Finite Element Analysis of Ellipsoidal Head Pressure Vessel

Vikram V. Mane*, Vinayak H.Khatawate.**Ashok Patole***

* (Faculty; Mechanical Engineering Department, Vidyavardhini's college of Engineering. and Technology, Vasai (w),
University of Mumbai,
Email: vikmane1973@yahoo.co.in)
** (Asst. Prof. (PIIT, New Panvel),University of Mumbai)
*** (Asst. Prof. (PIIT, New Panvel),University of Mumbai)

ABSTRACT

Pressure vessels are a commonly used device in engineering. The stress Analysis with with the help of Finite Element Method and Experimental Method is necessary to find out the exact behavior of pressure vessel .To find out the stresses at the interaction between pressure vessel cylinder walls and end caps; When modeled correctly, F.E.A. proved to be useful, but the operator also needs to be able to interpret the results properly. For the validation of F.E.A. result; the measurement of stresses by the experimental method is necessary. The analyst should be able to approximate the solution using classical methodology (hand calculations) in order to verify the solution. This project set out to explore applicable methods using finite element analysis in pressure vessel analysis.

Keywords – Air Booster Unit, Air Reservoir tank

1. INTRODUCTION

Pressure vessels are leak proof containers may be of any shape and size ranges from cold drink bottles to high-pressure steam boilers, used in engineering construction. The word 'design' means to create, to innovate and used not only for the calculation of the detail dimensions of the vessels but also to find out the mode of failure or damage, and selection of materials type and its environmental behavior.

The increasing use of pressure vessel under different conditions of pressure, temperature, operation cycle, contained fluid properties etc., has given special importance to analytical and experimental method for determining their operating stresses. Development in nuclear and chemical industry places new demand of finding most likely method of damage, method of stresses analysis and its reliability of result. The pressure vessel is designed for a 50-kg/cm^2 of internal pressure, 1bar atmosphere of external pressure, and a maximum working temperature of 70 degree Celsius. It is a Horizontal type of pressure vessel. Pressure vessel is the main body of the pneumatic Booster. An operating pressure of 50 Kg/cm^2 for testing of pressure vessels and other application.

Fig. 1.1 Air Booster unit.

2. MECHANICAL DESIGN OF AIR BOOSTER RESERVIOUR TANK

2.1 Shell Design [UG-27] [2]

If a thin circular cylinder is subjected to the action of radial forces uniformly distributed along its circumference, hoop stress will be produced throughout its thickness in tangential direction as given by Harvey [1].

$$\sigma_H = Pr /t, \qquad \sigma_L = Pr / 2t$$

Where P = Internal Pressure, r = Radius of shell,t = Thickness σ_H = Hoop stress,σl = longitudinal stress

Taking fabrication and inspection quality into account, ASME has suggested modified formula for finding thickness for given pressure.

i) Circumferential Stress (Longitudinal Joint) [1, 2,3]

t = PR / (SE – 0.6 P) or p = SET / (R + 0.6t)

ii)Longitudinal Stress (Circumferential Joint) [1, 2,3]

t = PR / (2SE + 0.4P) or p = 2SE t / (R – 0.4 t)

Where,

t = Minimum Vessel Thickness, mm,R = Inside radius of shell, mm,E = Joint Efficiency,P = Design Pressure, Mpa,S = Maximum allowable stress, Mpa

Similarly, there are formula's for design of heads, other component given by ASME.

2.1.1 Design Data

Operating pressure = 3 – 5 N/mm^2 (shell side)

Design pressure = 5 N/mm^2

Inside diameter = 295 mm

Shell material – SA 516 GR.70

Permissible stress, S = 138 N/mm^2

welding efficiency, E = 1 (for R.T)

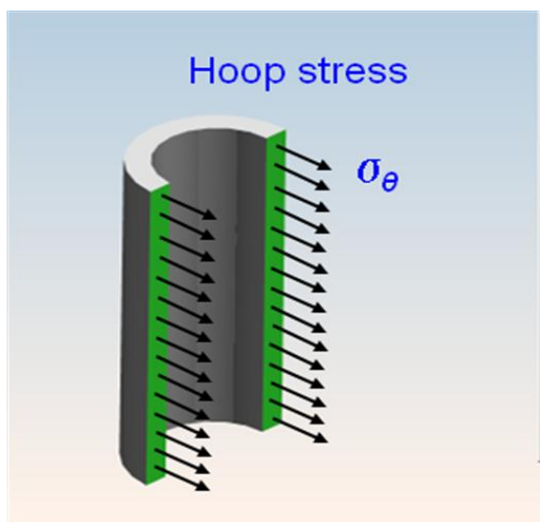**2.1.2 Shell Thickness (UG – 27):**

For circumferential stress

Fig.2.1 Hoop stresses in shell

t = PR / (SE – 0.6 P) or p = SET / (R + 0.6t) (i)

Thickness (t) = $\frac{5 \times 147.5}{138 \times 1 - 0.6 \times 5} + 0$

= 5.463 mm

2.1.3 Longitudinal Stress:
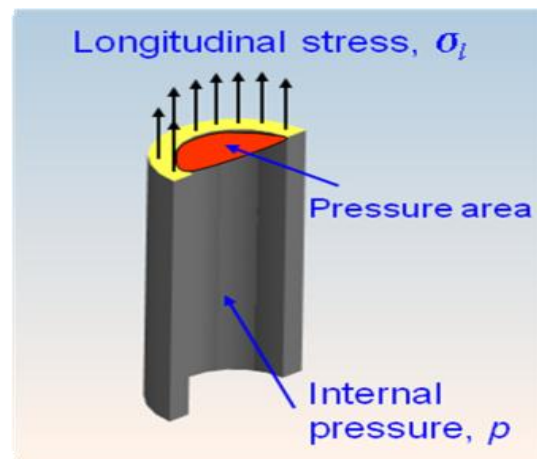
t = PR / (2SE + 0.4P) or p = 2SE t / (R – 0.4 t)   (ii)

Fig. 2.2 Longitudinal stresses in shell

Thickness (t) = $\frac{5 \times 147.5}{2 \times 138 \times 1 + 0.4 \times 5} + 0$

= 2.65 mm.

2.2 Ellipsoidal Head (Thickness Calculation) [UG - 32] [2]

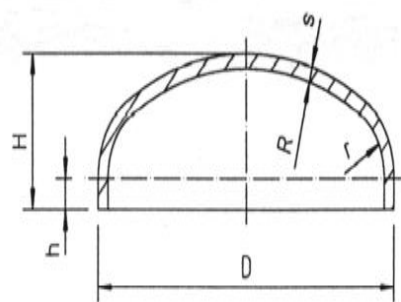Material: use SA516 Gr 70, E =1, Allowable Stress, S = 138 N/mm^2. P = 5 N/mm^2.
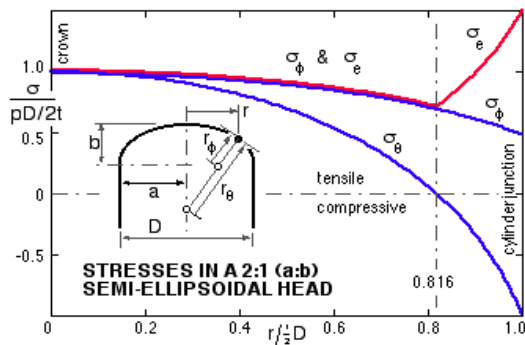

Fig.2.3 Ellipsoidal Head

Fig.2.3 Stress distribution in Ellipsoidal Head

$t = PDK / (SE – 0.2 P)$ (iii)

Where K is spherical radius factor [3

$K = [2 + (a/b)^2] / 6.$

Where, a-Major axis of ellipsoidal head, and

, b-Minor axis or depth of head.

$K = [2 + (295/2 \times 73.75)^2]/6$

$K = 1$

Thickness of Head $(t_h) = \frac{5 \times 295 \times 1}{138 \times 1 - 0.2 \times 5} + 0$

            = 5.348 mm.

3 Finite Element Analysis of Pressure Vessel [6] [7] [9]

Because of the complicated shape of the shell, stress analysis by using photo-elasticity will also be difficult. Stress analysis by finite element method is obviously the best choice. Hence finite element technique has been selected for the analysis purpose.

There are different types of commercial F.E.M. software's available in the market. ANSYS FEM software is one of the most popular commercial software is used for the Finite element analysis of the pressure vessel.

3.1 Finite Element Analysis using ANSYS FEM Software Package [6][9]

In Ansys there are three processors that are used in finite element analysis ;

i)Preprocessor
ii)Solution or Processor .
iii)General Postprocessor.

i) Preprocessor: In preprocessor following steps involved.

-Define element type.
-Define element real constant.
-Define material properties
-Create model or geometry.
-Define meshing controls
-Mesh the object created.

ii) Solution or Processor:

The solution processor has the commands that allow you to apply boundary conditions and loads and it solves for nodal solutions.

iii) General Postprocessor:

It contains the commands that allow you to list and display the result of analysis.

3.2. Finite Element Analysis of Pressure Vessel Using ANSYS
i. Define the Element [6] [9]:

Ansysis provides more than 150 various elements to be used to analyze different problems. Selecting the correct element type is a very important part of the analysis process. IN ANSYS each element type is identified by category name followed by a number.

PLANE -42 is a four node quadrilateral element used to model structural solid problems. The element is defined by four nodes having two degrees of freedom at each node. Translation in x and y direction.

Preprocessor →Element type→ Add /Edit/ Delete.

ii. Define Element Real constants.

Element real constants are quantities that are specific to a particular element. It is important to realize that real constants vary from one element type to another; not all the element requires real constants. Real constants can be defined by the command

Preprocessor →real constants → add/edit/delete.

iii.Define Material Properties.

At this point we have to define the material physical properties of the material used for the given problem. The material use for the construction of pressure vessel is SA516GR.70. and its properties are as shown in Table 1.

Table 1

Material Properties SA516GR.70

Sr.	Properties	Values
1	Modules of elasticity	2×10^5 N/mm^2
2	Poison's ratio	0.3
3	Density of material	7833 Kg/m^3

You can define material properties by command;

Preprocessor →Material props→ Material models.

iv. Create the model Geometry

Geometric modeling is the process of generating three dimensional objects of the real world for the purpose of analysis, design, drafting and manufacture etc. Geometry of the part can be created in several different ways. These include starting from primitives. A key philosophy in

geometric modeling is to sketch sections on work plane or on a face of an object and modify the dimensions later. There are three types of geometric models commonly used to represent physical objects. They are as follows:

a) Wire-frame modeling
b) Surface modeling
c) Solid modeling

3.3 Modeling of Problem

For the particular problem, the model was prepared by using solid works graphic software as shown in Fig.3.1.

Fig. 3.1 Shows geometric model of section of Shell with Head. Having uniform thickness 6 mm.

3.4 Define Meshing Control [6] [9]:

In this step the created geometrical model is discretized into nodes and elements. The process is called meshing. The Ansys program can automatically generate the nodes and elements, provided that you specify the element size. The element size controls the fineness of the mesh. The smaller the element size the finer the mesh. The following commands used for element size;
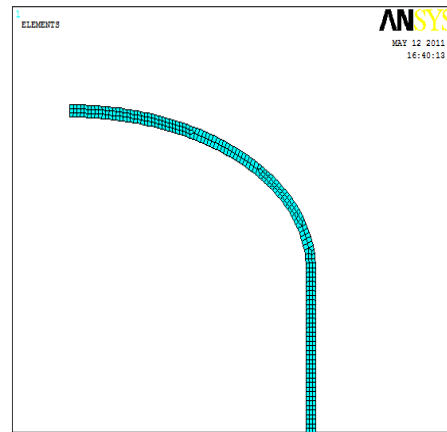

Fig. 3.2 Shows Meshing of Geometric Model of Pressure Vessel.

Preprocessor→ Meshing → Size Control→ Manual Size→ Global→ Size.

For meshing command is;

Preprocessor →Meshing →Mesh →Area

3.5 Solution or Processor: In this phase the following steps involved**.**

3.6 Loading and Boundary Conditions:The loading is given as:

Internal pressure and displacement constrains applied as per the deformation takes place in the pressure vessel. The commands to apply the Boundary conditions are;

Solution → Define loads → Apply → structural→ Displacement

In this problem we have applied the boundary conditions as shown in Fig.3.3 and those are at upper section line DX=0 and at bottom section line all DOF =0.

3.6.1 Loads: In order to specify the distributed loads on a line or surface of a model, the following commands need to issue.

Solution → define loads →Apply →structural →Pressures→on line or Areas

The pressure applied on the line is 5 MPa as shown in Fig 3.3.

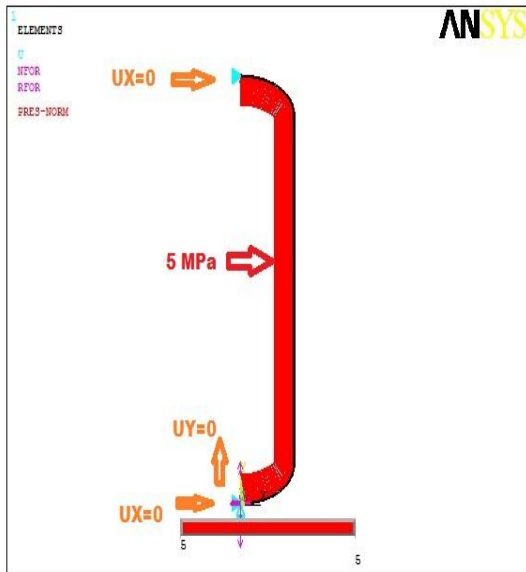Fig. 3.3 Internal pressure applied and displacement constrain.

3.7 Solution: In this step the problems get solved. To solve the above problem we have to issue the following commands;

Solution → Solve → Current LS

3.8 General Postprocessor:

In this step we get the results and its display in the deformed shape and contour displays. we have to issue the commands.

General Postprocessor→ Plot Results → Contour Plot → Nodal Solutions.

Fig. 3.4 Vonmises stress in N/mm^2 in the pressure vessel.

In the above deformation the maximum stress occurs at the junction of ellipsoidal head and shell and which is in the range 174 -192 N/mm^2
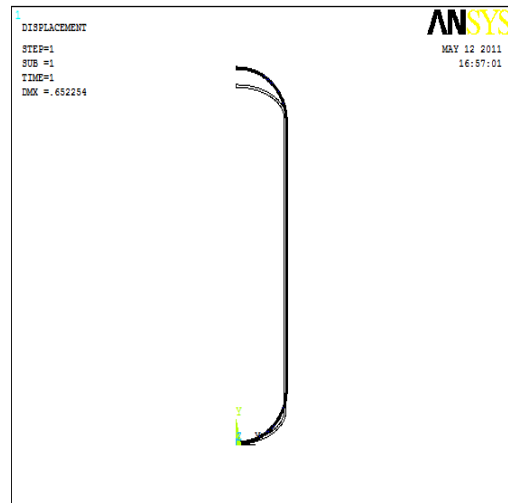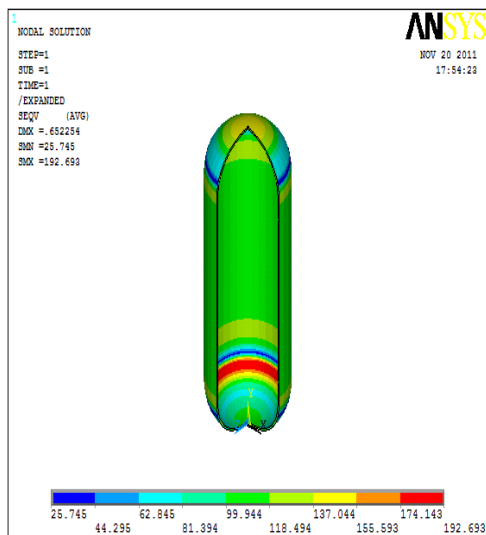

Fig. 3.5 Displacements (mm)

3.9 Convergence: - Element type: - Plane -42 [6]

Table 2.

Convergence Stress Table

Sr. No.	Element size	No. of Elements	No. of nodes	Max. Stress	Min. Stress	Stresses at crown	Stress at shell
1	14	77	154	170.53	21.36	120.81	87.66
2	12	89	178	174.44	26.36	125.0	92.18
3	10	108	215	175.57	28	126.40	93.62
4	8	134	268	175.8	28	126.4	93.89
5	6	180	357	179.48	30.2	129.7	96.55
6	4	398	664	179.0	30.47	129.5	96.50
7	2	1558	2087	185.96	25.18	132.36	96.6
8	1	6285	7340	185.81	23.15	131.4	95.29
9	0.7	13912	15419	185.86	22.95	131.3	95.24
10	0.5	25216	27324	186.06	22.72	131.60	95.32

(Note: All stresses in N/mm^2)

convergence chart of Element : -PLANE-42
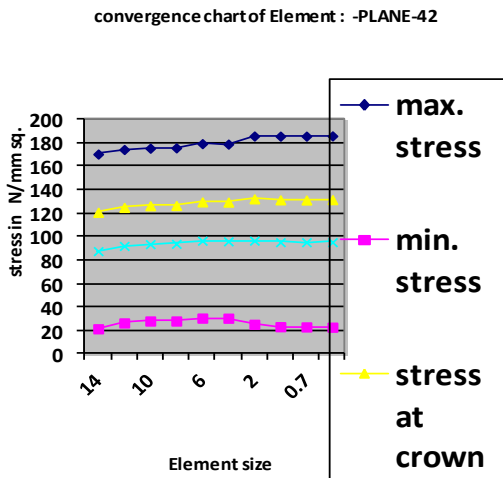

Fig.3.6 Convergence Chart of Element: Plane -42.

4. Strain Measurement Strain Gauge [1][12]:

4.1 Electrical Resistance Strain Gauges

Electrical Resistance strain Gauges are metallic resistors that can be pasted onto the surface of a part on which you want to measure strain. When the part deforms the strain gauge deforms along with it. This deformation is reflected as a proportional change in the resistance of the strain gauge.

Fig 4.1 Strain measurement with electrical strain gauges

4.2 Observations: Following observations are found with operating conditions.

Table shows stresses in the pressure vessel shell from mechanics of material laboratory [12]:-

From mechanics of material laboratory:-

For E- Young's Module – (for M.S. E= $2x10^5$ N/mm^2)

v- Poisons ratio = for M.S. v=0.3)

$$e_2 = \frac{\sigma_2 - v \, \sigma_1}{E}$$

$$308 \times 10^{-6} \times 2X10^5 = \sigma_2 - 0.3 \, \sigma_1 \qquad (iv)$$

$$e_1 = \frac{\sigma_1 - v \, \sigma_2}{E}$$

$$-81x10^{-6}x2x10^5 = \sigma_1 - 0.3 \, \sigma_2 \qquad (v)$$

From (iv) and (vi)

$Б_2 = 68.44$ N/mm^2 and $\sigma_1 = 22.8$ N/mm^2

4.3. Circumferential Stresses in Head:

$\sigma_c = E \times e_h = 2x10^5 \times 508 \times 10^{-6}$

$= 101.6$ N/mm^2

4.4 Comparisons of Stresses: The comparisons between the results of all the methods. Table 3.z

Table 3

The comparisons between the results of all the method

Sr. No.	Part of Pressure Vessel (Booster Tank)	Finite Element Method (N/mm^2)	Strain gauge Measurement Results (N/mm^2)
1	Shell Hoop Stress	99.3	68.44
2	Longitudinal Stress	-9.3	-16.2
3	Head Stress (Maximum)	138	101.16

5. CONCLUSION

I With the help of Finite element analysis we can study the actual stress distributions in the different components of pressure vessel and the actual behavior of pressure vessel.

II It is found that Different types of finite elements behavior is different for the same operating or load conditions.

III It is found that Element size affects the results of finite element analysis but   after particular size of the element it does not affects on the analysis results.

IV There is 25 % variation between the results by finite elements and experimental method.

V From the table no 2.4, it is observed with the help of electrical strain gauge ;   the actual values of stresses at different locations at pressure vessel are having some variations with reference to theoretical and finite element analysis.

VI With the comparison of the results from all the methods, it is observed that the results of the stress analysis by classical methods   are more than the actual stresses measured by strain gauges and less than the finite element analysis.

REFERENCES

[1] John F. Harvey, edition -1987, reprint -2001 "Theory and Design of Pressure Vessel",   Van No strand Reinhold Company.

[2] M.V.Joshi, Edition 1981, "Process Equipment Design", Macmillan India Limited, New Dehli.

[3] ASME codes -2003 -Section VIII, Division –I, Pressure Vessel codes, .

[4] Bronwell L.E and Young E.H ;-2010," Process Equipments Design ", John Wiley & Sons. Inc, New York, ,Printed in Delhi .

[5] B.C. Bhattacharya, Edition –I-1985, "Chemical Equipment design"; CBS Publisher New Delhi.

[6] Saeed Moaveni, Third edition -2008 "Finite Element Analysis , Theory and Application with ANSYS",Dorling Kindersley India pvt. Ltd.

[7]   Krishnamurty   C.S,   "FiniteElement Analysis,Theroy and Programming ", Tata McGraw-Hill Publication Co.Ltd .New Delhi.

[8] Reddy J.N, Edition second -1993,"Finite Element Method ", McGrw-Hill International .

[9] "ANSYS" Theory Manual and Expanded ANSYS Workbook,

[10] Mikel P. Grover and E.W.Zimmer-printed-1999 "CAD/

CAM" ;Prentice hall India.

[11] P.N. Rao; "CAD/CAM" ;   TATA MCgraw Hill .

[12] Strain measurement of Thin walled pressure vessel by   Mechanics of materials Laboratory