# STRESS CONCENTRATION ANALYSIS IN PRESSURE VESSEL USING ANSYS

**R.MAHESWARAN**
**Assistant Professor**
**Department of Mechanical Engineering**
**Unnamalai Institute of Technology**
**Suba Nagar, Kovilpatti, India**

*Abstract:* In this study, we discuss some of the recent developments in the determination of stress concentration factor in the pressure vessels. The literature indicates a growing interest in the field of stress concentration analysis in the pressure vessels. Pressure vessels find wide operations in thermal and nuclear power shops, process and chemical diligence in space, ocean depths and fluid force systems in diligence. The main idea of this study is to design and assay the features of pressure vessels. Colourful parameters of solid pressure vessel are designed and checked and according to the principles specified in American Society of Mechanical Engineers (A.S.M.E) Sec VIII Division 1. The stresses developed in solid wall pressure vessel and Head of pressure vessels.

In general, pressure vessels designed in accordance with the ASME Code, Section VIII, Division 1, are designed by rules and do not require a detailed evaluation of all stresses. It is recognized that high localized and secondary bending stresses may exist but are allowed for by use of a higher safety factor and design rules for details. It is required, however, that all loadings (the forces applied to a vessel or its structural attachments) must be considered.

*Key words:* **Design, Analysis, Pressure vessel, ANSYS.**

## 1. INTRODUCTION

Vessels, tanks and pipelines that carry, store, or receive fluids are called pressure vessels. A pressure vessel is defined as a container with a pressure differential between inside and outside. The inside pressure is usually higher than the outside, except for some isolated situations.

When discussing pressure vessels we must also consider tanks. Pressure vessels and tanks are significantly different in both design and construction: tanks, unlike pressure vessels, are limited to atmospheric pressure; and pressure vessels often have internals while most tanks do not (and those that do are limited to heating coils or mixers).

### 1.1 COMPONENT OF PRESSURE VESSEL
There are three main Type are given below
1. Shell

2. Head

3. Nozzle

### 1.1.1 Shell

The shell is the primary component that contains the pressure. Pressure vessel shells are welded together to form a structure that has a common rotational axis.

### 1.1.2 Head

All pressure vessel shells must be closed at the ends by heads (or another shell section). Heads are typically curved rather than flat. Curved configurations are stronger and allow the heads to be thinner, lighter, and less expensive than flat heads. Heads can also be used in side a vessel. These "intermediate heads" separate sections of the pressure vessel to permit different design conditions in each section. Heads are usually categorized by their shapes. Ellipsoidal, hemispherical, torispherical, conical, toriconical and flat are the common types of heads are shown below.
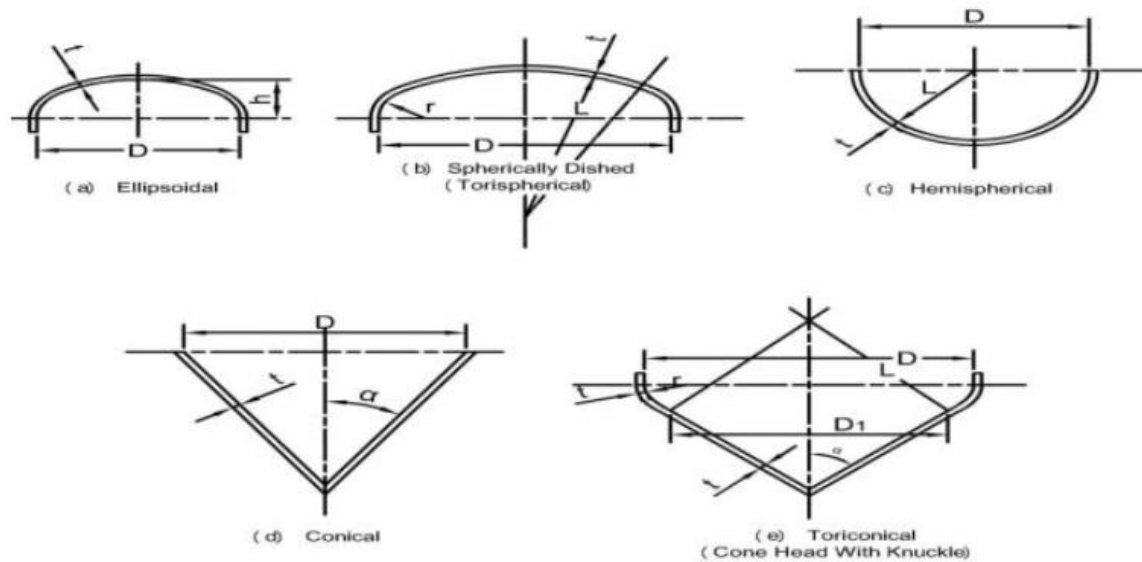
*Fig.1.1 Different types of heads*

### 1.1.3 Nozzle

A nozzle is a cylindrical component that penetrates the shell or heads of a pressure vessel. The nozzle ends are usually flanged to allow for the necessary connections and to permit easy disassembly for maintenance or access. Nozzles are used for the following applications

• Attach piping for flow into or out of the vessel.

• Attach instrument connections, (e.g., level gauges, thermo wells, or pressure gauges).

• Provide access to the vessel interior at midways.

• Provide for direct attachment of other equipment items, (e.g., a heat exchanger or mixer).

• Nozzles are also sometimes extended into the vessel interior for some applications, such as for inlet flow distribution or to permit the entry of thermo wells

## 2. DESIGN PARAMETER OF PRESSURE VESSEL

The following are design parameters of pressure vessel

1. Design Pressure

2. Allowable stress

3. Corrosion Allowance

### 2.1 Design Pressure

In the pressure vessels, three terms related to pressure are commonly used

• Maximum Working pressure is the maximum pressure to which the pressure vessel is subjected.

• Design pressure is the pressure for which the pressure vessel designed

• Hydrostatic test pressure is the pressure at which the vessel is tested. The pressure vessel is finally tested by the hydrostatic test before it is put into operation.

• The design pressure and the hydrostatic test pressure are obtained as follows:

*Design pressure = 1.05\* (Maximum working pressure)*
*Hydrostatic test pressure = 1.3\* (Design pressure)*

### 2.2 Allowable Stress

As per the IS Code and ASME Code, the allowable stress is based on the ultimate tensile strength with a factor of safety of 3 and 4 respectively. As per the IS Code, the following stress is obtained on the yield strength with a factor of safety of 1.
Therefore,
Allowable stress, $\sigma all = Sut/3$ or $\sigma all = Syt/ (1.5)$
Where, $\sigma all$ = allowable tensile stress for the pressure vessel, N/mm2
Sut = ultimate tensile strength for the pressure vessel material, N/mm2
Syt = yield strength for pressure vessel material, N/mm2

### 2.3 Corrosion Allowance

The walls of the pressure vessel are subjected to thinning due to corrosion which reduces the life of the pressure vessel. The corrosion in pressure vessel is due to the following reasons:

• Chemical attack by reagents on the inner wall surface of the vessel.

• Rusting due to atmospheric air and moisture.

• High temperature oxidation.

• Erosion due to flow of reagent over the wall surface at high velocities.

Every attempt should be made avoid the corrosion. However, this may not be always possible. An allowance is, therefore, required to be made by suitable increase in wall thickness to compensate for the thinning due to corrosion.
Corrosion allowance is an additional thickness of the pressure vessel wall over and above that required to withstand the internal pressure.

Guidelines for providing corrosion allowance:

1. For cast iron, plain carbon steel and low alloy steel component, the corrosion allowance of 1.5 mm is provided. However, in case of these chemical industries where severe conditions are expected, the corrosion allowance may be 3mm

2. For high alloy steel and non-ferrous components, no corrosion allowance is necessary.

3. When the thickness of cylinder wall is more than 30mm, no corrosion allowance is necessary.

## 2.4. Factors Considered in Designing Pressure Vessels

1. Dimensions-Diameter, length and their limitations.

2. Operating conditions – Pressure and temperature.

3. Available materials and their physical properties and cost.

4. Corrosive nature of reactants and products.

5. Theories of failure.

6. Types of construction i.e. forged, welded or casted.

7. Method of Fabrication.

8. Fatigue, Brittle failure and Creep.

9. Economic consideration.

### 2.4.1. The design of openings and nozzles is based on two considerations

1. Primary membrane stress in the vessel must be within the limits set by allowable tensile stress.

2. Peak stresses should be kept within acceptable limits to ensure satisfactory fatigue life.

### 2.4.2. Future Considerations

Equipment such as these vessels are normally intended for a specific project and period of use. They should not be used beyond their design life, or adapted to other experimental uses beyond their safe operational parameters. Internal labeling and record keeping controls are extremely important to prevent use beyond the design life and safe operating parameters of each vessel.

## 3. ANSYS Case Study: Axisymmetric Analysis of A Pressure Vessel

The pressure vessel shown below is made of cast iron (E = 14.5 Msi, v = 0.21) and contains an internal pressure of p = 1700 psi. The cylindrical vessel has an inner diameter of 8 in with hemispherical end caps. The end caps have a wall thickness of 0.25 in, while the cylinder walls are 0.5 in thick. In addition, there are two small circumferential grooves of 1/8 in radius along the inner surface, and a 2 in wide by 0.25 in deep circumferential groove at the centre of the cylinder along the outer surface.
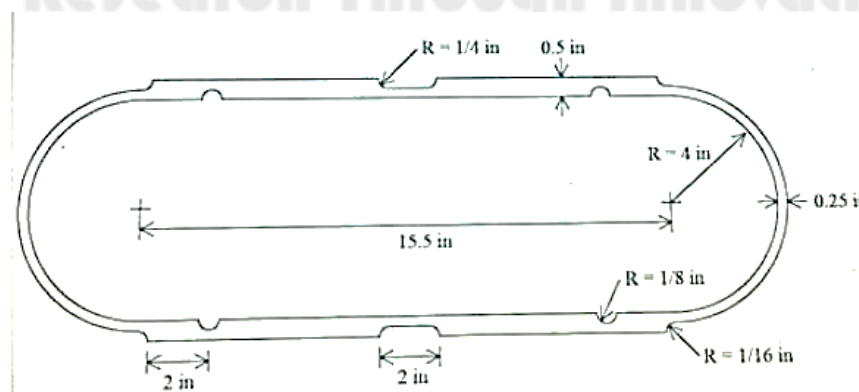


**Figure 3.1** pressure vessel

In this example, ANSYS 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 82) with the axisymmetric option activated. In addition, the vessel is symmetric about a plane through the centre of the cylinder. Thus, only a quarter section of the vessel needs to be modelled.

## 3.1 Formulation using ANSYS:

Start > Programs > ANSYS Mechanical APDL Product Launcher
*Utility Menu >File >Change Jobname…> Enter 'Pressure_Vessel' > OK*
Utility Menu > File >Change Title… > Enter 'Stress Analysis of an Axisymmetric Pressure Vessel' > OK
*ANSYS Main Menu > Preferences > Preferences for GUI Filtering >Select 'Structural' and 'h-method' > OK*

## 3.2 Enter the Pre processor to define the model geometry

Define Element Type (Axisymmetric Option) and Material Properties.
ANSYS Main Menu > Preprocessor > Element Type > Add/Edit/Delete > Add… > Structural Solid Quad 8 node 82 (PLANE82) (define 'Element type reference number' as 1) > OK > Click Options… > Select 'Axisymmetric' for K3 (Element behavior) > OK > Close
ANSYS Main Menu > Preprocessor > Material Props > Material Models > Double Click Structural > Linear > Elastic > Isotropic > Enter 14.5e6 for EX and 0.21 for PRXY > Click OK > Click Exit (under 'Material')
Begin creating the geometry by defining two Circles for the spherical endcap, and Subtract Areas to create the vessel wall.
ANSYS Main Menu Preprocessor > Modeling > Create > Areas > Circle > Solid Circle > Enter 0 for WP X, 0 for WP Y, and 4 for Radius > Apply > Enter 0 for WP X, 0 for WP Y, and 4.25 for Radius > OK
ANSYS Main Menu > Preprocessor > Modeling > Operate > Booleans > Subtract > Areas > Select (with the mouse) Area 2 (bigger circle) > OK > Select Area 1 (smaller circle) > OK
Create Lines through the center of the Circles and Divide the Areas along these Lines.
ANSYS Main Menu > Preprocessor > Modeling > Create > Lines > Lines > Straight line > Click on the Keypoints on the outer circle which are on the X-axis to create a Line parallel to the X-axis (Circles are divided into four arcs by Ansys, with a Keypoint placed at the end of each arc). Similarly, click on the Keypoints on the outer circle which are on the Y-axis to create a Line parallel to the Y-axis > OK
ANSYS Main Menu > Preprocessor > Modeling > Operate > Booleans > Divide > Area by Line > Select (with the mouse) the remaining Area (annulus)> OK > Select the two Lines that we have created > OK
ANSYS Main Menu > Preprocessor > Modeling > Delete > Area and Below > Select the three Areas in the first, second, and third quadrants > OK

Define two Rectangles to create the walls of the cylindrical portion of the vessel (thick and thin sections). Define a Circle to create the circumferential groove on the inside of the vessel.

ANSYS Main Menu > Preprocessor > Modeling > Create > Areas > Rectangle > By Dimensions > Enter 4 and 4.5 for X-coordinates and 0 and 7.75 for Y-coordinates > Click Apply > Enter 4.25 and 4.5 for X-coordinates and 6.75 and 7.75 for Y-coordinates > OK
ANSYS Main Menu > Preprocessor > Modeling > Create > Areas > Circle > Solid Circle > Enter 4 for WP X, 2 for WP Y, and 1/8 for Radius > OK
Subtract Areas to eliminate unused segments, and then Add all Areas to create a single Area for meshing.

ANSYS Main Menu > Preprocessor > Modeling > Operate > Booleans > Subtract > Areas > Select (with the mouse) the bigger rectangle > OK > Select the small rectangle and circle > OK
ANSYS Main Menu > Preprocessor > Modeling > Operate > Booleans > Add > Areas > Select 'Pick All' > OK
Create Line Fillets at the two transitions between the thick and thin sections.
Utility Menu > Plot > Lines
Utility Menu > Plot Ctrls > Numbering… > Click 'Line numbers' On > OK

ANSYS Main Menu > Preprocessor > Modeling > Create > Lines > Line Fillet > Select (with the mouse) the two Lines near the lower Fillet > OK > Enter 1/16 for Fillet radius > Apply > Select the two Lines near the upper Fillet > OK > Enter 1/4 for Fillet radius > OK
Create Areas within the two Fillets and add these Areas to the main Area. First zoom in on the area of interest using the plot controls.

ANSYS Main Menu > Preprocessor > Modeling > Create > Areas > Arbitrary > By Lines > Select (with the mouse) the Fillet and adjacent two Lines > OK
Repeat for the other Fillet

ANSYS Main Menu > Preprocessor > Modeling > Operate > Booleans > Add > Areas > Select 'Pick All' > OK
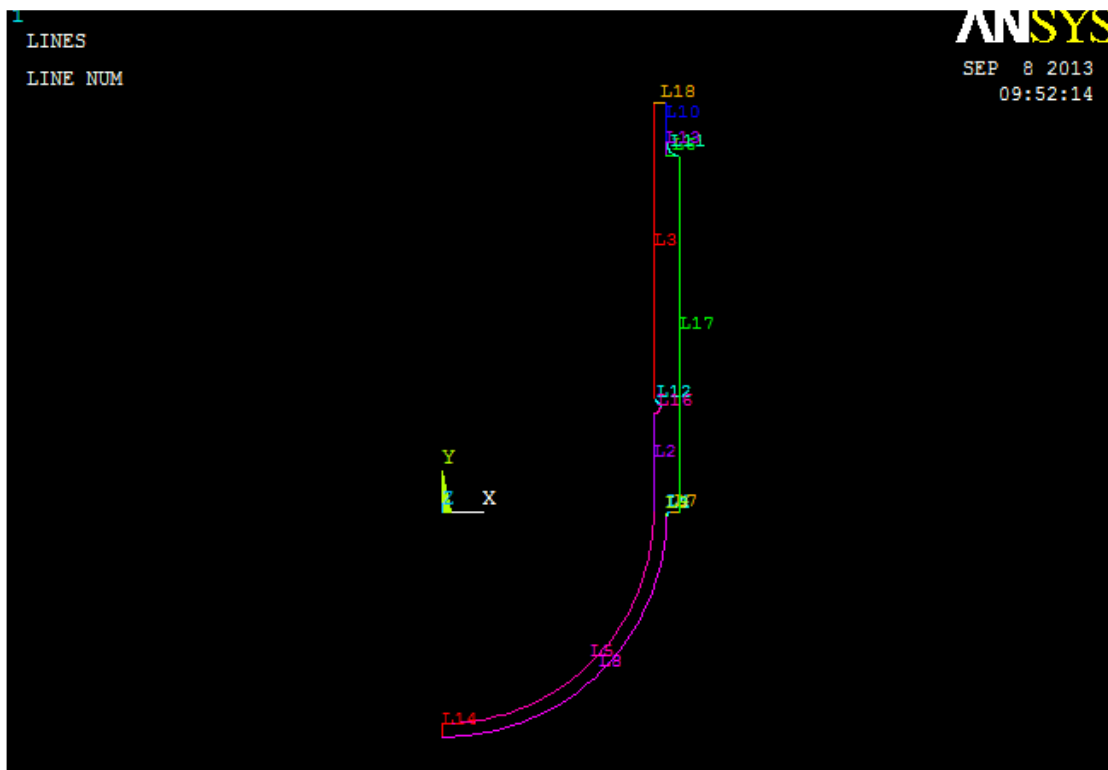Utility Menu > Plot > Lines

**Figure 3.2** line dig. of pressure vessel

In this example, the irregular geometry will be Free Meshed with Quad Elements. Better control of Element sizing and distribution can be obtained with Mapped Meshing, but this would require that additional sub-Areas be defined within the main Area that have a regular (four-sided) geometry. Using Free Meshing, all Elements in the model will be approximately the same size. In the first run, we will choose a Global Size (approximate Element edge length) of 0.1 in.

ANSYS Main Menu > Preprocessor > Meshing > Mesh Tool > Under 'Size Controls: Global' click Set > Enter 0.1 for 'Element edge length' > OK > Under 'Mesh:' select Areas, Quad and Free > Click Mesh > Select (with the mouse) the Area > OK
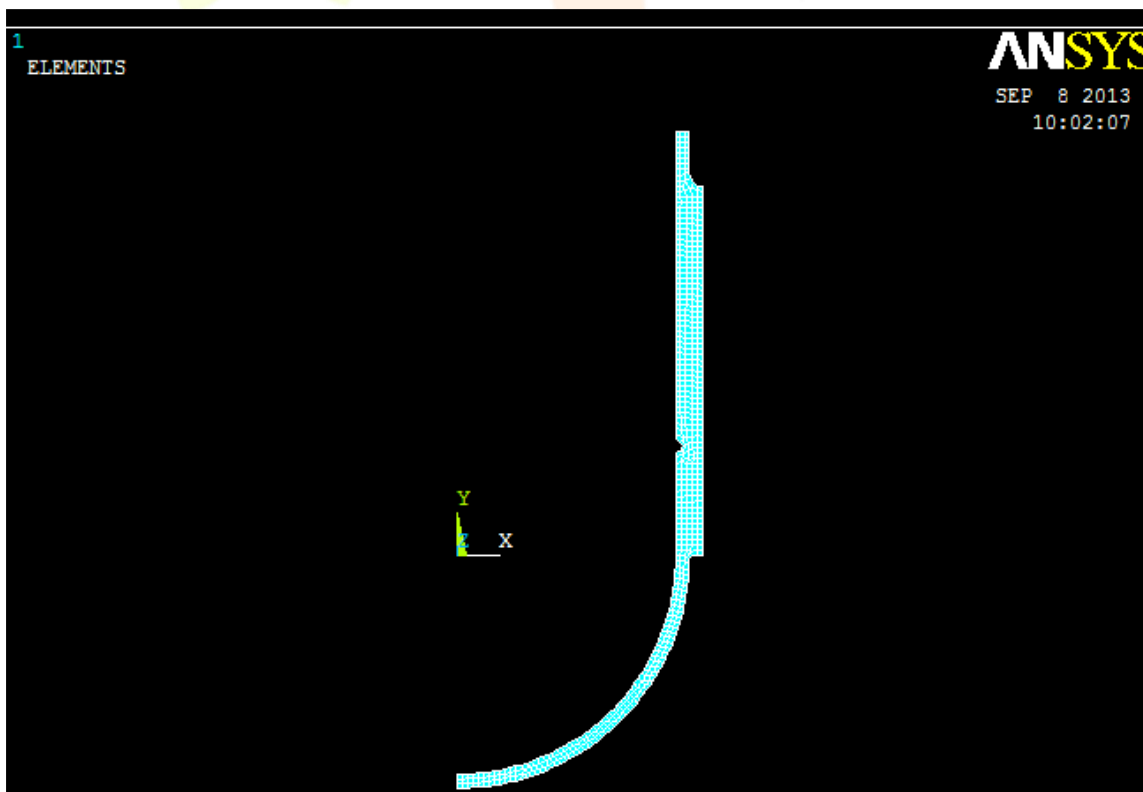


**Figure 3.3** Pressure vessel with Meshing

**3.3 Enter the Solution Menu to define boundary conditions and run the analysis**

ANSYS Main Menu > Solution > Analysis Type > New Analysis > Select Static > OK

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 axisymmetry (ANSYS automatically makes the necessary adjustment of Loads in an Axisymmetric model).

ANSYS Main Menu > Solution > Define Loads > Apply > Structural > Displacement > Symmetry B.C. > On Lines > Select the Line on top of the model (18) > OK

ANSYS Main Menu > Solution > Define Loads > Apply > Structural > Pressure > On Lines > Select (with the mouse) all the Lines on the inside of the vessel (3, 12, 16, 2 and5) > OK > Enter 1700 for 'Load PRES value' > OK

The pressure will be indicated by arrows, as shown above in the figure on the right.
Save the Database and initiate the Solution using the current Load Step (LS).
ANSYS Toolbar > SAVE_DB

ANSYS Main Menu > Solution > Solve > Current LS > OK > Close the information window when solution is done > Close the /STATUS Command window
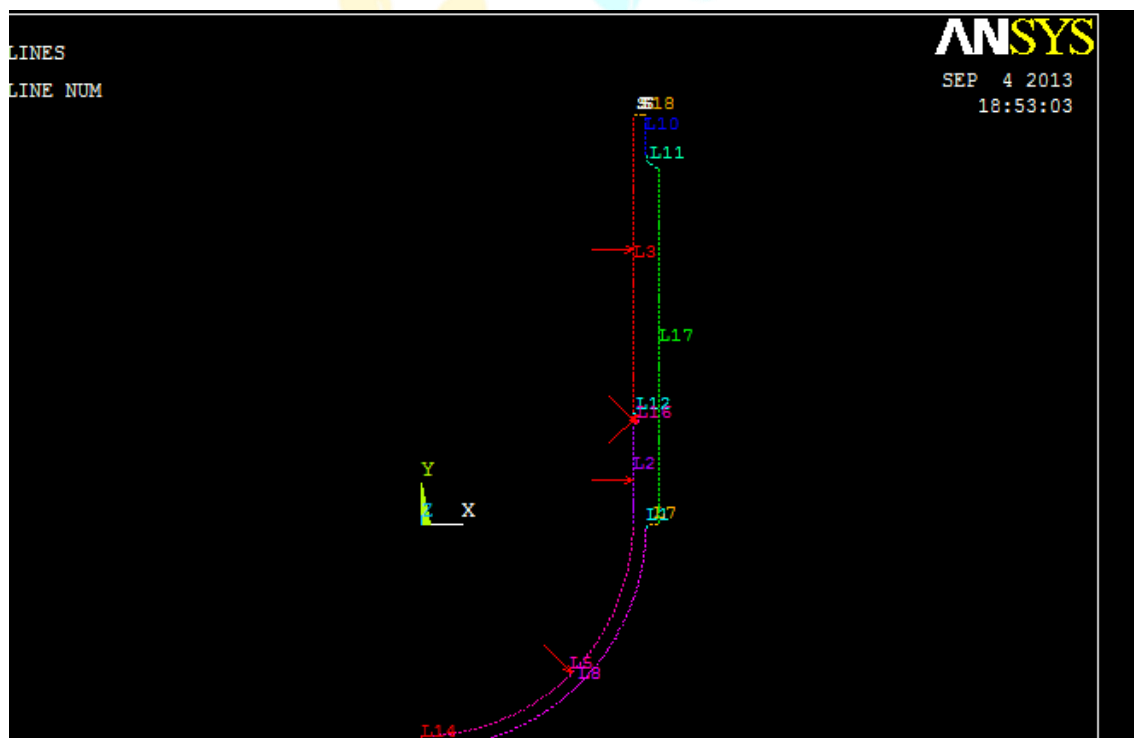


**Figure 3.4** pressure vessel with load

**3.4 Enter the General Postprocessor to examine results**

First, plot the Deformed Shape.
ANSYS Main Menu > General Postproc > Plot Results > Deformed Shape > Select Def undeformed > OK
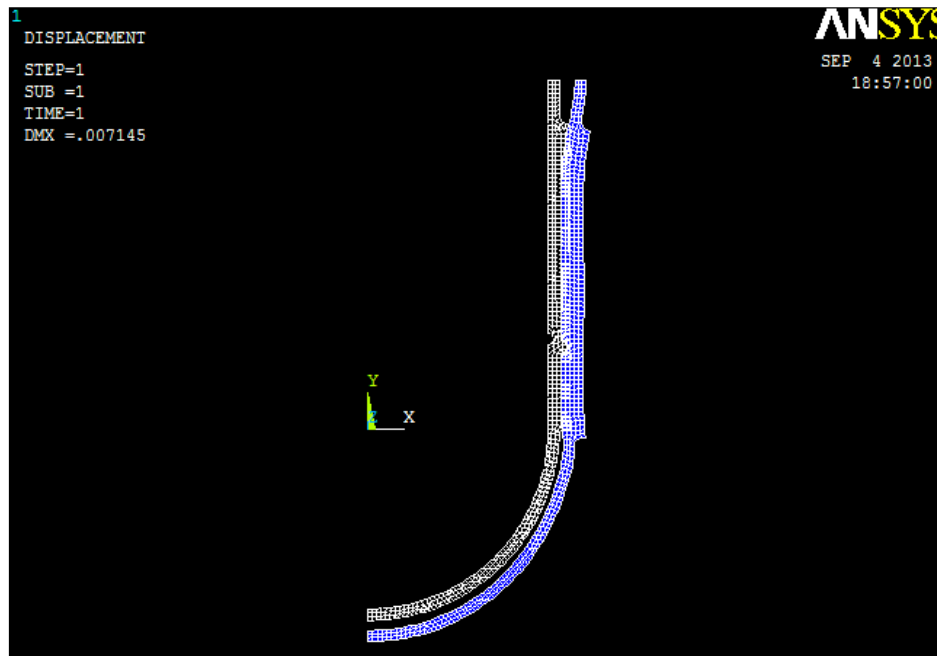
**Figure 3.5** deformed shape

A Contour Plot of any stress component can be created. The radial, hoop (tangential), and longitudinal stresses should be checked to verify the model. Also, stress values at any particular node can be checked by using the "Query Results" command, selecting the desired component, and then picking the appropriate node. For this model, along the cylindrical portion of the vessel, x represents the radial direction, y represents the longitudinal direction, and z represents the hoop (tangential) direction. Power graphics must be disabled to query results at nodes.
ANSYS Toolbar > POWRGRPH > Select OFF > OK
ANSYS Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solu > Select 'Stress' and 'X-Component of stress' (or Y or Z) > OK
ANSYS Main Menu > General Postproc > Query Results > Nodal Solution > Select 'Stress' and 'X-direction SX' (or SY or SZ) > OK > Select Nodes in the region of interest (may be helpful to zoom in on region)
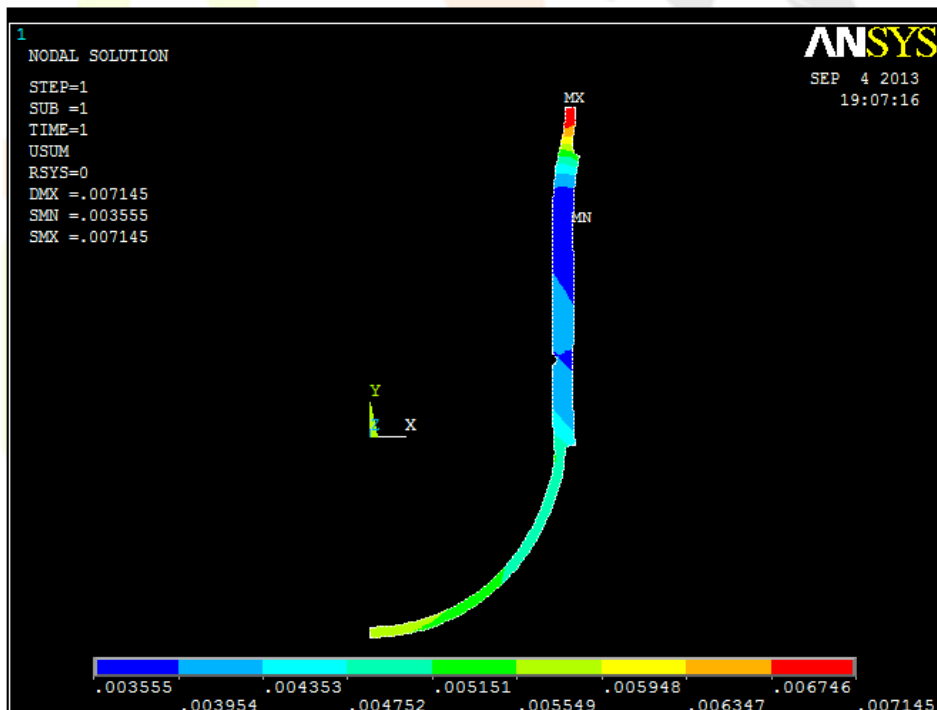


**Figure 3.6** displacement diag.

Compare 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. However, since the vessel is made of cast iron (brittle material), the "Maximum-Normal-Stress" failure criterion may be more appropriate (or Coulomb-Mohr or other similar failure theories). Create Contour Plots of the von Mises and 1st Principal stresses.

ANSYS Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solu > Select 'Stress' and 'von Mises stress' > OK

ANSYS Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solu > Select 'Stress' and '1st Principal stress' > OK
The plot of the model can be expanded around the axisymmetric axis to get a better view of the full model. For this plot, Powergraphics must be enabled.

ANSYS Toolbar > POWRGRPH > Select ON > OK
Utility Menu > PlotCtrls > Style > Symmetry Expansion > 2-D Axi-Symmetric > Select 'Full expansion' > OK

## 4. Numerical Analysis: Axisymmetric Analysis of a Pressure Vessel

### 4.1 For Hemispherical Head

$Pd = 1.05Pi$

$t = Pd*Di/(4\sigma all*\eta) - 0.4Pi$

$6.35 = 2210*203.2/(4*\sigma all*1) - 0.4*2210$

$\sigma all = 17901 \ N/mm2$

$\sigma all = \epsilon E$

$\epsilon = 17901/14.5e6$

$dl/l = 1.23e-3$

$dl = 1.23e-3* 8.25$
$dl = 0.01018 \ mm$

### 4.2 For Cylinder

$\sigma t = Pi(Do2+Di2)/(Do2-Di2)$
$\sigma t = 1700(215.92+203.22)/(215.92-203.22)$
$\sigma t = 28774 \ N/mm2$
$\sigma t = \epsilon E$
$\epsilon = 28774/14.5e6$
$dl/l = 1.9844e-3$
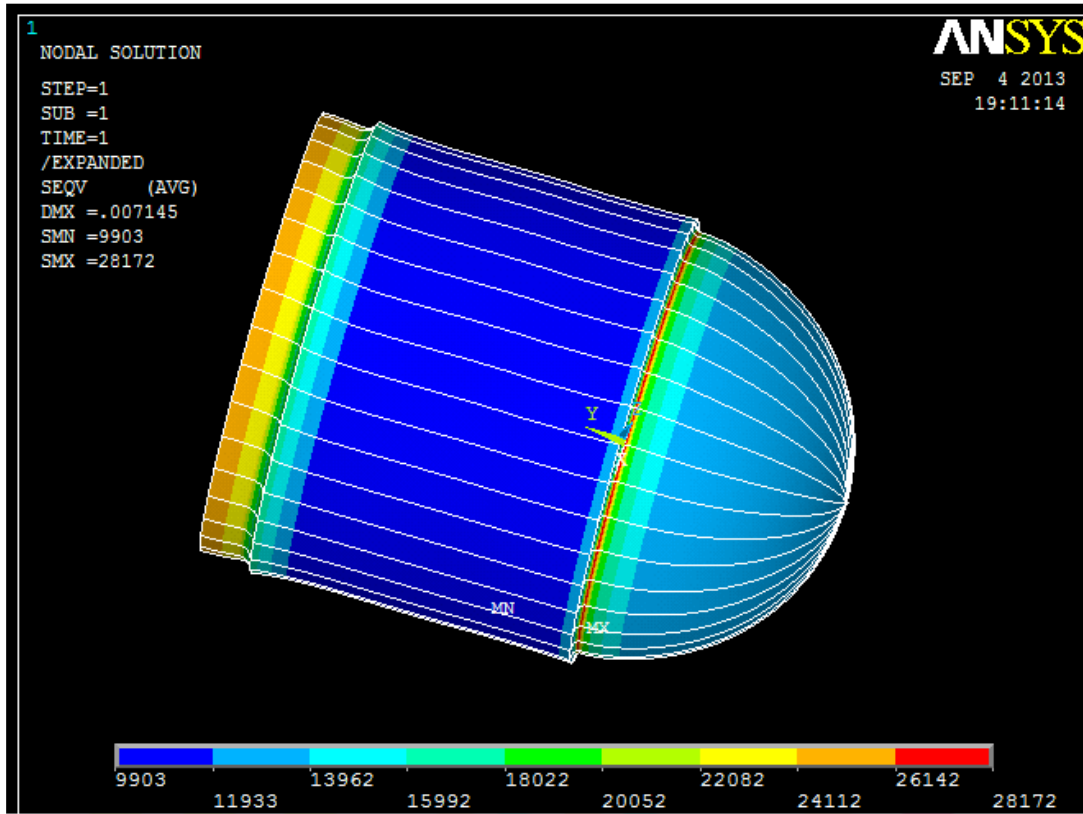$dl = 1.9844e-3* 15.5$
$dl = 0.03 \ mm$

**5. RESULTS & DISCUSSION**



**Figure.5.1** Maximum Stress in Cylinder is tensile stress σt= 28172N/mm2



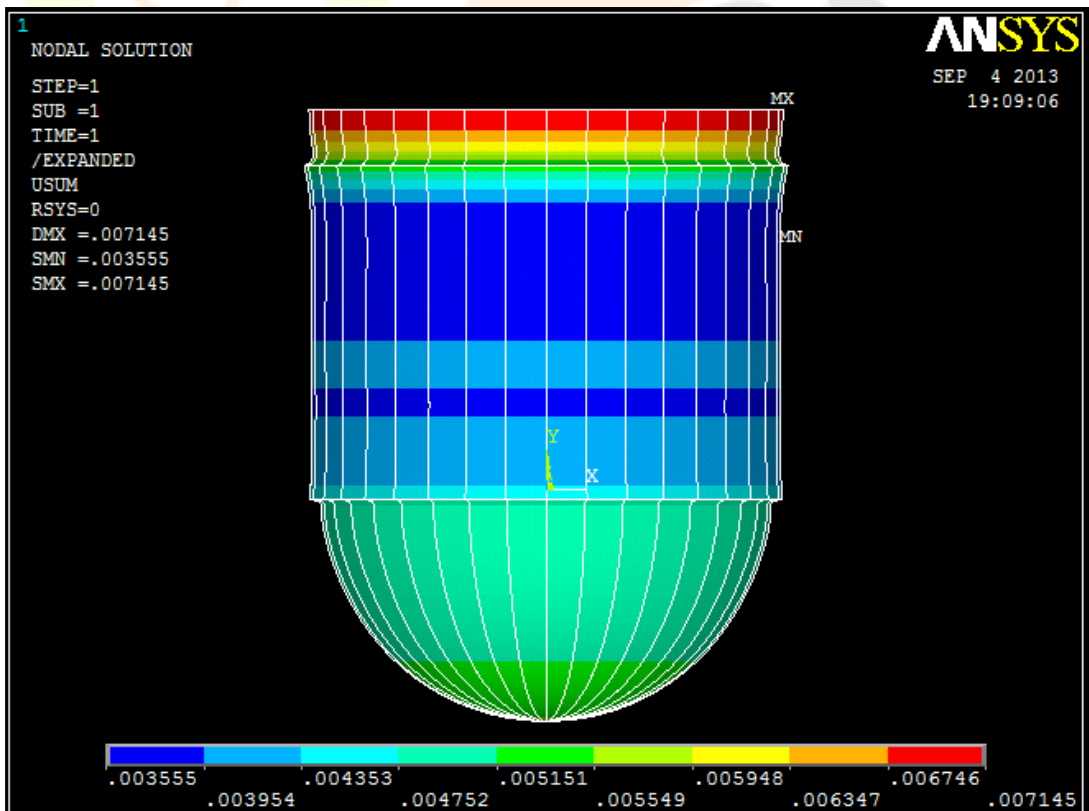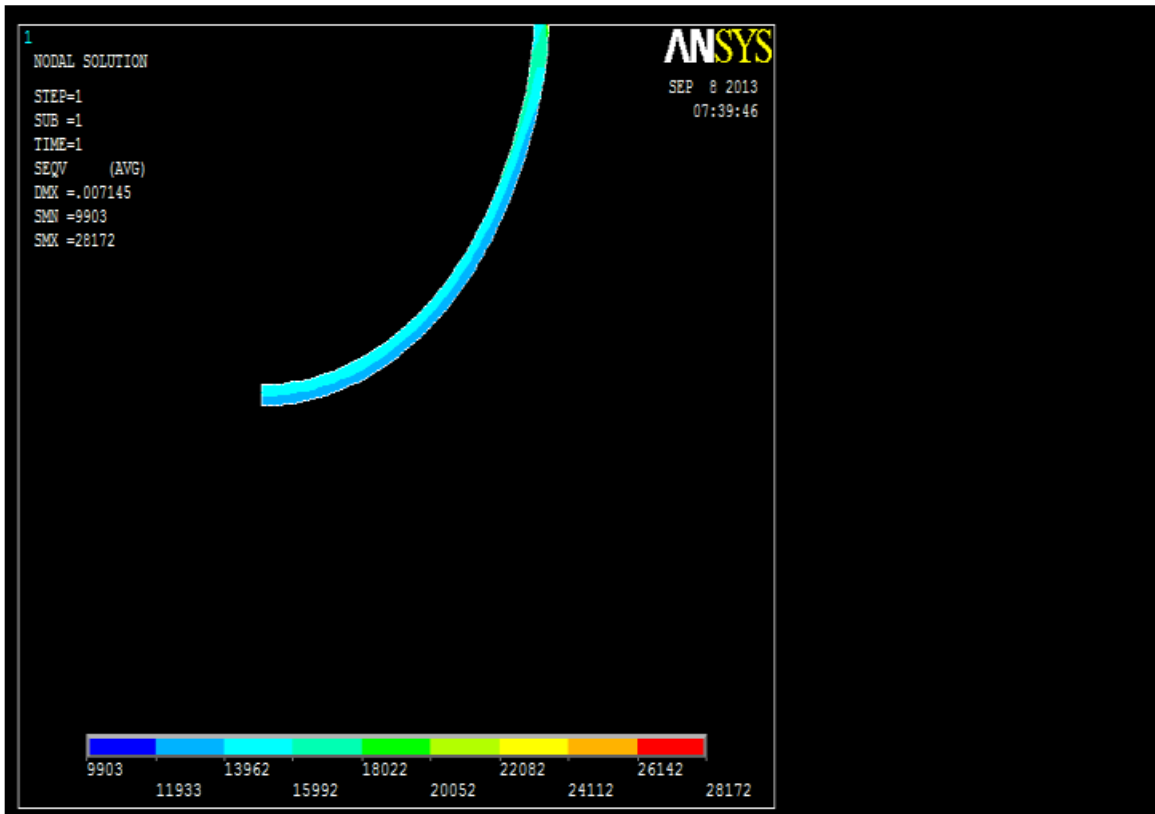**Figure.5.2** Maximum Displacement in Cylinder=0.007145 mm

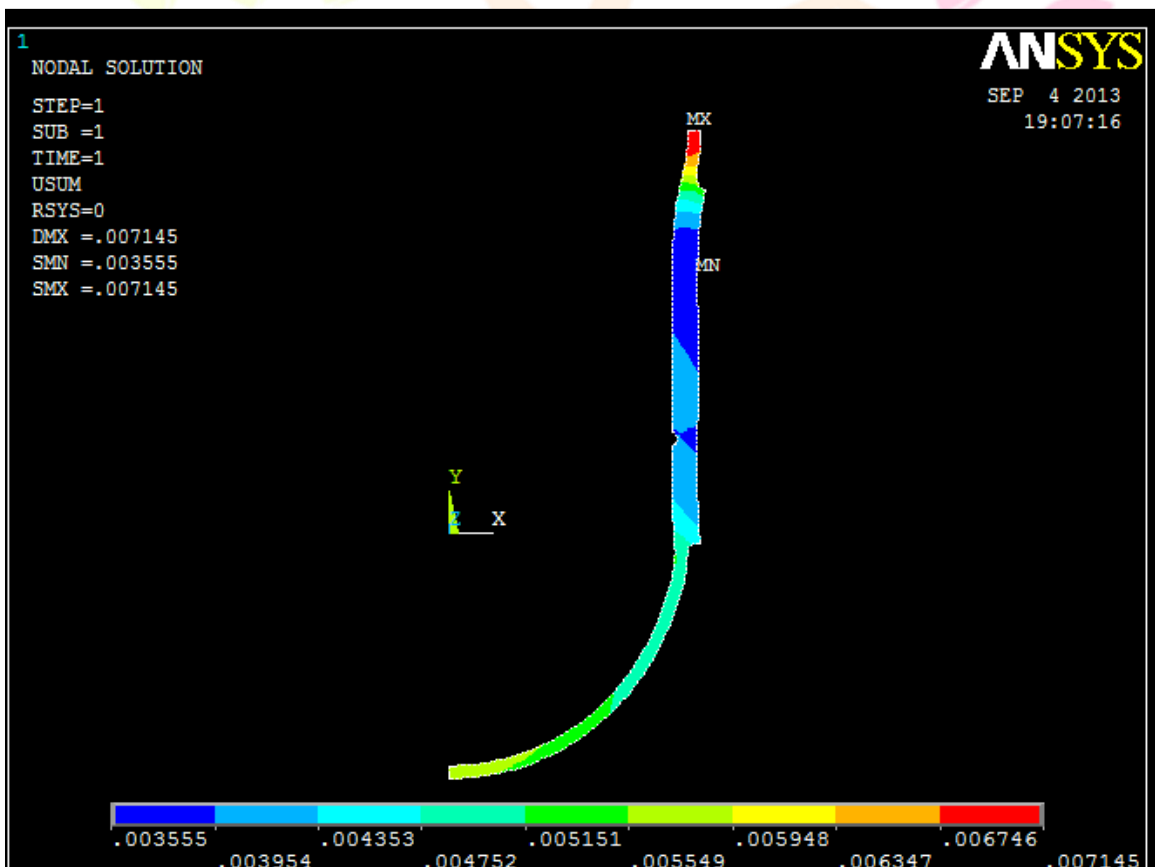**Figure.5.3** Maximum Stress in Head is tensile stress σt= 15992N/mm2



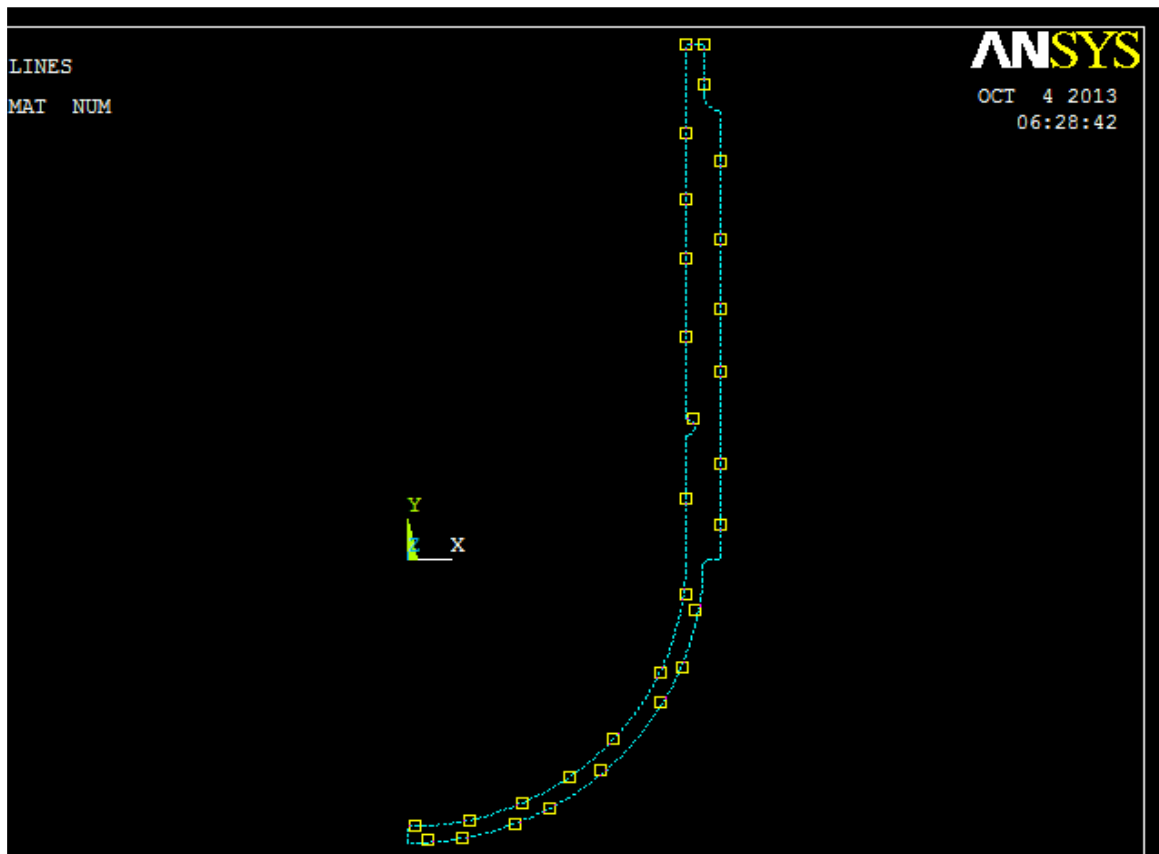**Figure.5.4** Maximum Displacement in Head =0.005151 mm

**Figure.5.5** Path under consideration
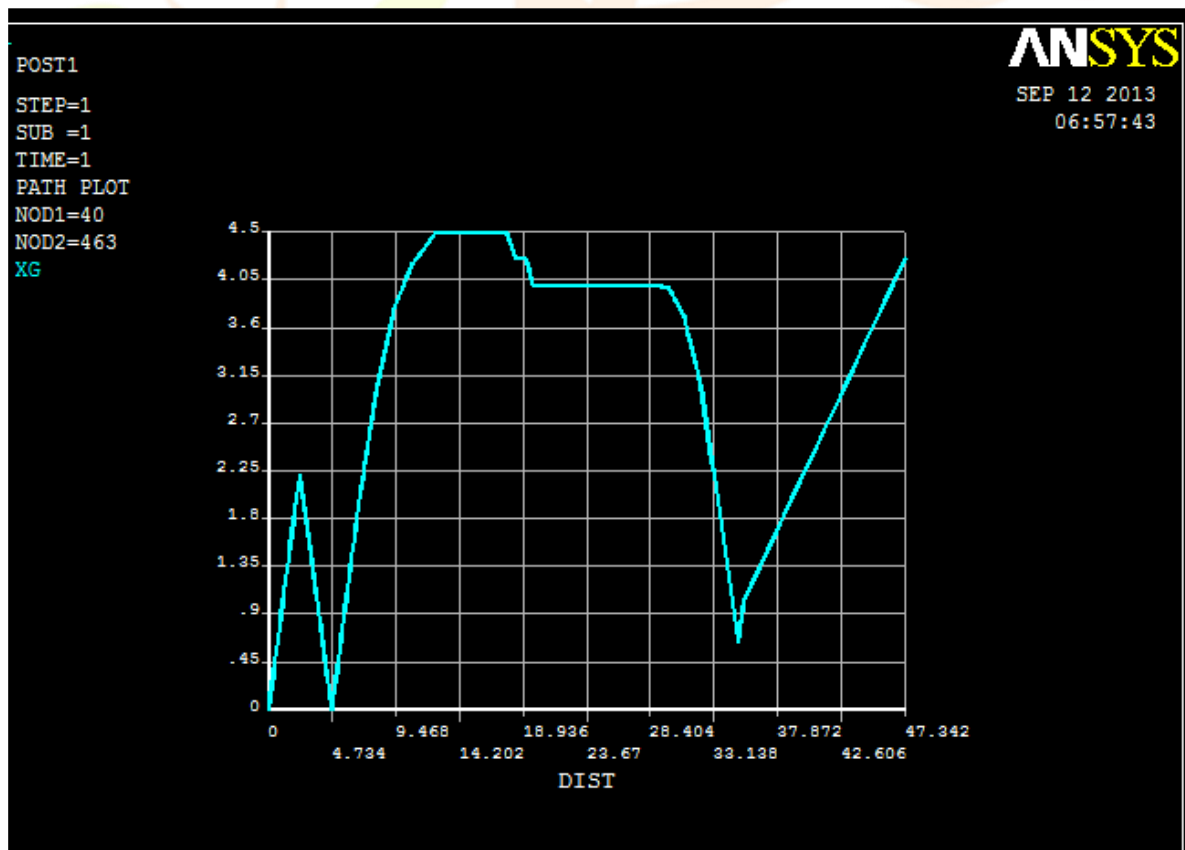


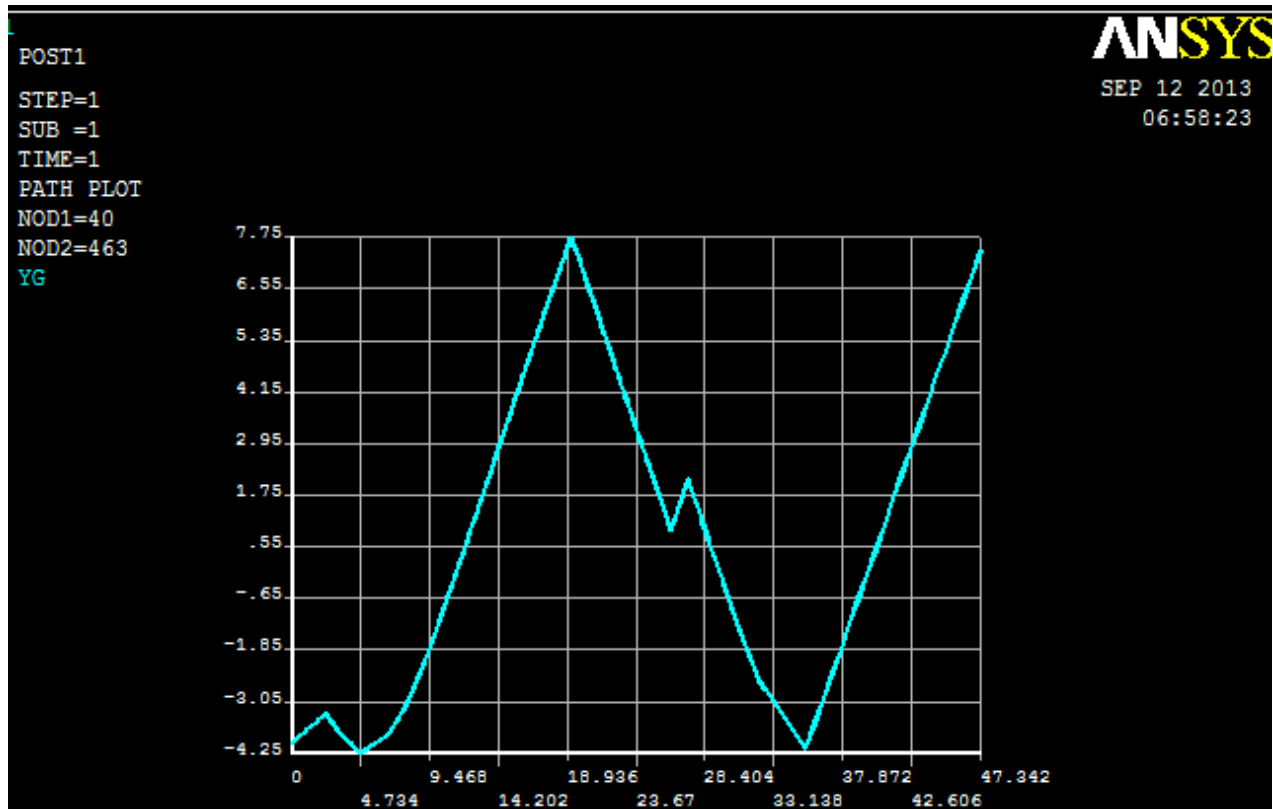**Figure.5.6** Stress in X Direction

**Figure.5.7** Stress in Y Direction
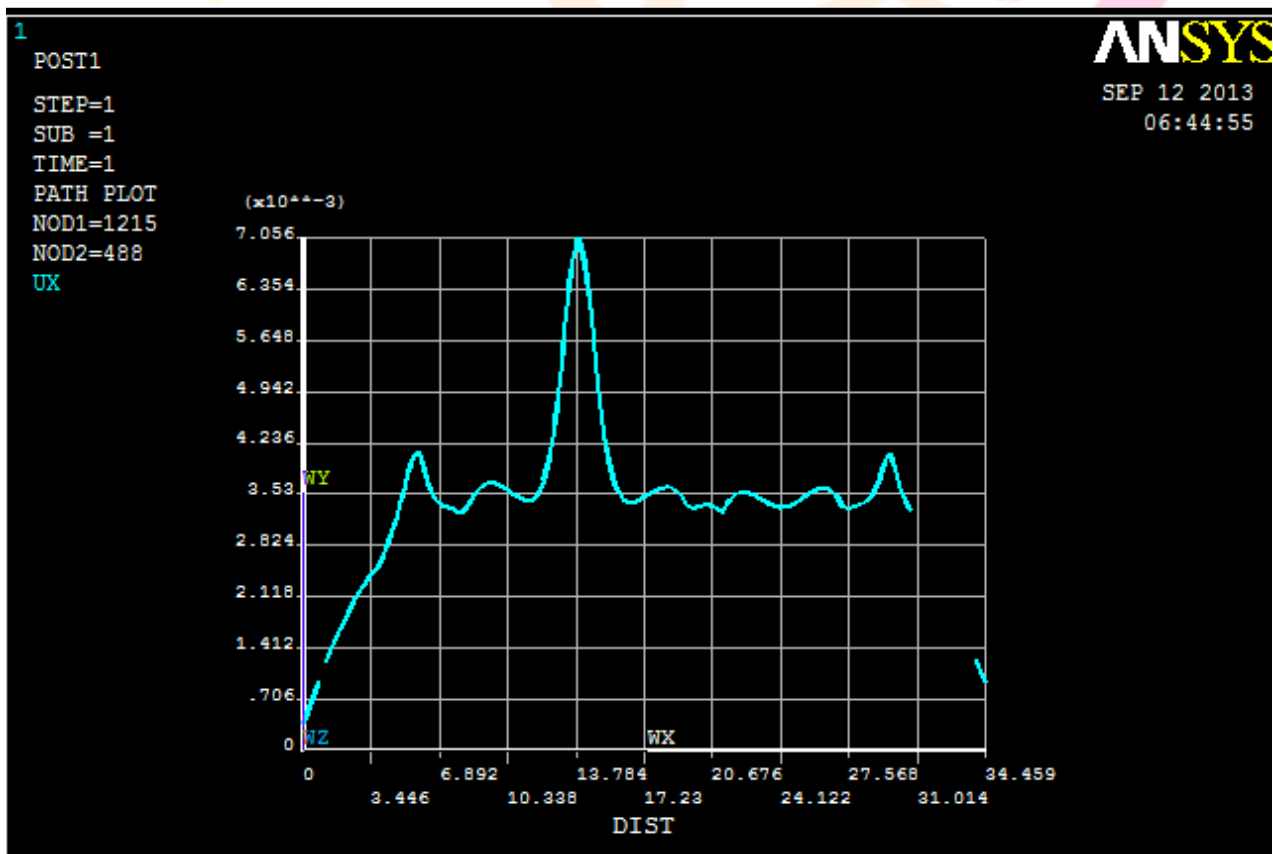


**Figure.5.8** Displacement Graph

**Table 5.1** Comparison of results in ANSYS with Numerical

|  | ANSYS | | NUMARICAL | |
|---|---|---|---|---|
|  | HEAD | CYCLINDER | HEAD | CYCLINDER |
| DISPLACEMENT (mm) | 0.005151 | 0.007145 | 0.01018 | 0.03 |
| STRESS N/mm2 | 15992 | 28172 | 17901 | 28774 |

## 7. CONCLUSION

1. Various parameters of Pressure vessels are designed and checked according to the principles specified in American Society of Mechanical Engineers (A.S.M.E) Sec VIII Division 1.

2. Theoretical calculated values by using different formulae are very close to that of the values obtained from ANSYS. This indicates that analysis is done with a valid model of pressure vessels.

3. Analysis of cylindrical pressure vessel with hemispherical head type is performed and maximum equivalent stresses found by ANSYS are compared with theoretical values. It is concluded that smaller values of equivalent stresses are appearing in pressure vessel with hemispherical heads, and equivalent stress distribution is advantageous too in that case of head geometry.

## REFERENCES

[1] Introduction to Finite Element in Engineering, Chandrapatala, Belgunda, third edition PHI; 2012; pp276-286

[2] Design of Machine Elements, V.B. Bhandari; *Tata Mc Graw Hill Publication*; 2012;

[3] Mechanical System Design, R.B. Patil; Tech. *Max Publication*; 2012;

[4] Finite Element Analysis – Theory and Applications with ANSYS, Saeed Moaveni; 1999;

[5] Pressure Vessel Design Guides & Procedures. G. Ghanbari M.A.Liaghat, 2000;

[6] Calculations of Plastic Collapse Load of Pressure Vessel Using Fea, Peng-Fei Liu,Jin; Institute of Chemical Machinery and Process Equipment, Zhejiang University, Hangzhou 310027, China)

[7] Stress Concentration At Openings In Pressure Vessels – A Review, Avinash Kharat1, V.V. Kulkarni2; Research Scholar, Dept. of Production Engineering, K. I. T."S.

[8] Stresses In A Pressurized Ribbed Cylindrical Shell with A Reinforced Hole, A. J. Durelii & V. J. Parks The Journal of Strain Analysis For Engineering Design, 1973, **8**, Pp-140-150.;

[9] Pressure Vessel - User's Design SpeciCation, Derek Shuman1, Sara C_Arcel2 A. Mart__Nez2 Berkeley Ca, Usa 2instituto De Fsica Corpuscular (Ific), Csic, Univ. De Valencia, Valencia, Spain, 2 April 2012

[10] Stress Analysis of Reactor Nozzle to Head Junction, A. Hardik B. Nayak and B. R. R. Trivedi; A. Assistant Professor, Mechanical Engg. Department, Faculty of Engineering, Technology and Research; Bardoli B. Professor, Mechanical Engg. Department, Institute of Technology, Nirma University, Ahmadabad;

[11] Unfired Pressure Vessels, Background To The Rules In Part 3 Design Editors: Guy Baylac Consultant and Technical Advisor To Eperc 114 Avenue Félix Faure F-75015 Paris Danielle Koplewicz Technical Director Union De Normalization De La Mécanique F-92038 Paris La Defense Issue 2 – 20 August 2004

[12] Pallavi J.Pudke, Dr. S.B. Rane and Mr. Yashwant T. Naik, Design and Analysis Of Saddle Support: A Case Study In Vessel Design and Consulting Industry, *International Journal of Mechanical Engineering and Technology*, 4(5), 2013, pp. 139-149.

[13] Design and Analysis of Multilayer High Pressure Vessels, Siva Krishna Raparla1 ,T.Seshaiah2 1 Pg Student, 2 Associate Professor Department of Mechanical Engineering Qis College of Engineering &Technology Ongole, Andhra Pradesh.