Finite Element Analysis & Thickness Optimization of Vacuum Chamber for Electron Microscopy Applications

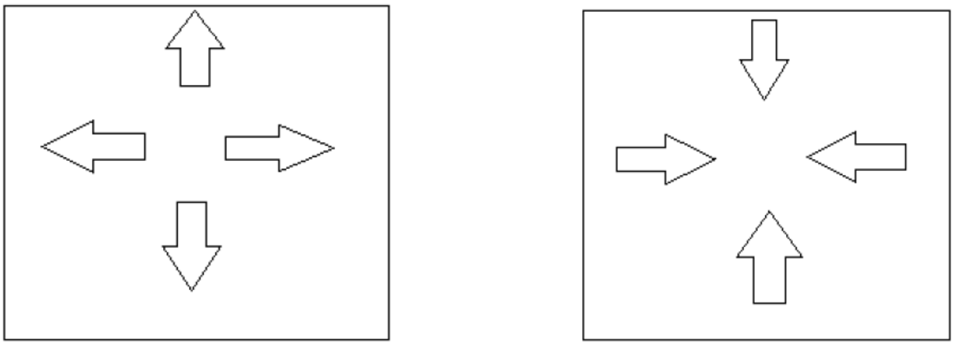
Mr. Abdul Sayeed AW Shaikh[1], Prof. P.T.Nitnaware[2]
[1,2] D. Y. Patil college of Engineering Akurdi, Pune,India

ABSTRACT: *This paper shows the Finite Element analysis of vacuum chamber. The analysis is done for electron microscopy applications, for scanning electron microscope it require vacuum atmosphere for viewing of the specimen. The specimen is to be viewed in vacuum. The vacuum level required for that is in the range of 93324 Pa, which is less than atmospheric pressure which lead to the compressive forces acting inside the chamber The vacuum chamber is modeled in Catia and Simulation is done in Ansys. Also theoretical calculation is done for safety of vacuum chamber against buckling failure. Also, shell analysis is done for considering the thickness of the vacuum chamber which is done by using Hypermesh.*

Keywords: *FEA, Electron Microscope, Buckling Propagation, ANSYS, Vacuum Chamber*

I. INTRODUCTION

An Electron Microscope uses particle beam of electrons to illuminate the specimen and produce a magnified image. Scanning electron microscope (SEM) and Transmission electron microscope (TEM) are types of its applications. Both these applications requires an vacuum atmosphere to operate. Also the level of vacuum requirement is same for both the applications. The electron gun without vacuum will experience constant interference from air particles in the atmosphere. The distraction would lead to block the path of electron beam and also they would be knocked out of the air and onto the specimen which ultimately distort the surface of the specimen. The vacuum level required for that is in the range of 91991 to 95990 Pa, which is less than that of atmospheric pressure which lead to compressive forces on chamber which causes buckling.



Typical vessel Vacuum vessel

Fig 1.1: Direction of Internal Pressure

II. FINITE ELEMENT METHOD

The finite element method (FEM), sometimes referred to as finite element analysis (FEA), is a computational technique used to obtain approximate solutions of boundary value problems in engineering. Simply stated, a boundary value problem is a mathematical problem in which one or more dependent variables must satisfy a differential equation everywhere within a known domain of independent variables and satisfy specific conditions on the boundary of the domain. Boundary value problems are also sometimes called field problems. The field is the domain of interest and most often represents a physical structure. The field variables are the dependent variables of interest governed by the differential equation. The boundary conditions are the specified values of the field variables (or related variables such as derivatives) on the boundaries of the field. Depending on the type of physical problem being analyzed, the field variables may include physical displacement, temperature, heat flux, and fluid velocity to name only a few. A finite element analysis has been done to the model for the expected stresses for 93324 Pa.

1.1 General Dimension of vacuum chamber

Table No.2.1: Dimensions of vacuum chamber

Description	Value (m)
Vessel Inner Diameter	1.070
Vessel Thickness	0.007
Length of Cylindrical Portion	1.330
Cover inner diameter	0.950
Cover Thickness	0.007
Cover Height	0.250
Base Plate Thickness	0.023
Vessel Flange Thickness	0.035
Cover Flange Thickness	0.035

Table No.2.2: Dimensions of Flanges

Valves	Value (m)	Thickness (m)	Flange Thickness (m)
1	0.205	0.0045	0.023
2	0.154	0.0045	0.023
3	0.120	0.0034	0.023

2.2 Critical Buckling Pressure

Buckling is a failure mode characterized by a sudden failure of a structural member subjected to high compressive stress. Cylindrical vacuum vessels subject to external pressure are subject to compressive hoop stresses. Consider a length L of the vessel,

$$\text{The total force acting} = \text{Intensity of pressure} \times \text{area}$$
$$= p \times D \times L$$

$$\text{The total resisting force acting on the vessel walls}$$
$$= \sigma_h \times 2t \times L$$

From above two equation's,

$$\sigma_h \times 2t \times L = p \times D \times L$$
$$\sigma_h = \frac{p \times D}{2t}$$

The compressive hoop force,

$$W_h = \sigma_h \times L \times t$$

$$W_h = \frac{\nabla p_{buckle} \times D \times L \times t}{2t} \quad \ldots\ldots\ldots\ldots\ldots (1)$$

Buckling will occur when compressive hoop force will equal to buckle force, W

$$W = \frac{4\pi^2 EI}{(\pi D)^2}$$

$$\text{Since, } I = \frac{L \times t^3}{12}$$

$$W = \frac{4\pi^2 E \times (\frac{Lt^3}{12})}{(\pi D)^2}$$

$$W = \frac{ELt^3}{3D^2} \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots (2)$$

Equating eq^n (1) and (2)

$$\frac{\nabla p_{buckle} \times D \times L}{2} = \frac{ELt^3}{3D^2}$$

$$\nabla p_{buckle} = \frac{2E}{3}(\frac{t}{D})^3$$

$$\nabla p_{buckle} = \frac{2 \times 2 \times 10^{11}}{3} \times (\frac{0.007}{1.056})^3$$

$$\nabla p_{buckle} = 38836.59 \text{ Pa}$$

We have applied approximately p = 7999.2 Pa for 700 torr (10 to 15) % of max buckling pressure as per requirement, the impact will be very small. So it is safe from buckling failure.

Thickness of the vacuum chamber,

$$t = \frac{pDo}{(2 \times f \times J + p)}$$

The design pressure is taken as 1.1 times of the pressure,

$$t = \frac{0.1026 \times 957}{(2 \times 205 \times 0.85 + 0.1026)}$$

$$t = 0.3 \times 10^{-3} \text{ m}$$

In reality the vacuum chamber consist of nozzles and flanges and the above thickness is considered as the vessel is plane without any nozzles and flanges. Due to the nozzles and flanges which create abrupt changes in the cross-section gives rise to stress concentration and reduces the strength of the material. To overcome this problem and to be on the safer side we have taken the thickness of the vacuum chamber as 7 mm.

III. MODELLING

The vacuum chamber comprises of cylindrical vessel with hemi spherical elliptical cover on one side and base plate on other side. The vessel and end cover is modeled in Catia part body workbench separately and fixed together in Catia assembly workbench. The assembly model is then save in stp format so it can be imported to Ansys workbench. The model is split by YZ plane as it is symmetric about it, and giving translation constrain in X-axis. A face to face contact is defined between cover flange and vessel flange

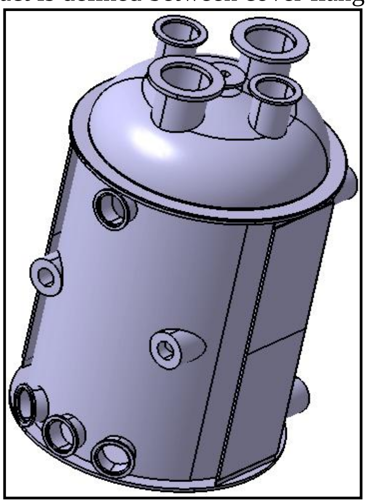


Fig. 3.1 Solid Model

IV. MESHING

Meshing has been done by using the method of Tetrahedron . In Tetrahedron method the component is been divided into small triangle on its surface which gives no of nodes and elements of that component and. The meshing has been done by changing the mesh size of the various component of the vacuum chamber. Due to change in the density of the meshing, it results in the variation of the no of nodes and elements of the meshed parts

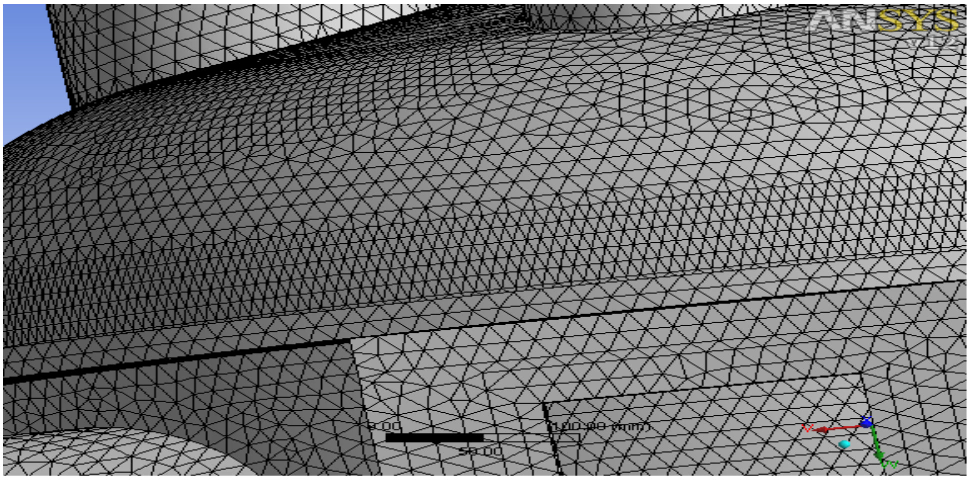


Fig 4.1 Tetradreon Meshing

V. BOUNDARY CONDITIONS

The simulation is done for internal pressure 700 torr which is less than of atmospheric pressure 760 torr that leads to compressive pressure on chamber. One atmospheric pressure equals to 760 torr (mm of Hg).
(700 -760) torr = -60 torr = -7999.342 Pa.Thus 7999.342 pa pressures is applied on vacuum chamber walls. Due to negative sign, the direction of pressure is inwards. The faces of valve flanges and side base plate are fixed .

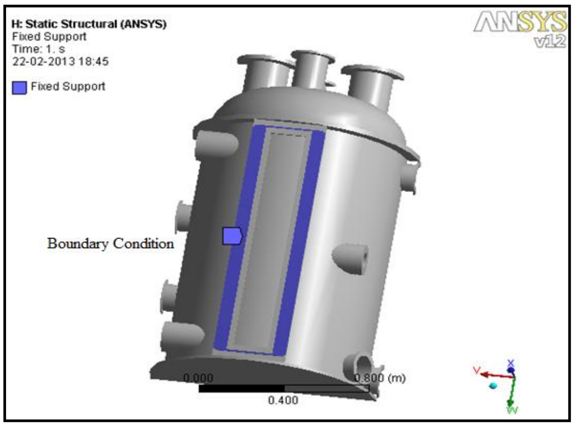


Fig. 5.1 Fix Support

Fig. 5.2 Pressure Applied

5.1 Material Properties

Material: Structural Steel
Young's Mod. of Elasticity, E = 2e11 pa
Poisson's ratio, μ = 0.3
Density, ρ = 7850 kg / m3
Compressive yield strength = 205e6 pa

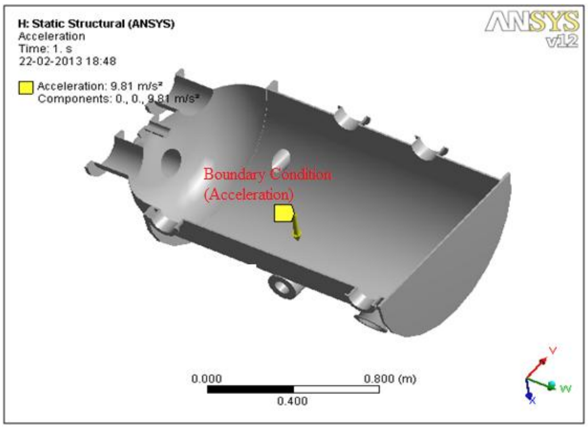


Fig. 5.3 Acceleration

VI. FINITE ELEMENT ANALYSIS

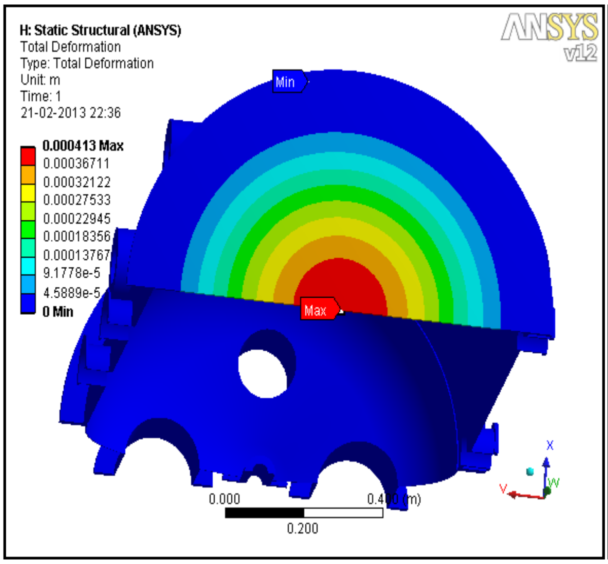


Fig 6.1 Max Deformation for the full body

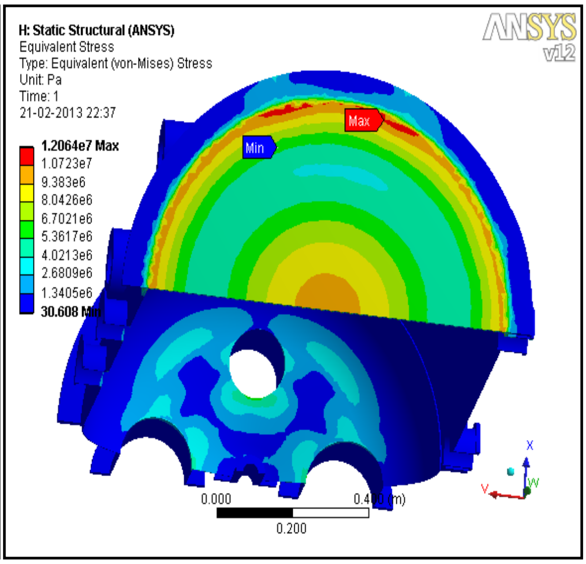


Fig 6.2 Max Von Mises Stress for the full body

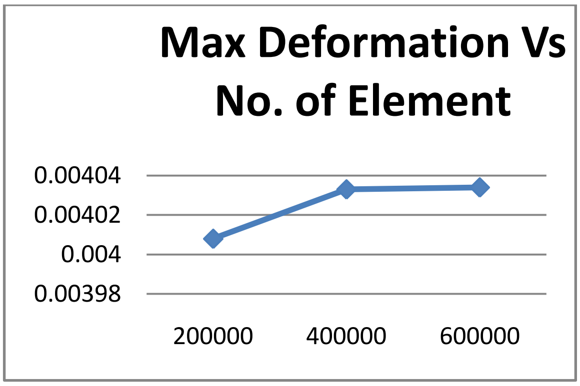


Fig. 6.3

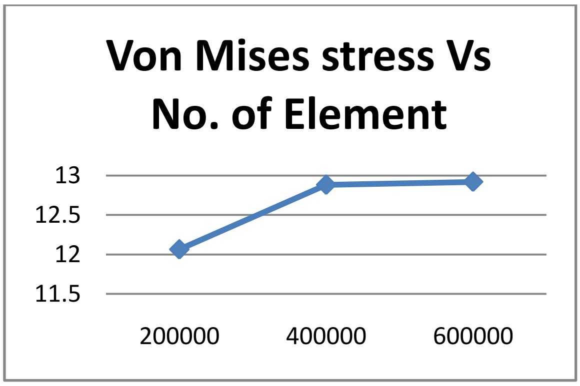


Fig 6.4

VII. RESULTS

Deformation and Von Mises Stress at different regions has been calculated using post processor. VON Mises Criterion also known as the maximum distortion energy criterion, octahedral, is often used to estimate the yield of ductile materials. The Von Mises criterion states that failure occurs when the energy of distortion reaches the same energy for yield/failure in uniaxial compression/tension. Max Von Mises stress calculated from the analysis is 1.4e7 Pa on the vessel between valve and base plate as shown in Fig 6.8 which is very less than compressive yield strength 205e6 Pa. So it is safe from this criterion.

VIII. SHELL ANALYSIS

To verify the results of solid structural analysis, shell structural analysis for internal pressure 700 torr is done. The difference is that, here middle surface of the solid body is modeled and thickness is given in Hypermesh. It includes similar steps as that of solid structural analysis. Here middle surface of body is to be modeled which go inside and outside the half of thickness and turns into solid body in Ansys workbench. There are six degree of freedom, three for translation and three for rotation. Vacuum chamber middle surface is modeled in Catia surfacing workbench by making all components individually and assembled in Catia assembly workbench. And save in igs format so it can be imported in Ansys workbench. In Ansys workbench, for every part thickness is defined. Symmetry is applied normal to X- axis. i.e. about YZ plane, the vacuum chamber is constrained to translate and rotate in X direction . Each part of vacuum chamber is connected to each other defining line contact b/w valve to flange and valve to vessel and face to face contact between cover flange to vessel flange and side base plate to vessel.

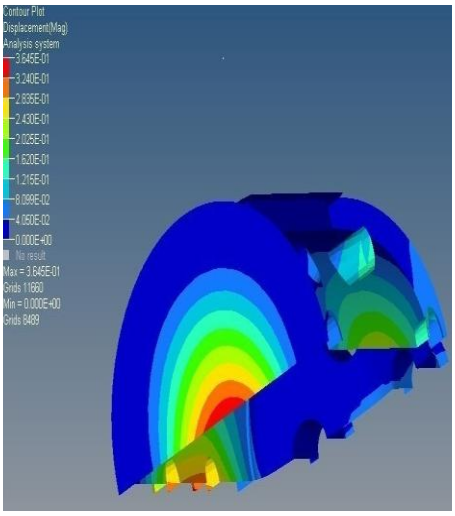


Fig.8.1 Surface Model 8.2 Line contact between valve & flange 8.3 Deformation

Max Deformation = 3.16 E-4 m. The max deformation in both analysis is on the base plate also their difference 1.2 E-5 m is very small. So the result given by solid structural analysis is correct.

IX. CONCLUSION

- The Vacuum chamber is safe from buckling failure as applied p is very small in comparison to max theoretical buckling pressure.
- Compressive yield strength of structural steel is greater than Von Mises stress is calculated by solid structural analysis so it is safe from this criterion.
- A linear graph comes out for max deformation and max Von Mises stress with internal pressure so for different value of internal pressure theoretically they can be calculated

REFERENCES

[1] WANG Wenlong, CAI Guobiao, ZHOU Jianping " Large-Scale Vacuum Vessel Design and Finite Element Analysis" Chinese Journal of Aeronautics 25(2012) 189-197.
[2] M. J. Kraan, Buskop, M. Doets, C. Snippe on "Structral analysis of the vacuum vessel for Lhcb Vertax locator."
[3] J.N.Reddy , An introduction to the Finite Element Method 3rd edn. McGraw-Hill, New Delhi, 2009.
[4] K. Ioki, P. Barabaschi, V. Barabash, S. Chiocchio,W. Daenner,F. Elio, et al., Design improvements and R&D achievements for vacuum vessel and in-vessel components towards ITER construction, Nucl. Fusion 43, 268–273.