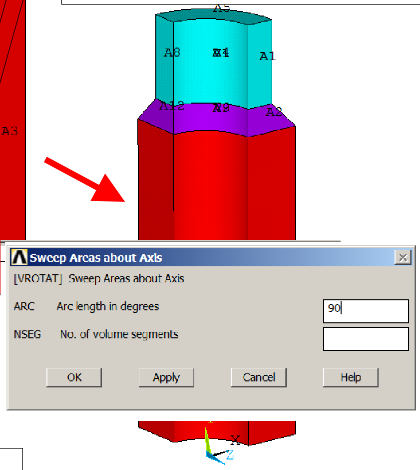
4. Create Keypoints in active Coordinate System (*on axis of revolution - Y)*

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

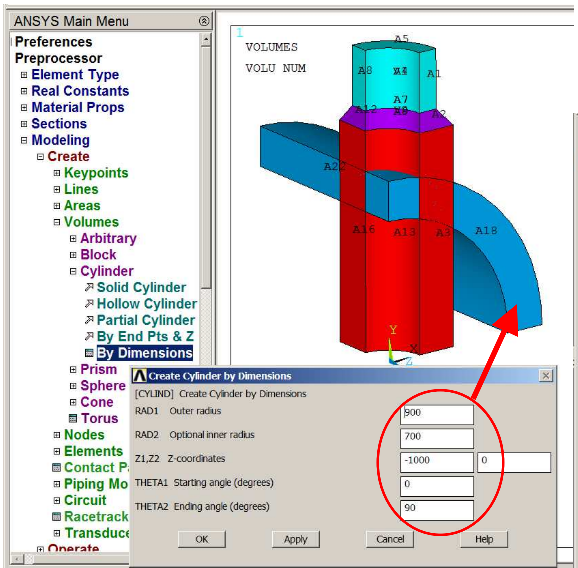
Preprocessor> Modeling>Create>Keypoint> In Active CS: X=0, Y=0



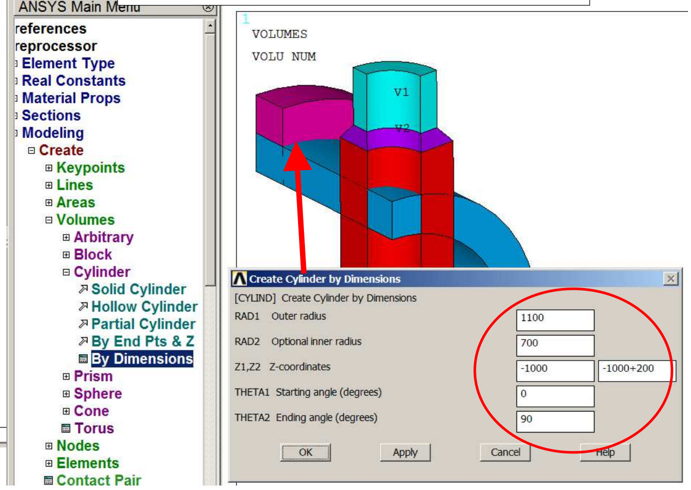
5. Extrude areas about axis defined by two Keypoints:
Preprocessor>Modeling>Operate> Extrude>Areas>About Axis



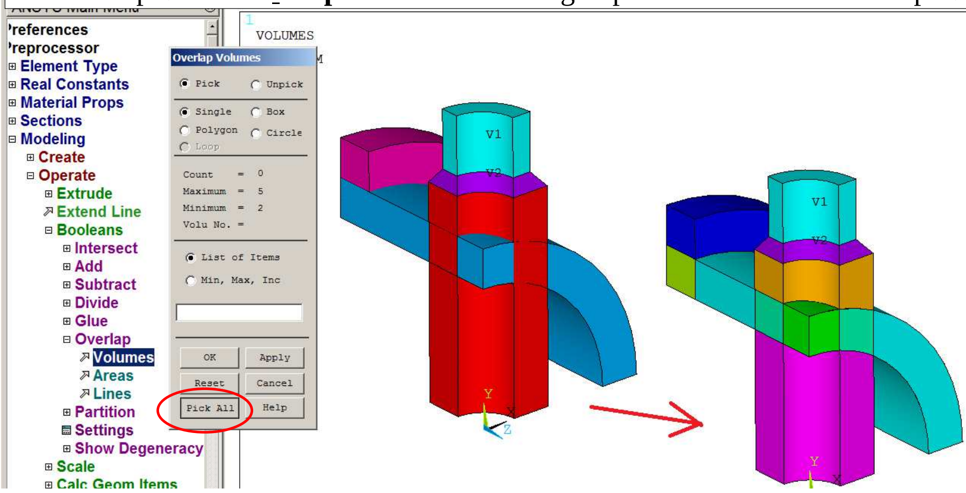
6. Create ¼ of cylinder:



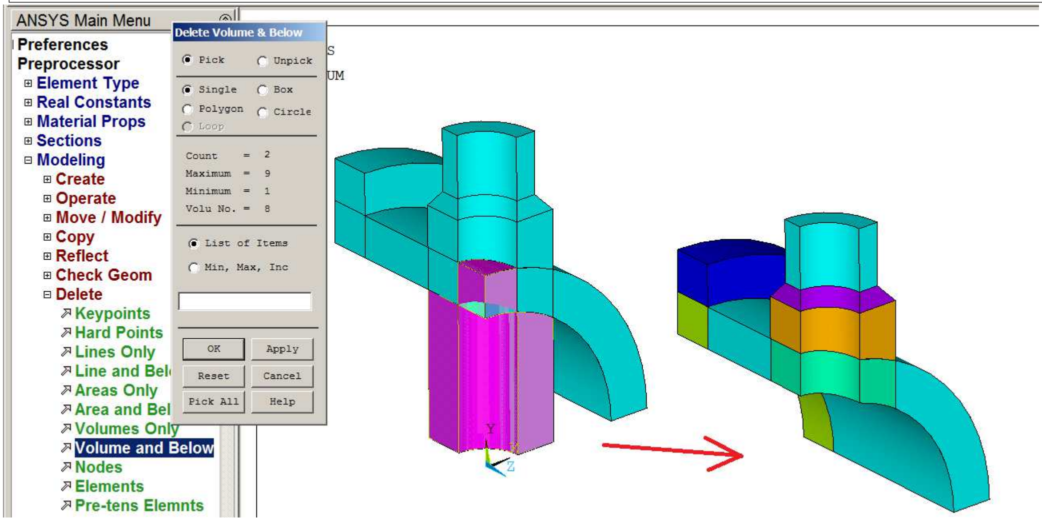
7. Create ¼ of cylinder for flange:



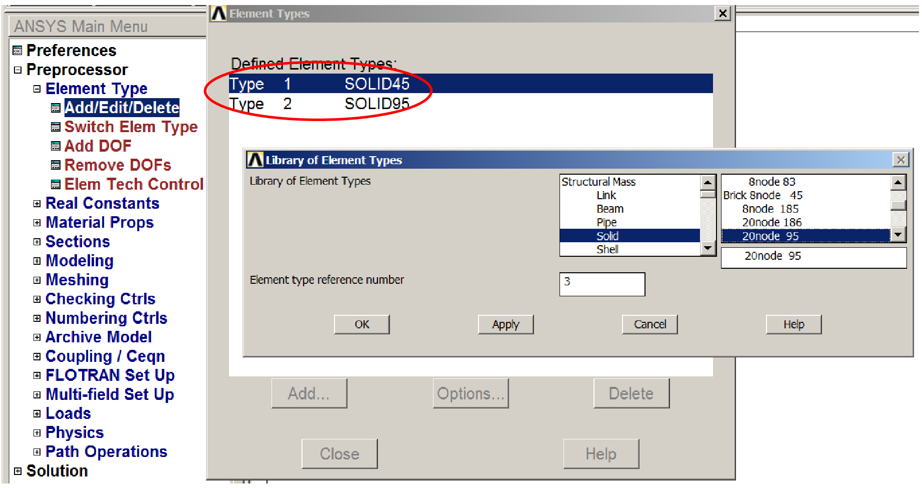
8. Overlap Volumes: **Preprocessor**>Modeling>Operate> Booleans>Overlap>Volumes: All



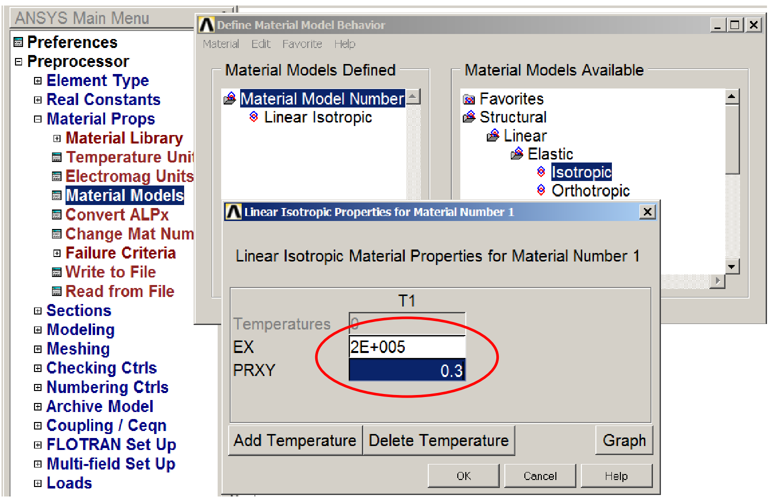
9. Delete unnecessary Volumes: **Preprocessor**>Modeling>Delete> Volumes and Below



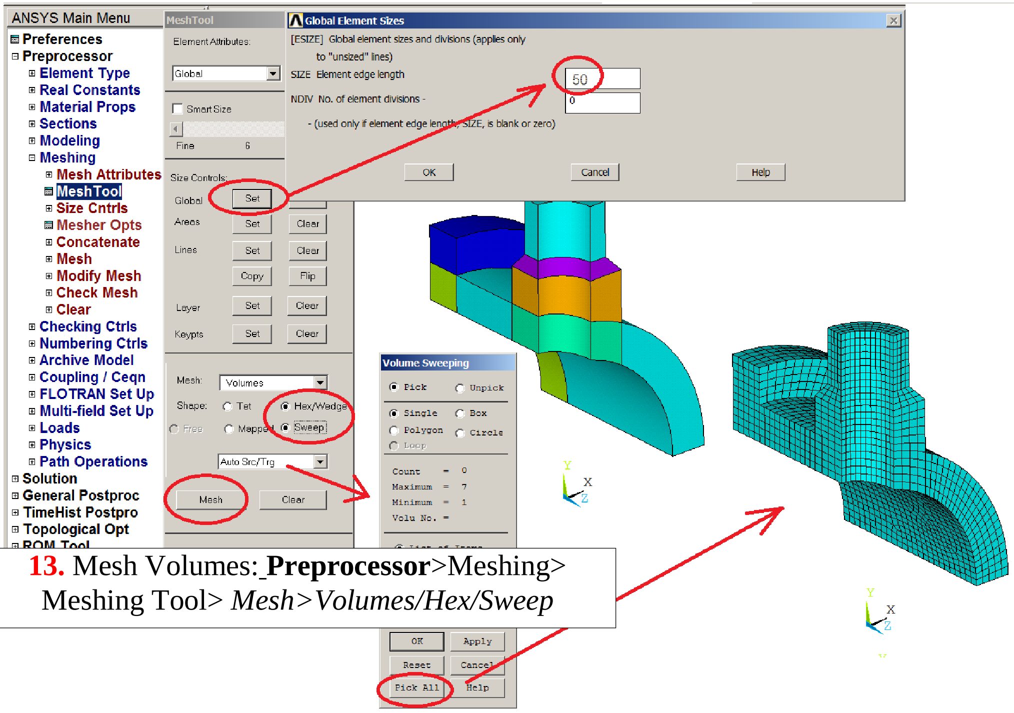
10. Select Element types: **Preprocessor**>Element Type>Add> (SOLID45 and SOLID95)



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*

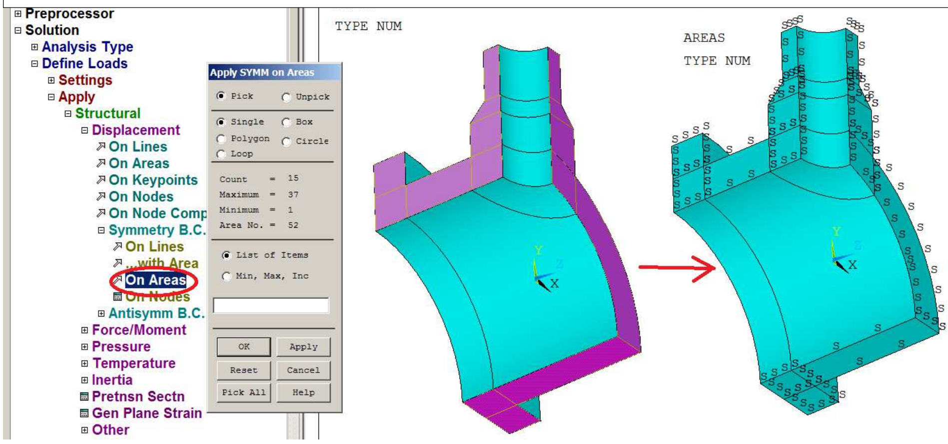


13. Mesh Volumes: **Preprocessor**>Meshing>
Meshing Tool> *Mesh>Volumes/Hex/Sweep*

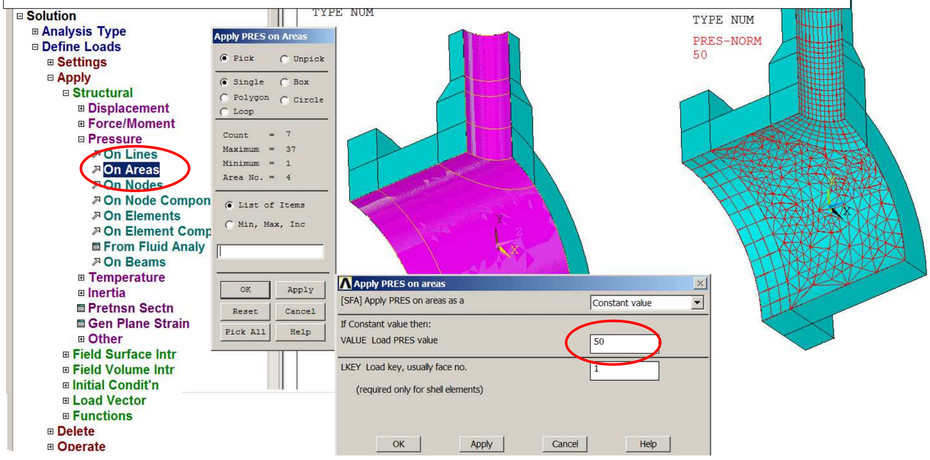
3.2. Solution

Define boundary conditions:

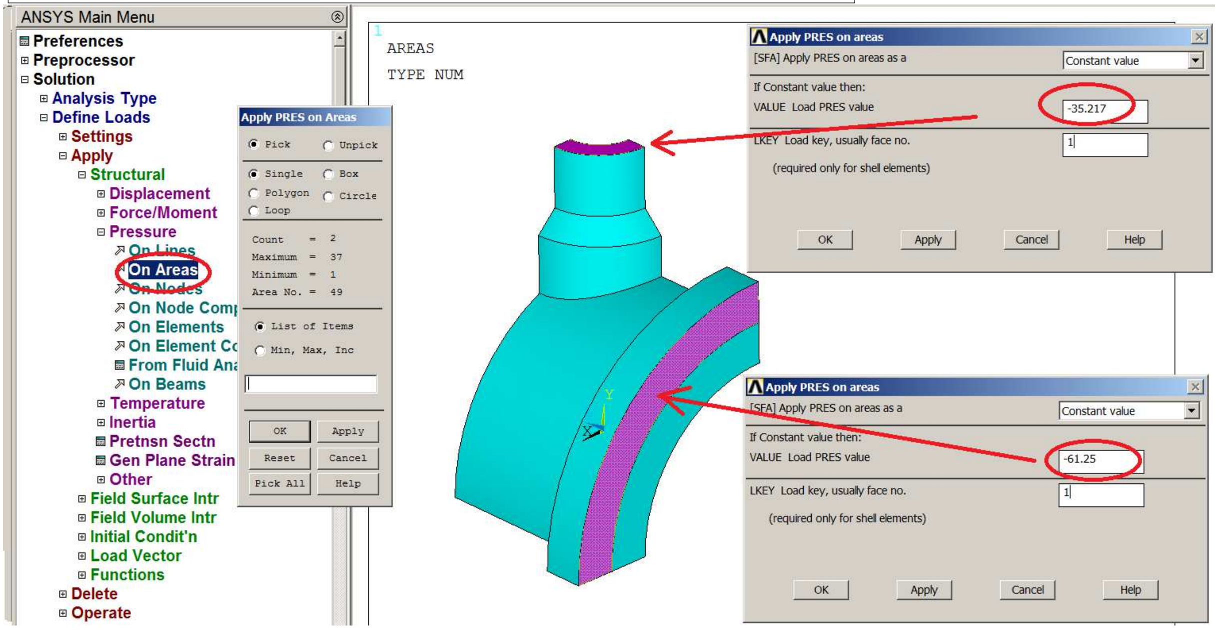
14. Define Symmetry B.C. on Areas:
Solution>Define Loads> Apply>Structural>Displacement> Symmetry BC>On Areas

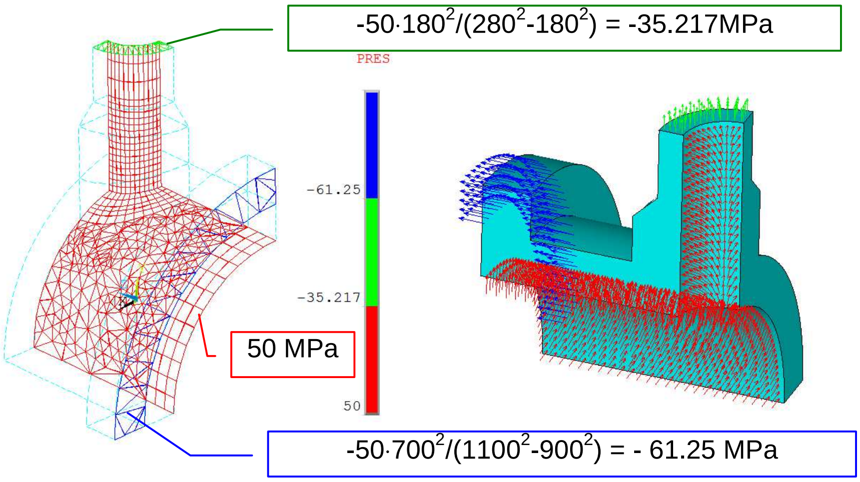


15. Define pressure on internal Areas:
Solution>Define Loads> Apply>Structural>Pressure>On Areas

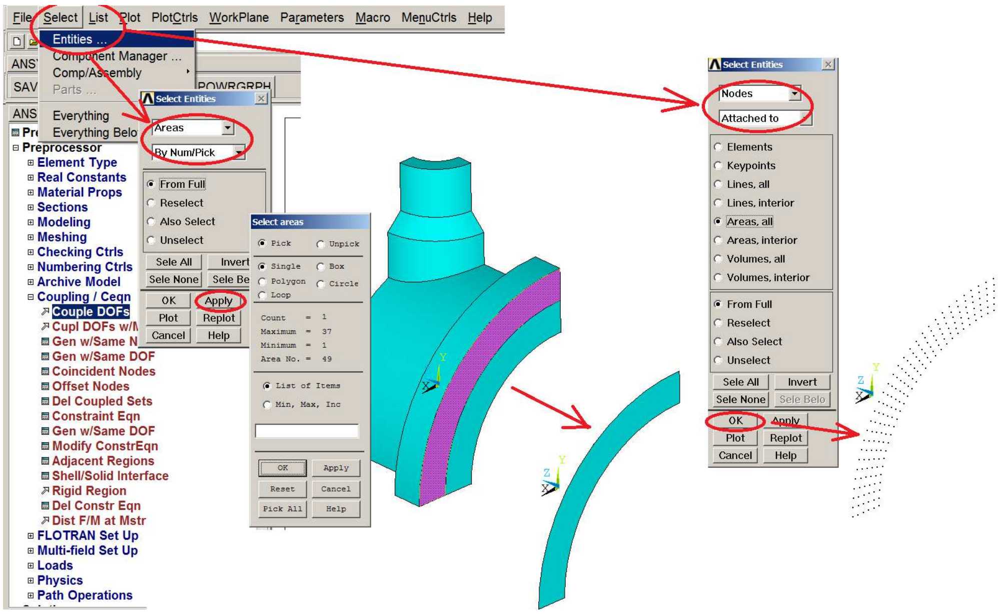


16. Define negative pressure on nozzle and flange areas:
Solution>Define Loads> Apply>Structural>Pressure>On Areas

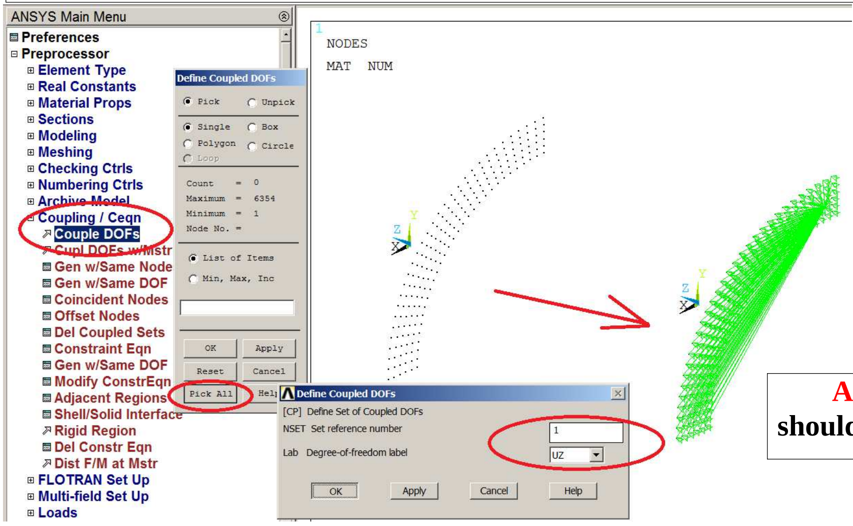




17. Select nodes on the sticking surface of the flange:

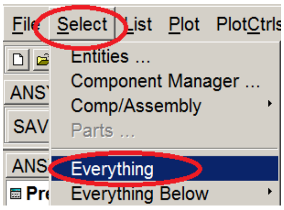


18. Couple DOFs (UZ) on the sticking surface of the flange:
Preprocessor>Coupling / Ceqn> Couple DOFs



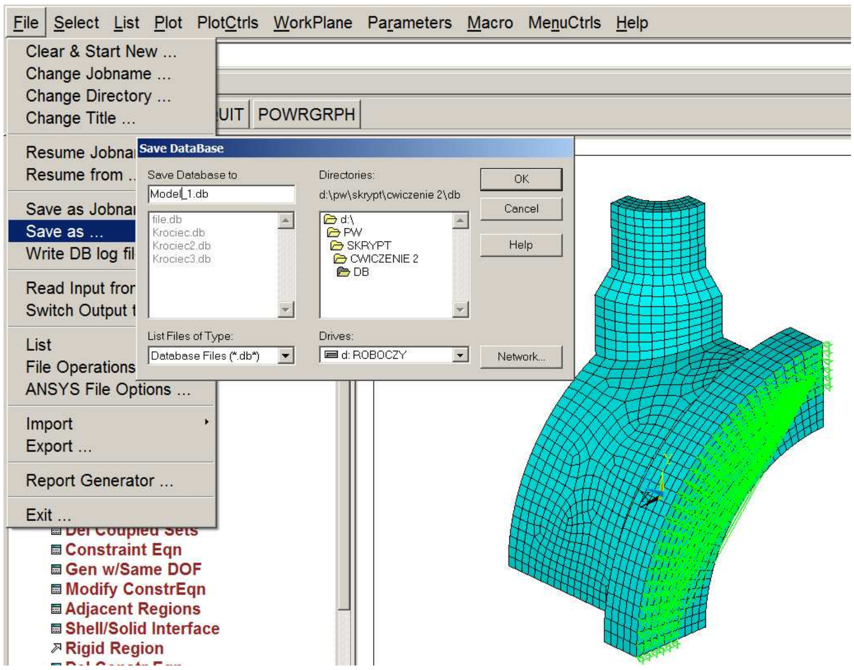
ATTENTION: Steps 17 and 18 should be performed for each model !!!

19. Select all entities:



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

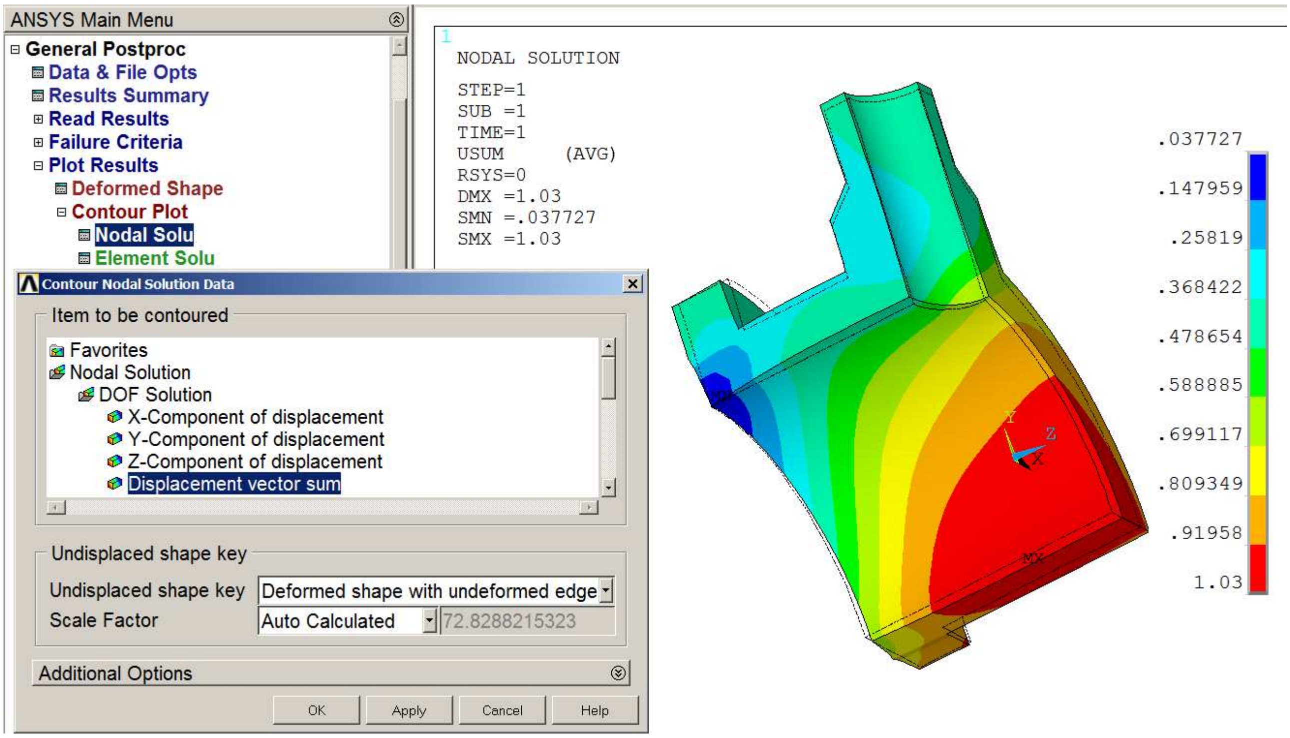
21. Save database with a unique name: ***Model_1.db***



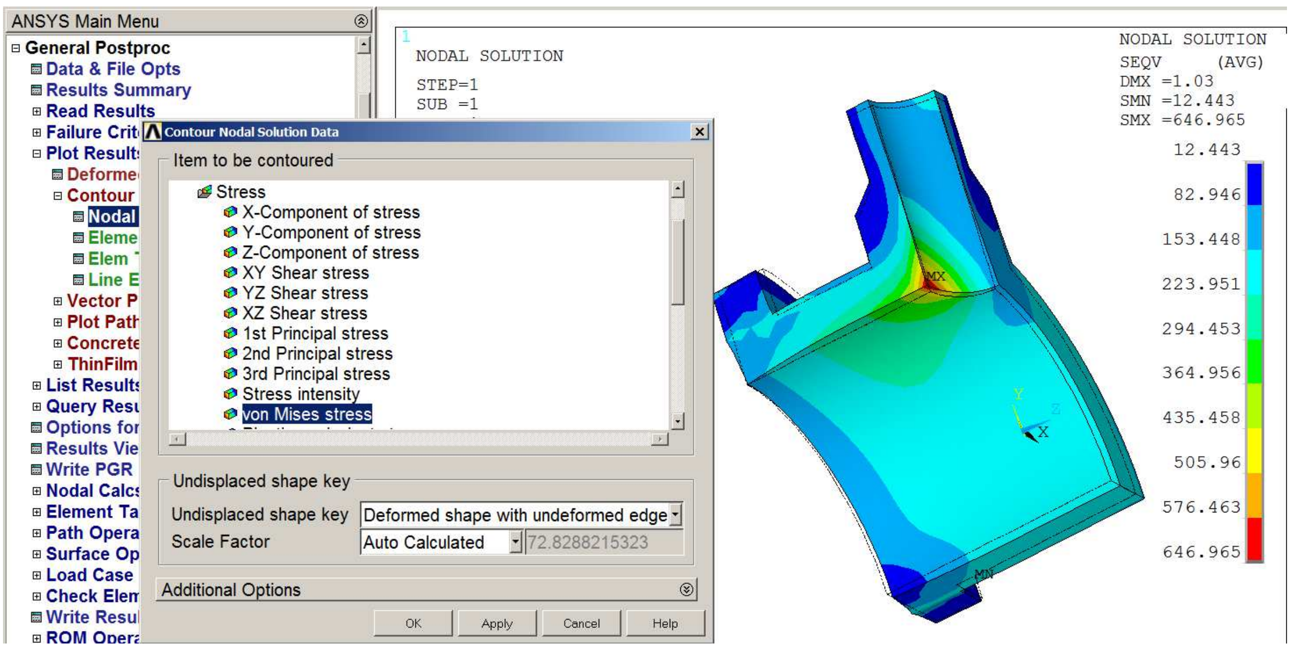
3.3. General postprocessor

<u>Show the results as contour maps</u>:
Show total displacements (USUM), Von Mises stress (SEQV) and stress components (SX, SY) in global cylindrical system related to cylindrical part of the model.

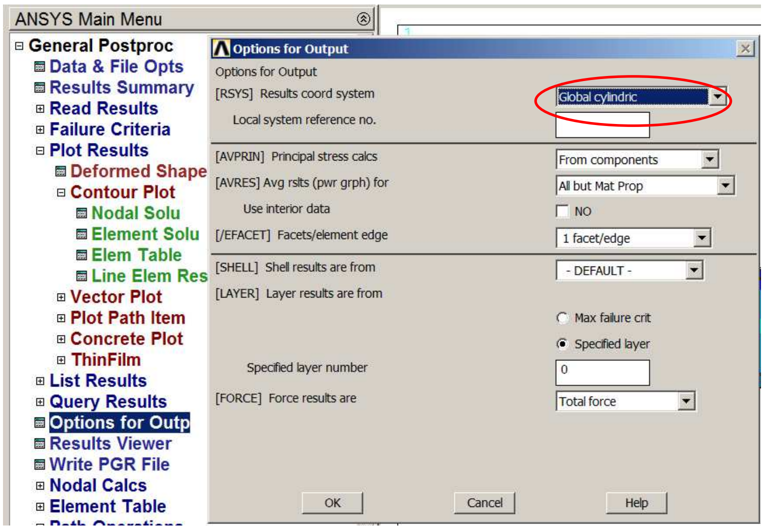
22. Plot Total displacements *(USUM)*



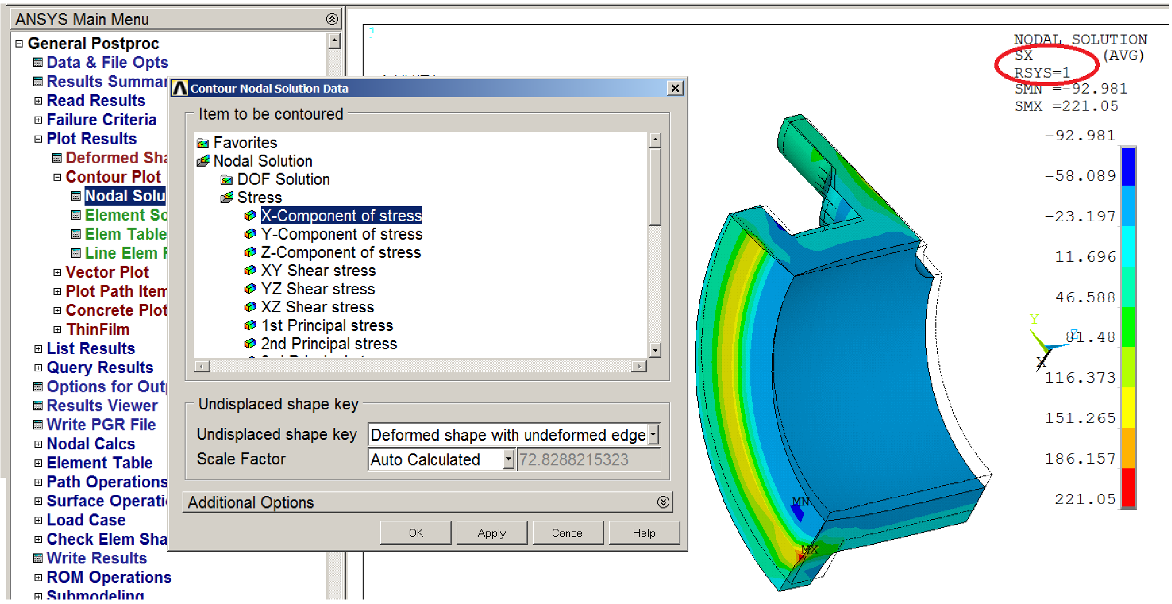
23. Plot Von Mises stress *(SEQV)*



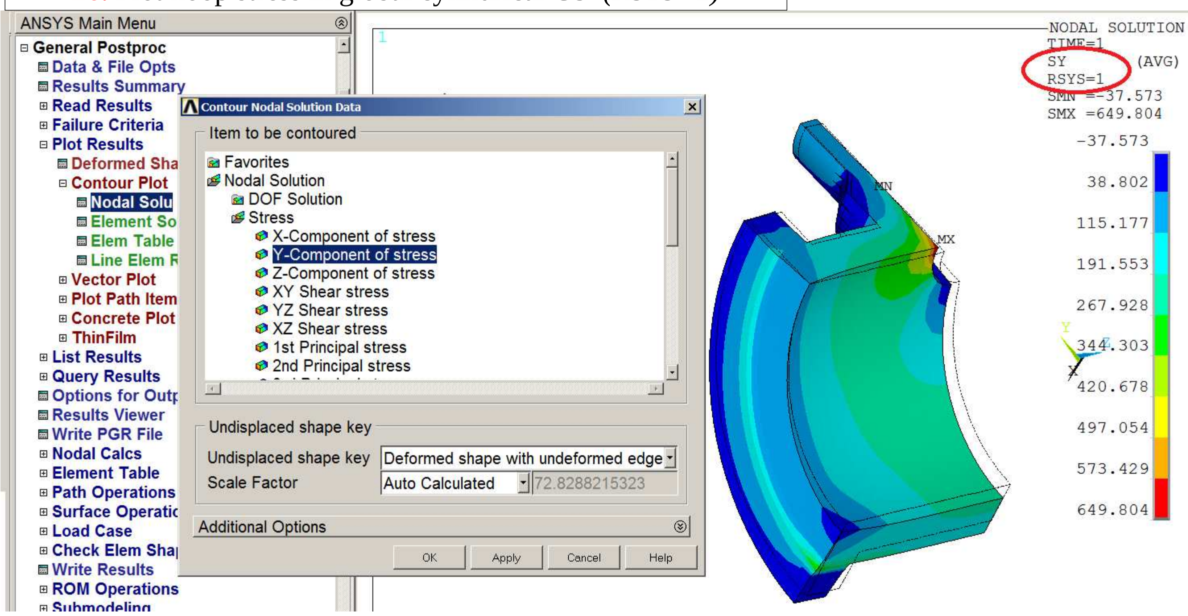
24. Select global cylindrical CS for results presentation:



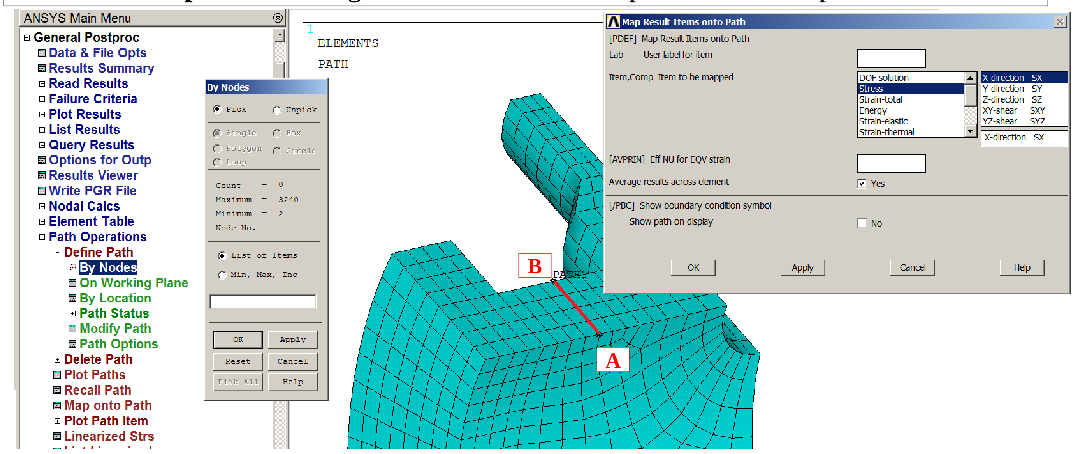
25. Plot radial stresses in global cylindrical CS *(RSYS=1)*



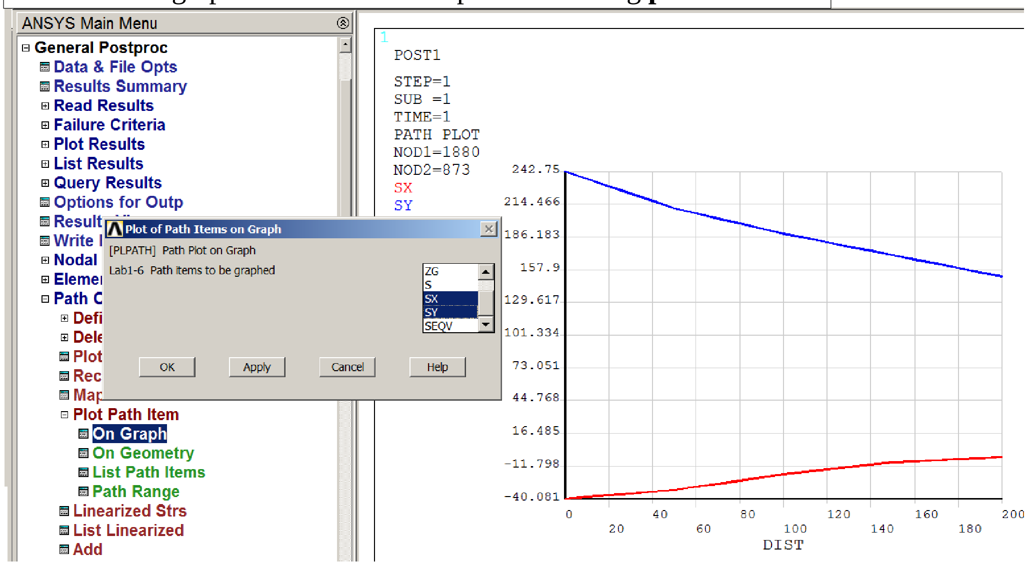
26. Plot hoop stress in global cylindrical CS *(RSYS=1)*



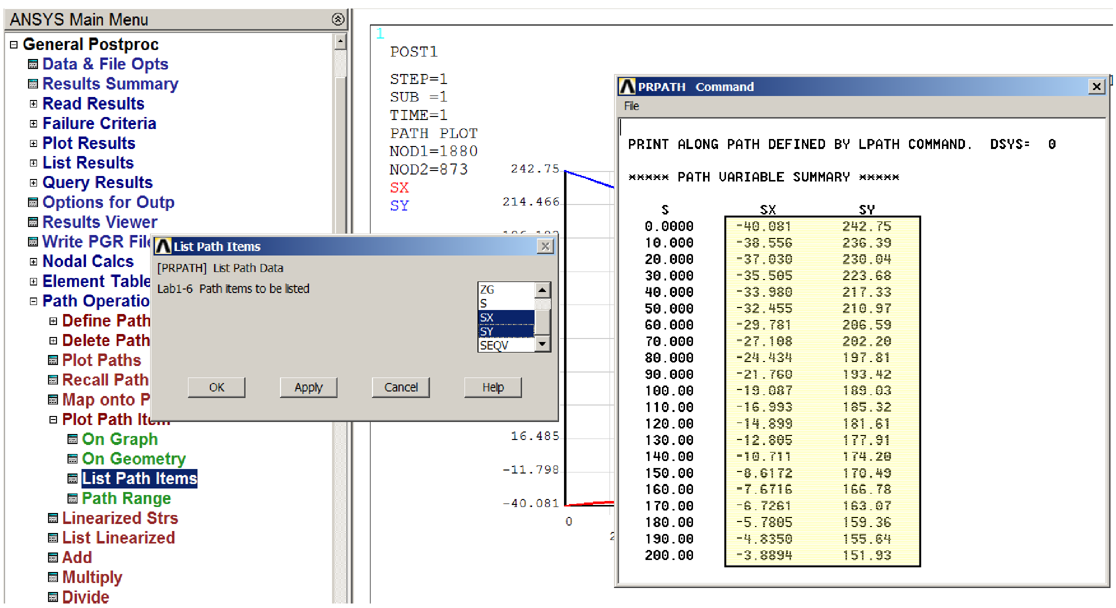
27. Define **path AB** along wall thickness and map radial and hoop stresses on it:



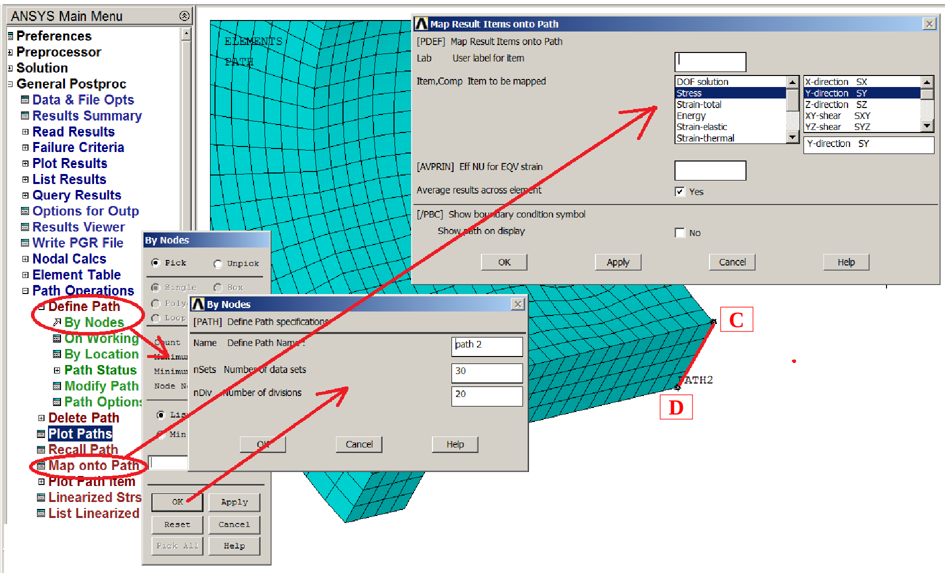
28. Plot graphs of radial and hoop stresses along **path AB**:



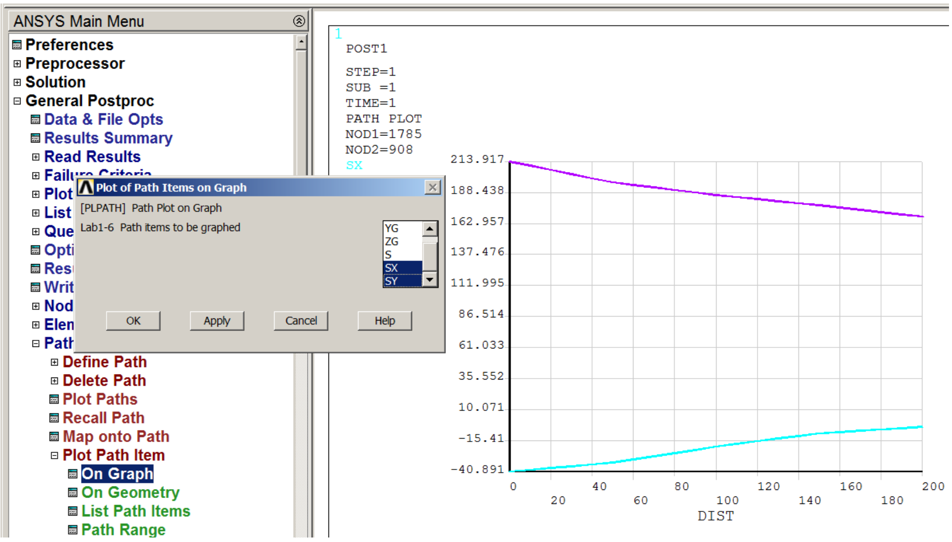
29. List radial and hoop stresses along **path AB**:



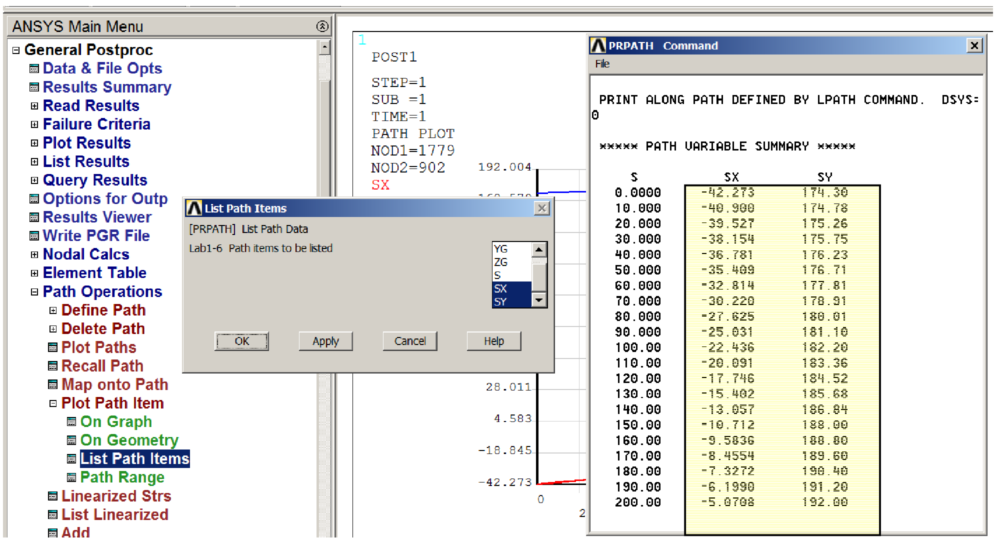
30. Define **path CD** along wall thickness and map radial and hoop stresses on it:

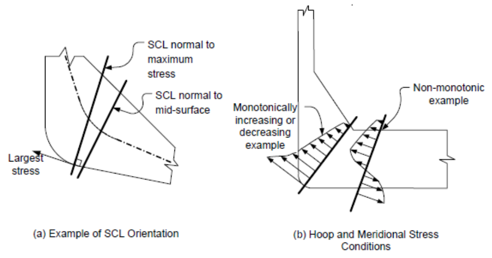


31. Plot graphs of radial and hoop stresses along **path CD**:

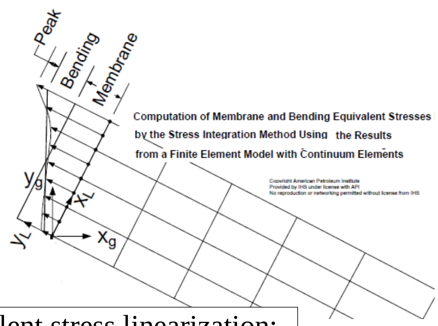


32. List radial and hoop stresses along **path CD**:

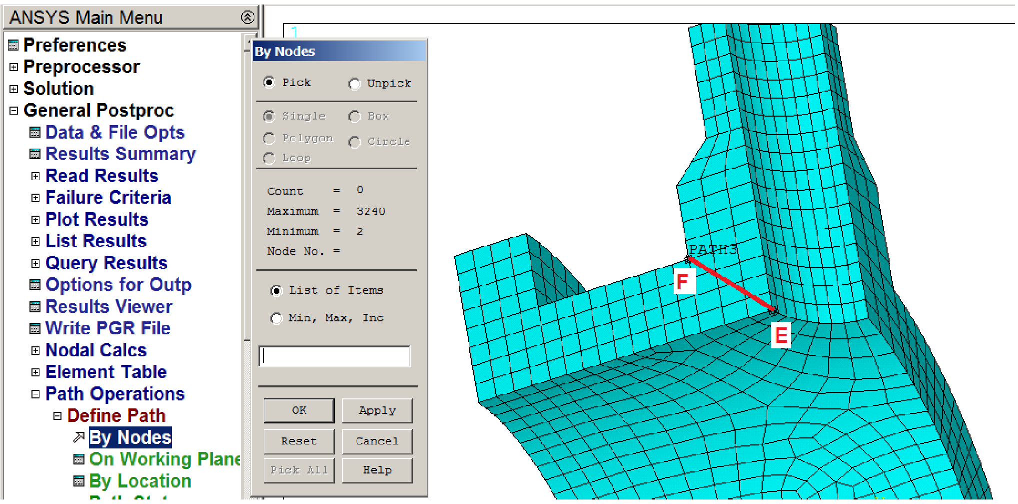




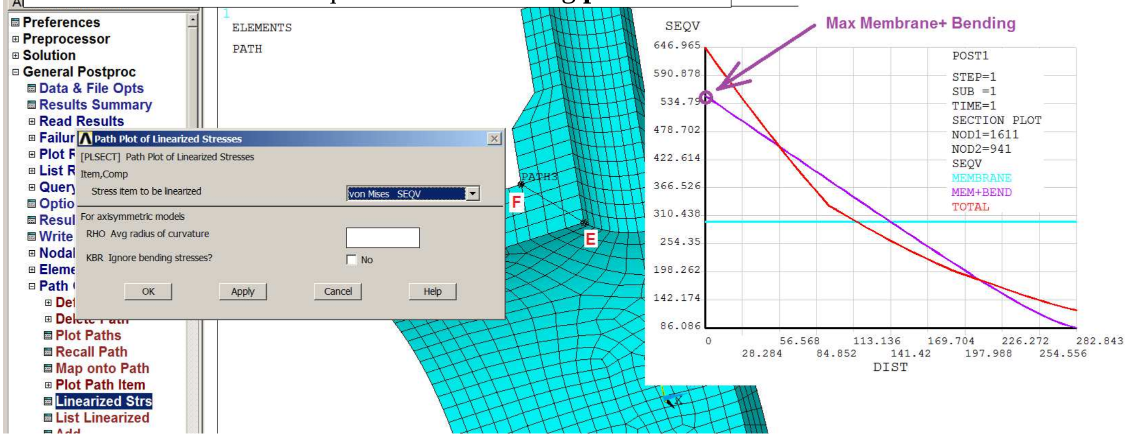
Stress Classification Line Orientation and Validity Guidelines



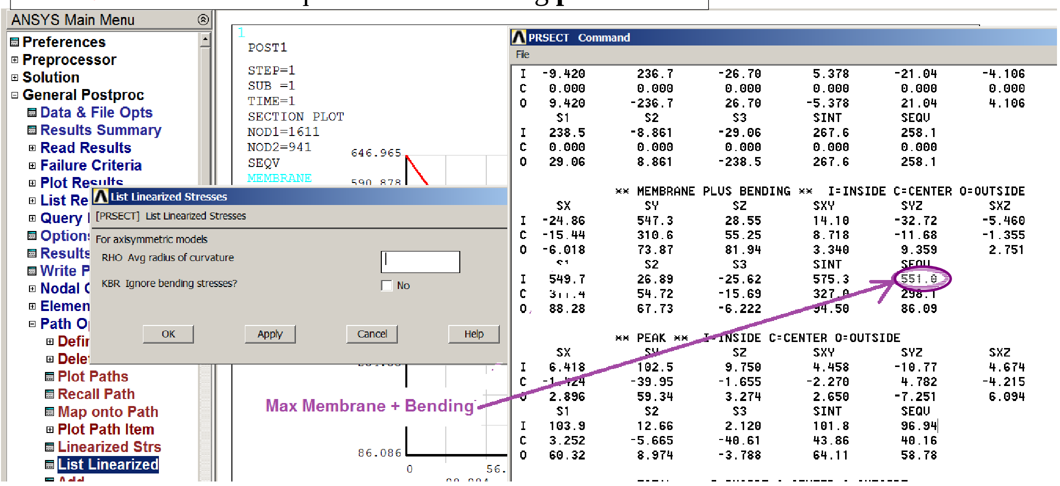
33. Define **path EF** along wall thickness for equivalent stress linearization:



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:

a) 8-noded elements (Solid45) using 'sweepping' HEX/WEDGE option (**Model 1**),
b) 20-noded elements (Solid95) using 'sweepping' HEX/WEDGE option (**Model 2**),
c) 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			
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 M**embrane +* ***B**ending **stress			

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

Raport finalny:

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

from Lame theorem *(for inside pressure)*:

$$\sigma_r = \frac{p_a \cdot a^2}{b^2 - a^2} \cdot \left(1 - \frac{b^2}{r^2}\right) \qquad \sigma_t = \frac{p_a \cdot a^2}{b^2 - a^2} \cdot \left(1 + \frac{b^2}{r^2}\right)$$

$\sigma_r(a) =$

$\sigma_t(a) =$

$\sigma_r(b) =$

$\sigma_t(b) =$