International Journal of Modern Engineering Research (IJMER)
www.ijmer.com Vol.2, Issue.3, May-June 2012 pp-991-995 ISSN: 2249-6645

Design Analysis of Pressure Vessels at high stress zones using Pro/E v4.0

*Yogesh Borse ** Avadesh K. Sharma
*(M.E. Scholar, Mechanical Engineering Department, MITS, Gwalior, MP, India)
** (Asst. Prof., Mechanical Engineering Department, MITS, Gwalior, MP, India)

ABSTRACT
This paper deals with the Finite element modelling and Analysis of Pressure vessels with different end connections (i.e. Hemispherical, Ellipsoidal & Toro spherical). Considering the fact that required thickness of hemispherical head for internal pressure loading is only half of that necessary for the cylindrical shell, authors have tried to develop a finite element model taking due consideration on welding involved at the end connections of cylinder to shell end in modelling using shell elements to model cylinder. The larger the shell in diameter the more the economic benefits it achieves. In this paper authors, describes its basic structure, stress characteristics and the engineering finite element modelling for analysing, testing and validation of pressure vessels under high stress zones. The equivalent von-misses stresses for different end connections are plotted.

Keywords **- High pressure vessels, reinforcing area (consideration to welding), hemispherical end, Ellipsoidal end & Toro spherical end connection**

1. INTRODUCTION

Pressure vessels may theoretically be almost of any shape, but shapes made of sections of spheres, cylinders, and cones are usually employed. A common design is a cylinder with hemispherical end caps called heads. More complicated shapes have historically been much harder to analyse for safe operation and are usually far more difficult to construct. The pressure vessel of hemispherical type end connection is shown in figure 1.

Figure 1

Many pressure vessels are made of steel. To manufacture a spherical pressure vessel, forged parts would have to be welded together. Some mechanical properties of steel are increased by forging, but welding can sometimes reduce these desirable properties. In case of welding, in order to make the pressure vessel meet international safety standards, carefully selected steel with a high impact resistance & corrosion resistant material should also be used.

2. LITERATURE REVIEW

High-pressure vessels, such as ammonia converters, urea reactors and supercritical fluid extractors, etc., are widely used in chemical, oil refining, energy industries, and so on. Such vessels are key equipments in various processes industries and have potential hazards. Much attention has been paid to using them safely and to lowering their costs, with great progress being made in the last century. For example, Analysis of Pressure Vessel junction by the Finite element Method written by Mahadeva Sivaramakrishna Iyer not only tells the use of method to solve such high tension zone problems but also gives a way to predict results for stresses and optimize the design [1], Finite element analysis of Pressure vessel by David Heckman also tells the use of computer programs instead of hand calculations for analyzing the high stress area's and different end connections [2]. The different types of stresses and modeling of pressure vessel joints are also depicted in ASME code in section "Design by analysis" [3].

Fig. 2

The use of hemispherical end in pressure vessels is the most economical and common use which can be seen in India and other developing countries. Although with the recent trends in Mechanical engineering with the use of Finite element software's the sheet thicknesses are validated for different end connections and for cylinder shell itself.

As per the conventional theory of mechanics of materials stated by S Timoshenko, the required thickness of hemispherical end is one-half the thickness of the shell to result in equivalent stresses in the cylinder.

3. RESEARCH METHODOLOGY

A pressure vessel with hemispherical end connection, up with internal pressure of 20 atm is analyzed for the study of Von-Misses Stresses at the connection using the finite element software Pro-Mechanica Integrated mode. Pro-Mechanica is a FE module integrated with CAD software Pro/E, which operates on P-element type method. A semi-mechanistic FE shell model is prepared with mean diameter of 5000 mm open at both ends and was tested in pro-mechanica for internal pressure to validate the working of thin pressure vessel theorem

Stress $(\sigma) = Pd/2t$ --- (1)

where

P: - Internal pressure

d: - Mean diameter

t: - thickness of vessel,

The pre-assumed thickness of cylindrical vessel was taken to be 50 mm for the validation problem.

Finite element software packages are more or less dependent upon the applied loads and Boundary conditions for the problem.. The theoretical result of the problem is 100 MPa while what was attained from the software lies in the range of 92-96 MPa. The results are discussed in section "**Result & Conclusion**".

The results given by Finite element method was nearly approaching the values of theoretical results. A designer is also interested in the pattern of the stress distribution for different other decisions***** to be taken while designing other related components of the vessel like baffles and stiffeners.

The same cylindrical surface model modeled in Pro/E Wildfire 4.0 with mean diameter was updated for the First case analyzed for a vessel with hemispherical end. The connection was analyzed for von-mises stresses. The polynomial was drawn with the resulted data as presented for the stress variation at the junction with different thickness of end connection.

The modeling of pressure vessel was done taking shell element for the cylinder which was also validated from the standard thin pressure vessel formula. The Hemispherical End connection was also modeled with shell element.

The difficulty of modeling the weld connection between the hemispherical end and cylindrical shell was solved by providing the alternate shell thickness greater than that of the base metal thickness, at the junction to provide it the strength of the weld. The case of similar thickness of the connecting parts is not presented here in the paper, where V or J joint type weld connection is considered to be safe taking no active participation in results of Finite element analysis. Considering the basis from J.E.Shighley Edition VII [1] that the weld joint is made stronger than the base metal thus the equations of statics are applied to base metal area of cross-section rather than the cross-section when the cantilever beam is analyzed for maximum bending.

The Hemispherical end was modeled in sub-sets of surfaces as shown in the figure given below, where different thickness of each sub-set was provided for consideration of weld strength. Here in the paper author have approached the problem with considering 3 sub-sets for the actual problem where the thickness of cylinder is 50 mm, first small surface shell element on hemispherical end is considered with thickness of 35 mm, the next element considered is of 30 mm joining the exact thickness of hemispherical end of

25 mm thickness. The solution to the approach is compared with solid element analysis considering the solid volume for the weld and analyzing the same. The comparisons of both the models are resulted. The paper was extended via analyzing the hemispherical end with different thickness to drive the polynomial for stress variation in the metal.

With the growing demand of industry to save cost, many researchers have put their efforts to optimize the shape of end connections resulting in optimization of weight. Many papers are presented on different shapes of end connections where one among the most common is ellipsoidal shape of the end-connection. Ellipsoidal end connection is shallower to that of hemispherical end connection and therefore is not able to resist the same pressure as that of hemispherical end. This is also called a 2:1 elliptical head. The shape of this head is more economical, because the height of the head is just a quarter of the diameter. Its radius varies between the major and minor axis.

The same surface model is prepared with same parameter of mean diameter and sub-sets of surfaces joining cylinder to elliptical end.

4. Experimental Work

A detailed three-dimensional (3-D) finite element (FE) computer model was constructed for Static qualification analysis of the Shell and End-Connection. The static qualification for stresses and deflection was performed using licensed FE analysis software Pro/E Wild-Fire.
The analysis was performed to determine structural integrity and performance characteristics of the components of Pressure Vessel under static loading conditions. The objective of the work performed is to quantify the part performance characteristics under the influence of static loads.

The static analysis results yield maximum stresses and maximum deflections in the structural component of the Pressure vessel. Results indicate that the components of pressure vessel will maintain its structural integrity during the specified load cases. Possible Optimization of the vessel is tried via changing the sheet thickness of the vessel.

The classical theory of thin shells of revolution is attempted here. The results from FE analysis are then be compared to the known results available in literature and also with the stresses predicted by the ASME Code. This section documents in details the technical approach, FE computer modelling, maximum static deflections at critical locations and maximum static stresses at different locations in different components of the vessel. The general program Pro-mechanica is used for the linear static analysis of a general Vessel.

The First Model Analyzed is with internal pressure loading of 20 atm i.e. 2 MPa is made with hemispherical end connection. The FE Meshed model along with loads and boundary condition and meshing information for the same is shown in the Figure 3 given below

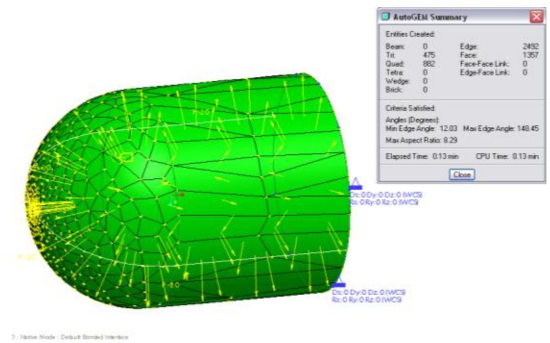


Figure 3 Sub-Structured Model for Hemispherical End Vessel

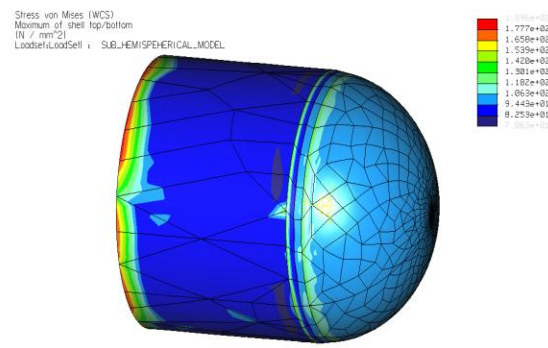


Figure 4 Von Mises Stresses with Hemispherical End Connection

Neglecting the stresses Near Constraint, we can see the value of stresses lying between the range of 106 MPa at Hemispherical end surface, while the connecting end gives the result in the range of 130 MPa.

The same analysis was performed for a Sub-structured model of the vessel with Elliptical end connection with same loads of 2 MPa and same boundary conditions at the end.

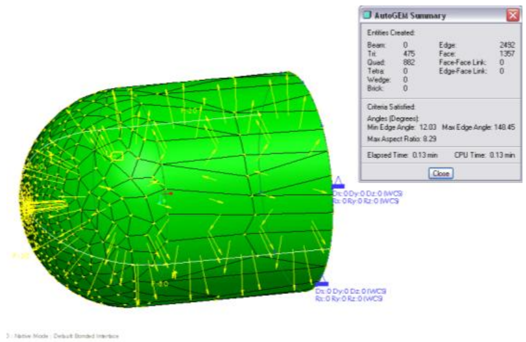


Fig.5: Meshed Model for Elliptical End

Figure 6 below shows the value of Membrane stresses for the sub-structured model with elliptical end connection which of the range of 166-195 MPa in orange to red zone.

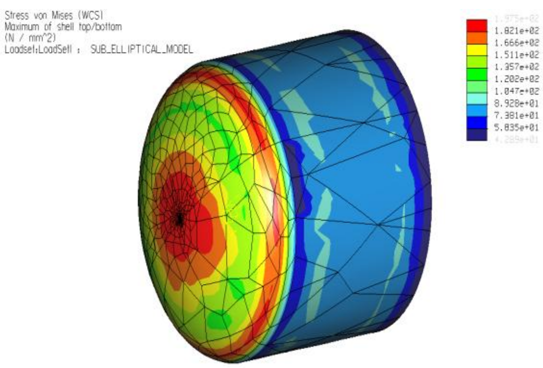


Figure 6 Membrane Stresses for Sub-Structured Model of Elliptical End Vessel

The Meshed Model for sub-model of the vessel with Toro spherical end is shown in the Figure 7, along with the details of elements used in the mesh.

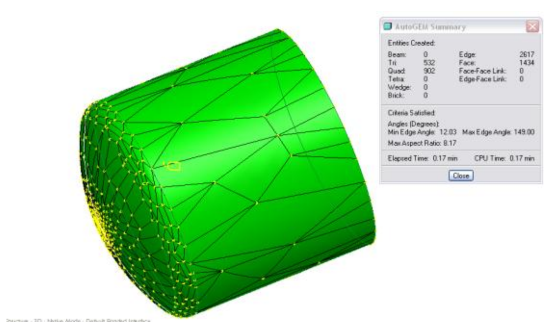


Figure 7 Meshed Model of Toro spherical End

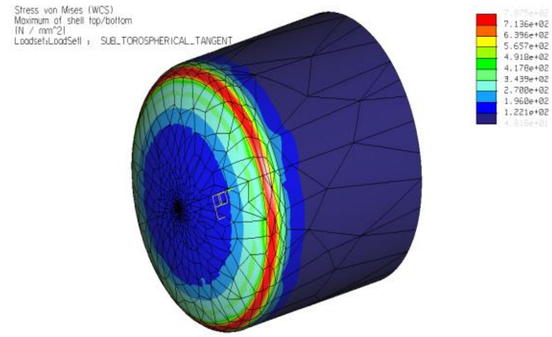


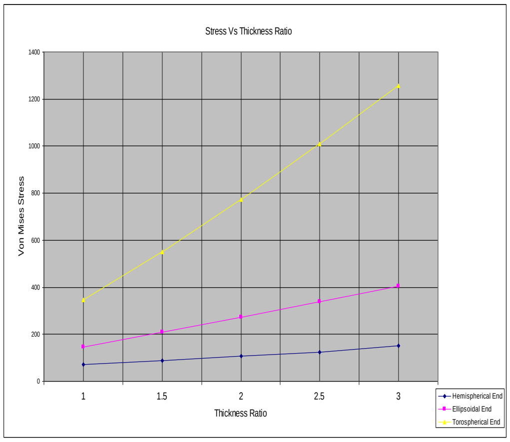
Figure 8 Von-Mises Stresses for Torospherical End

The values of stresses are found to be in range of 750-780 MPa

5. RESULT & DISCUSSION

The results with the used loads and boundary conditions which remain same for all the analysis with different end connections shows that the end connection with hemispherical shape results in the least stresses when compared to other models not only at weld zone but also at the far end of the end-connection. This can also be understood as the maximum pressure taking capacity for the hemispherical type end connection with least thickness (t/2 as per theory). While due to restrictions of the cost of material, and optimization in demand via industry, the use of ellipsoidal shape can be made with the increased thickness of the end connection for same level of stresses.

The results also shows that without varying the thickness of elliptical end connection i.e. with same thickness the results of stresses can be considered to lie in a safe zone and can be used for the safe working of the vessel. Third type of model analyzed was torospherical end type model where it was found the capability of undertaking pressure force is least and thus the stresses exceeds the limit of ASME standard [2] for the material under the same loads and boundary conditions, still the same can be used with a little increment in thickness as shown in the compared study chart below. The graph shown below in Figure 9 shows the results of polynomial equation developed with making number of analysis runs with varying thickness ratio which is defined via thickness of shell to shell end connection



Figurse 9 Stress Vs Thickness Ratio Graph
The chart in Fig 5.1 shows the Stress Vs Thickness Ratio curve for all three types of end connections analyzed.

6. CONCLUSION

A FE modeling study to analyze stresses and deformation is conducted as per the recommendation of ASME standard [2] for the comparison of three most recommended types of end connections.

With reference to literature studied [2-5]and[9], It was realized that the shell to shell end connection results best with the use of shell elements, while for nozzle to shell connections the solid to shell element connection should be used to result good quality results.

Different end connection sub-structured models were analyzed with same internal pressure of 2 MPa and the results for different end thickness were plotted with thickness ratio of 2.

Analysis results do not suggests the use of Torospherical end connection with the same thickness while the elliptical end connection can be used with FE analysis performed for the structural stability of the vessel as defined by ASME standard [2].

REFERENCES

[1] Shigley Josheph.E Mechanical Engineering Design, 2003, Sixth Edition, Mc GrawHill, Boston, Page 133, Equation 3.52-3.53, Fig 3.28.

[2] he American Society of Mechanical Engineers, “2007 ASME Boiler & Pressure Vessel Code” Section VIII Divisions 2, Alternative Rules for Construction of Pressure Vessels, Part-5, Design By Analysis Requirements.

[3] atts G.W & Lang H.A, “Stresses in Pressure Vessel with a Conical Head”, ASME Transactions, 207-218, 1952.

[4] yer Mahadevan Shrinivasan, “Analysis of Pressure Vessels Junction by the Finite Element Method”, Civil Engineering Texas Tech University, PhD, 1972, 1-147.

[5] amesh C. K., Kant Tarun & Jadhav V. B, “Elastic Analysis of Cylindrical Pressure Vessels with Various End Closures”, Pressure Vessels & Piping Vol-2, 1974, 143-154.

[6] wang D. Y. & Stallings J. M, “Finite Element Analysis of Bolted Flange Connections”, Computers & Structures Vol-51, 1992, 521-533.

[7] eib Eckart & Rudolph Jurgen, “Finite Element Analysis Concerning the Fatigue Strength of Nozzle-To-Spherical Shell Intersections”, Pressure Vessels & Piping Vol 64, 1995, 101-109.

[8] Meyer J.E., “Pressure Vessel Stress Analysis”, Structural Design Notes Topic-C, C1-C10, 1996.

[9] eckman David, “Finite Element Analysis of Pressure Vessels”, Monterey Bay Aquarium Research Institute, University of California, 1998, 1-7.

[10] Huang Fuang Yuan & Shi Gao-Liang “Finite Element Analysis of Pressure Vessel using Beam on Elastic Foundation Analysis”, Finite Elements in Analysis and Design Vol-28, 1998, 293-302.

[11] Dekker C. J. & Brink H. J, “Nozzle on Spheres with Outward Weld Area under Internal Pressure Analysed by FEM and Thin Shell Theory”, Pressure Vessels & Piping Vol-77, 2000, 399-415.

[12] Abbasi N. El, Meguid S. A. & Czekanski A, “Three-Dimensional Finite Element Analysis of Saddle Supported Pressure Vessels”, Mechanical Sciences Vol-43, 2001, 1229-1242.