CHAPTER FOUR

ANSYS ANALYSIS OF THE PRESSURE VESSEL

4-1: Introduction

In this chapter the pressure vessel previously designed according to ASME BOILER & PRESSURE VESSEL CODE Sec VIII will be check for strength uses ANSYS package.

The pressure vessel under analysis (shown in figure 4.1) is made of Miled steel (E = 207GPa, ν = 0.27) and contains an internal pressure of P = 1.75MPa. The cylindrical vessel has an inner diameter of 1.4m with ellipsoidal end caps. The end caps have a wall thickness of 15 mm, while the cylinder walls are 15mm thickness.

ANSYS package will be used to analyse the stresses and deflections in the vessel walls due to the internal pressure. Since the vessel is axially symmetric about its central axis, an axisymmetric analysis will be performed using two-dimensional, 8-node quadrilateral elements (Plane 183) with the axisymmetric option activated. In addition, the vessel is symmetric about a plane through the center of the cylinder. Thus, only a quarter section of the vessel needs to be modelled. The weld was not considered in the analysis because in the previous chapter the weld efficiency E assumed as unity.

ANSYS analysis has three distinct steps:

i. Build the model.
ii. Apply loads and boundary conditions and obtain the solution.
iii. Review the results.

4-2: Geometry

Figure: 4-1 shows the pressure vessel dimensions that it will be analysis against ANSYS.

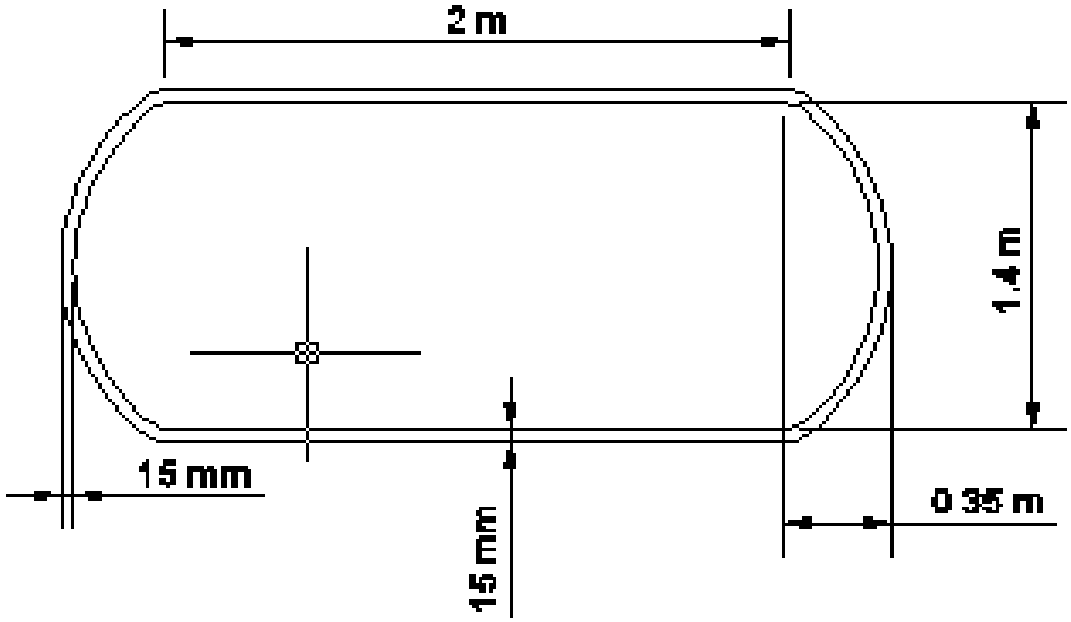


Figure 4-1: The Pressure Vessel Dimensions

4-3: Build the Model

Building a finite element model requires practice and knowledge considerable, and careful in dealing with orders at uses ANSYS program such as Preferences, Element types, Material properties, Create modelling, Mesh, Apply loads, Solution (solve) and Read results.

4-3-1: Preferences

This first step of the analysis, where we define the type of analysis and choose Structural.

4-3-2:Element Types

In the element type we will chose Solid and 8 node 183 then chose Axisymmetric

4-3-3:Material Properties

Material properties is very important during the analysis, and is available in ANSYS program many materials to perform the necessary analysis of various materials, Material we have chosen is structural steel. Which have subject stress$\sigma_y = 235$ MPa.

Then enter the properties of material which the pressure vessel made of

E= 207 GPa, V= 0.27 and Density$\rho = 7850\ \mathrm{kg}/m^3$

4-3-4:Create model

The model of the vessel was created using AutoCAD® then it was Exported to an IGES file format, and then it was Imported to ANSYS

4-4:Selection of Mesh Size

The analysis was carried out using different mesh sizes ranging from 0.1 up to 0.02m for each mesh size the maximum Von Mises stress in a specific critical point was recovered.

A plot of max-stress versus mesh size was obtained using Microsoft Excel; this plot is shown in figure 4-2.

Accordingly, a mesh size of 0.03m was considered for the subsequent analysis, it could be noted that 0.03 in the point where the curve started to decline.

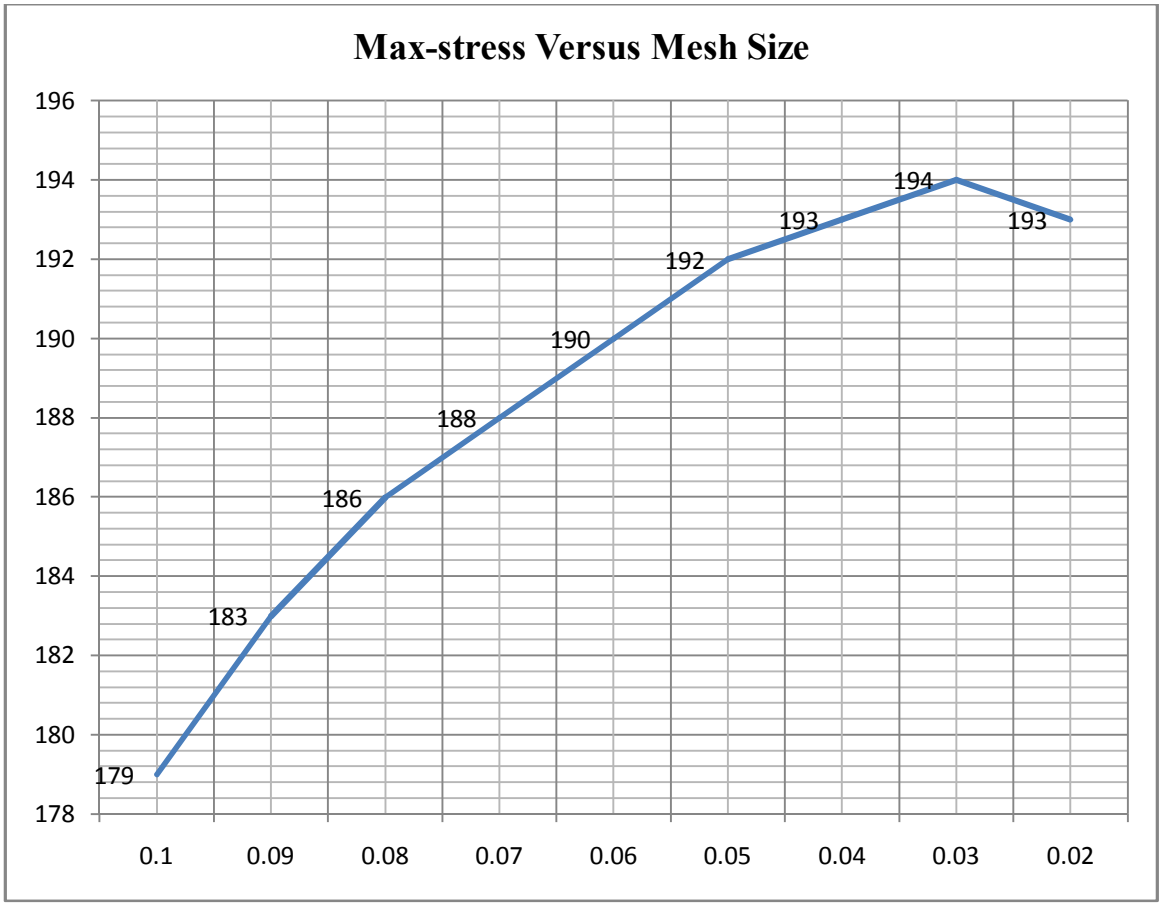
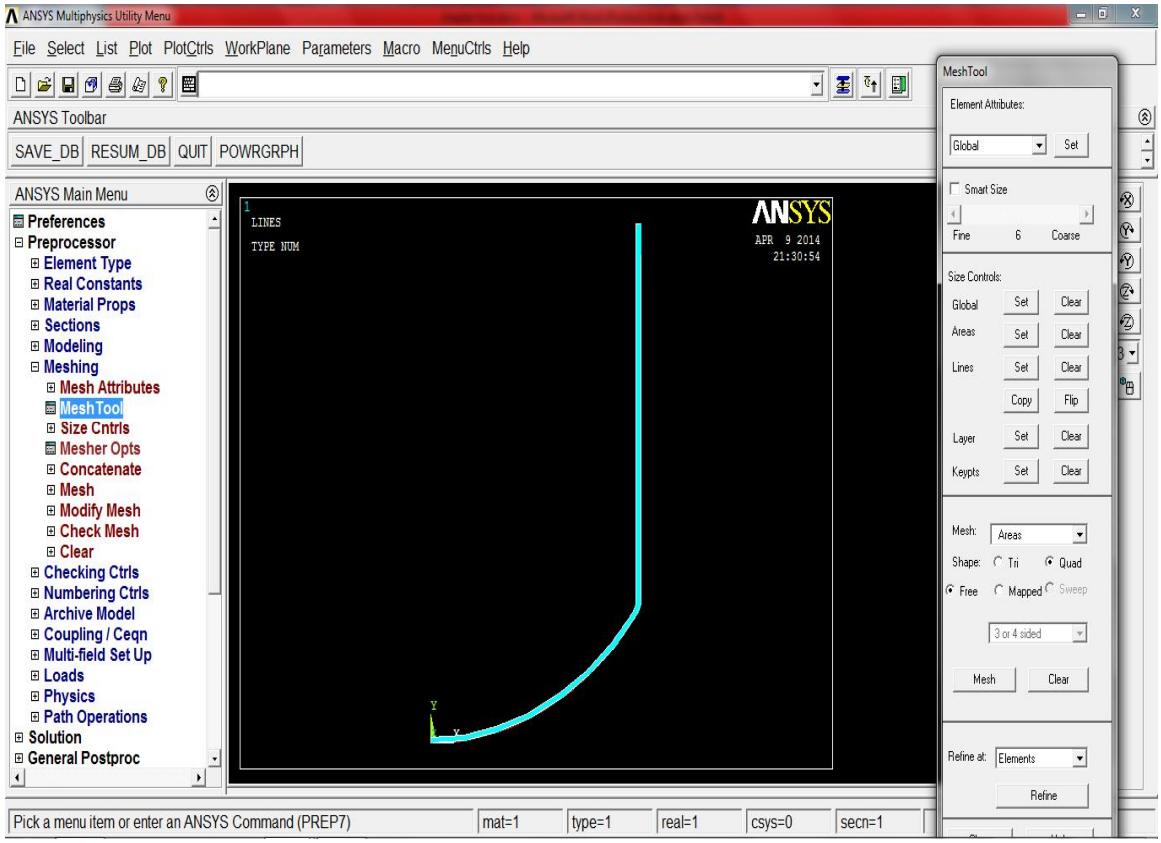


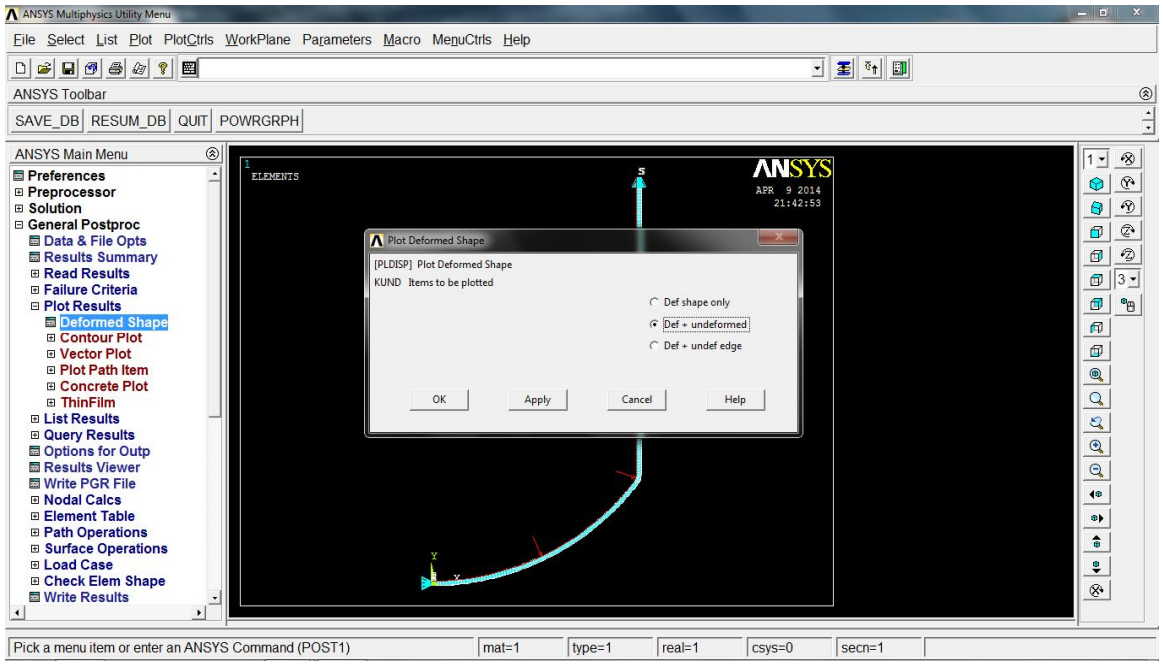
Figure 4-2: The mesh size selection plot

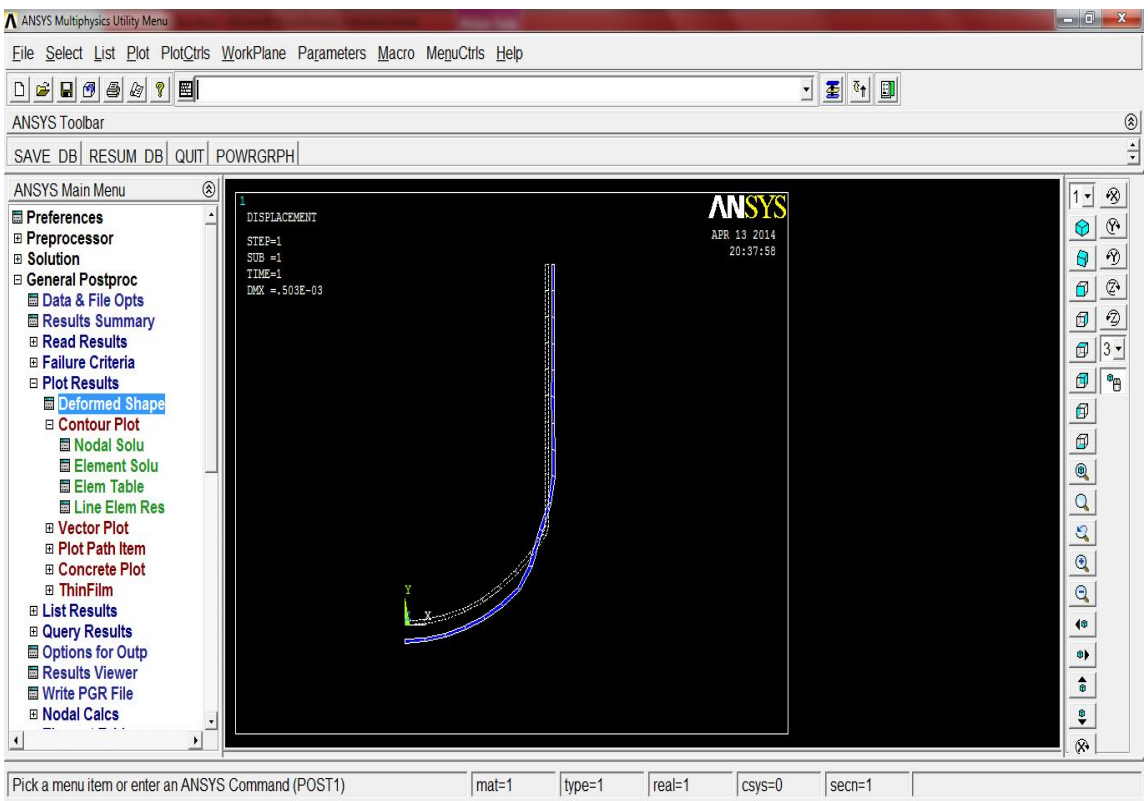


4-5: Define boundary conditions and loads and run the analysis

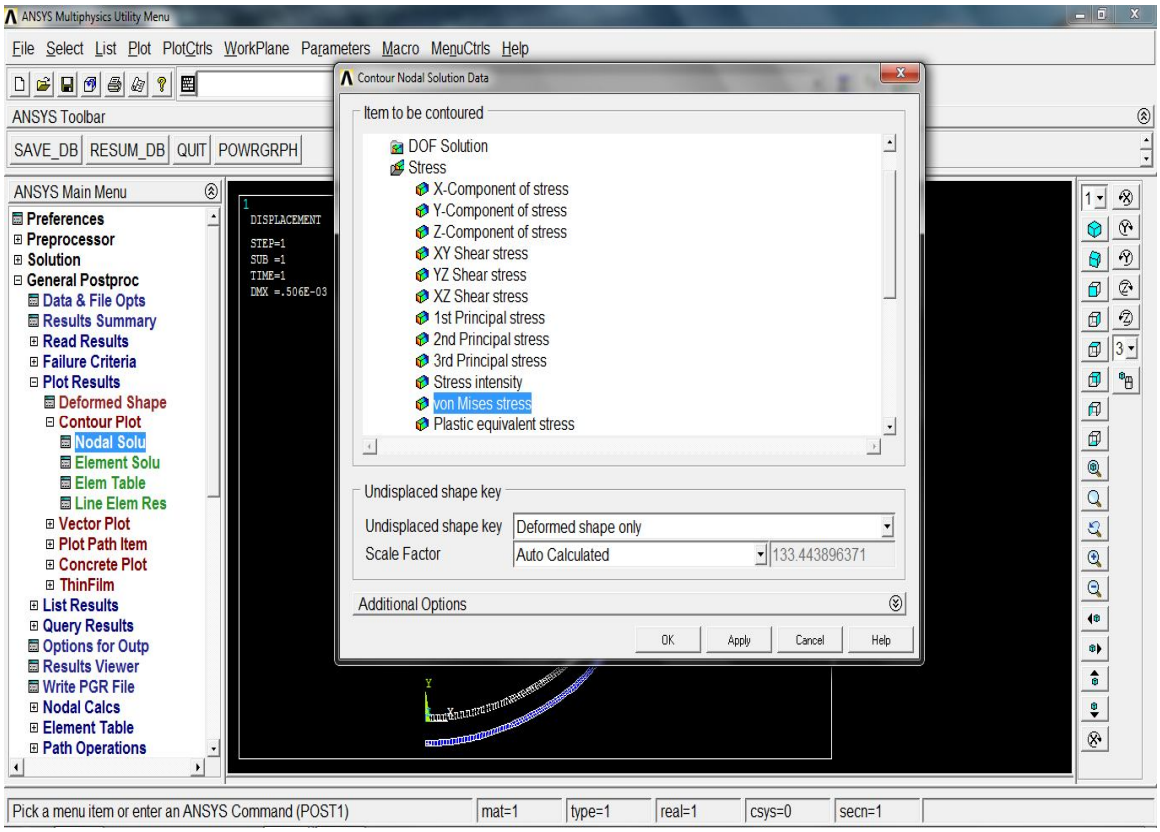
The Boundary Conditions and Loads can now be applied. ANSYS will automatically apply the Axisymmetric Boundary Conditions along the Y-axis. However, we must apply the Symmetry Boundary Conditions along the upper edge of the model. Finally, the Pressure can be applied on all lines that make up the inner surface of the vessel. The magnitude should be input as the actual value – no reduction is needed to account for axisymmetric (ANSYS automatically makes the necessary adjustment of Loads in an Axisymmetric model).

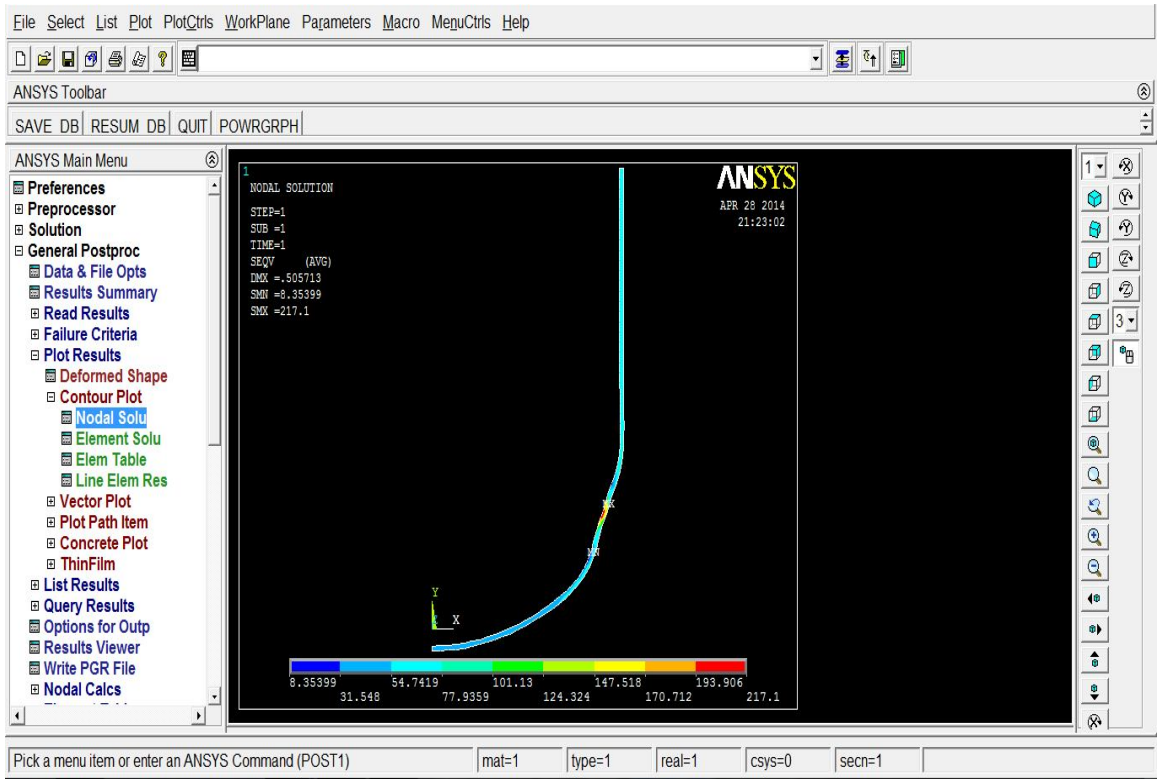
4-6:Deformed Shape

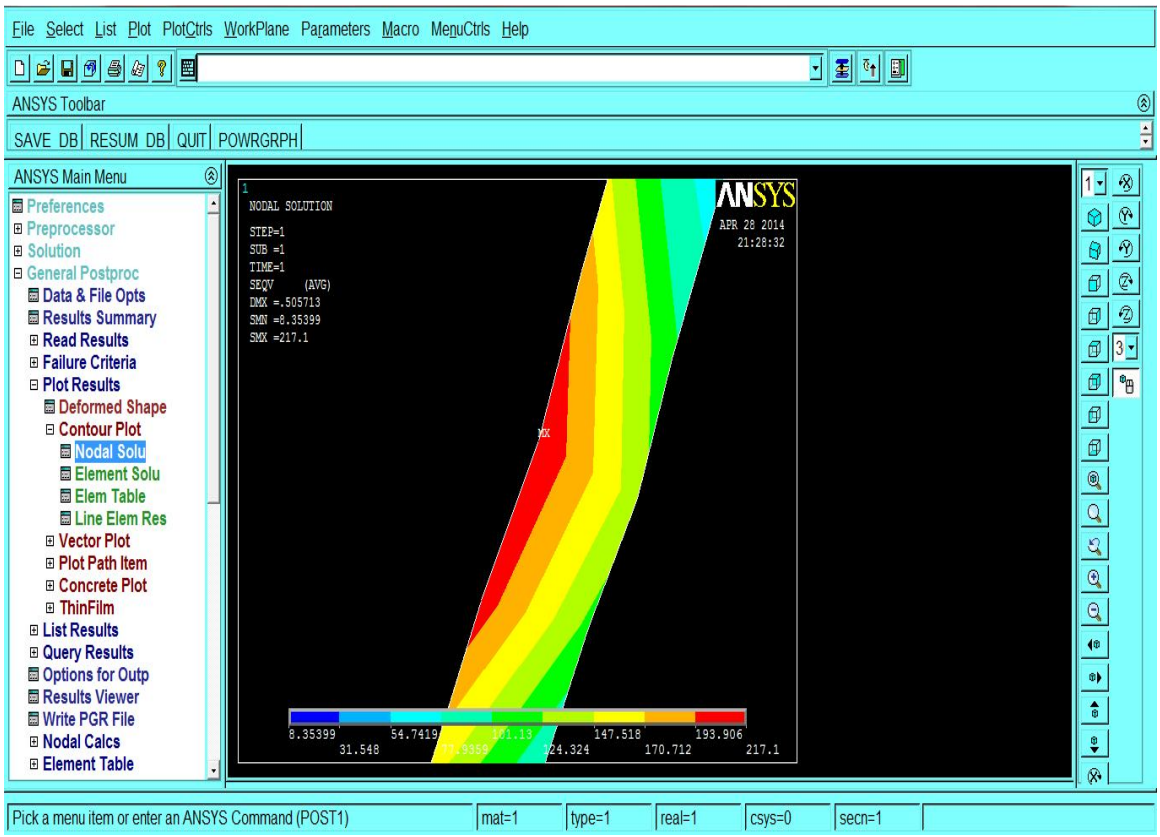




4-7: Create Contour Plots of the von Mises

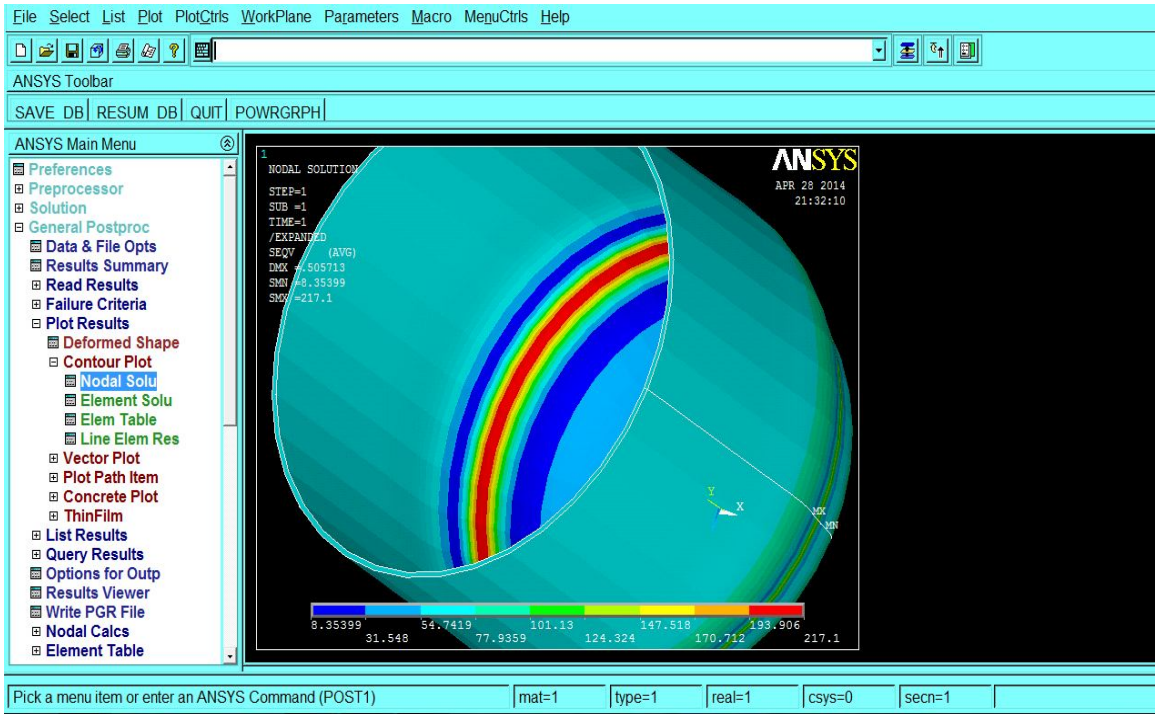



File Select List Plot PlotCtrls WorkPlane Parameters Macro MenuCtrls Help
ANSYS Toolbar
SAVE_DB RESUM_DB QUIT POWRGRPH
ANSYS Main Menu
Preferences
Preprocessor
Solution
General Postproc
Data & File Opts
Results Summary
Read Results
Failure Criteria
Plot Results
Deformed Shape
Contour Plot
Nodal Solu
Element Solu
Elem Table
Line Elem Res
Vector Plot
Plot Path Item
Concrete Plot
ThinFilm
List Results
Query Results
Options for Outp
Results Viewer
Write PGR File
Nodal Calcs
NODAL SOLUTION
STEP=1
SUB =1
TIME=1
SEQV (AVG)
DMX =.505713
SMN =8.35399
SMX =217.1
ANSYS
APR 28 2014
21:23:02
8.35399
31.548
54.7419
77.9359
101.13
124.324
147.518
170.712
193.906
217.1
Pick a menu item or enter an ANSYS Command (POST1)
mat=1
type=1
real=1
csys=0
secn=1


File Select List Plot PlotCtrls WorkPlane Parameters Macro MenuCtrls Help
ANSYS Toolbar
SAVE_DB RESUM_DB QUIT POWRGRPH
ANSYS Main Menu
Preferences
Preprocessor
Solution
General Postproc
Data & File Opts
Results Summary
Read Results
Failure Criteria
Plot Results
Deformed Shape
Contour Plot
Nodal Solu
Element Solu
Elem Table
Line Elem Res
Vector Plot
Plot Path Item
Concrete Plot
ThinFilm
List Results
Query Results
Options for Outp
Results Viewer
Write PGR File
Nodal Calcs
Element Table
NODAL SOLUTION
STEP=1
SUB =1
TIME=1
SEQV (AVG)
DMX =.505713
SMN =8.35399
SMX =217.1
ANSYS
APR 28 2014
21:28:32
8.35399
31.548
54.7419
77.9359
101.13
124.324
147.518
170.712
193.906
217.1
Pick a menu item or enter an ANSYS Command (POST1)
mat=1
type=1
real=1
csys=0
secn=1

4-8:Full Model

The plot of the model can be expanded around the axisymmetric axis to get a better view of the full model.



4-9: Verification of the Results

This is done by comparing the value of the stresses obtained using ANSYS package.

The finite element stresses to the values calculated using the thin-wall equations. If the values are within reason (away from notches, etc.), proceed. For the purposes of failure analysis, we must select an appropriate failure theory. A plot of the von Mises stress is useful for identifying critical locations in the vessel.