

## Modelling of Pressure Vessels with different End Connections using Pro Mechanica

**\*Yogesh Borse \*\* Avadesh Sharma**

\*(M.E. Scholar, Mechanical Engineering Department, MITS, Gwalior, MP, India)

\*\* (Asst. Prof. Mechanical Engineering Department Madhav Institute of Technology & Management, Gwalior, MP, India)

### ABSTRACT

This paper deals with the Finite element modelling of Pressure vessels. 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. Theoretical investigation and engineering applications shows some junctions are rational in structure, convenient in fabrication and less in cost. Thus some end connections are tested under FEA for the cause of resulting weight reduction of 30-35 %. The larger the shell in diameter the more the economic benefits it achieves. In this paper authors, describes its basic structure and the engineering finite element modelling for analysing, testing and validation of pressure vessels under high stress zones.

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

### I. INTRODUCTION

A pressure vessel is a leak-proof container designed to hold gases or liquids at a pressure substantially different from the ambient pressure. 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.



Fig. No. 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.

### II. 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 equipment's 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: 2.1

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.

### III. 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

$$\text{Stress } (\sigma) = Pd/2t \quad \text{--- (1)}$$

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 convergence level for the problem can be defined using the package where in P-element type software the order of polynomial is increased until the solution is converged to a preset level, here the convergence of solution is set to 5%, which was achieved in 3<sup>rd</sup> pass of the software (i.e. 4<sup>th</sup> order equation). 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 along with the images in section "Results & Conclusions".

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 whose results are depicted in the table format with different thickness of the Hemispherical End connection. 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 [3, 4] for the cylinder which was also validated from the standard thin pressure vessel formula. The Hemispherical End connection was also modeled with shell element [3, 4].

[3]- ASME "Design by analysis"

[4]- Elastic analysis of cylindrical pressure vessel with various end closures by C.K Ramesh IITB.

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 Machine design (stated by J.E. Shigley Edition VII chapter 6<sup>th</sup> "Permanent Joints") 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 2 or 3 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 analysing the same. The comparisons of both the models are resulted. The Paper was extended via analysing 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 2 sub-sets of surfaces joining cylinder to elliptical end.

**IV. EXPERIMENTAL WORK**

A detailed three-dimensional (3-D) finite element (FE) computer model was constructed for Static qualification analysis of the Shell, Saddle 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 Analysed 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 Fig.4.15 given below

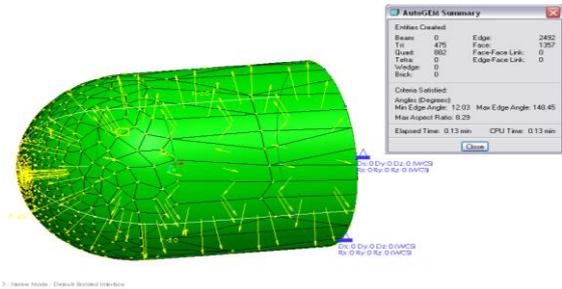


Fig.4.15 Sub-Structured Model for Hemispherical End Vessel

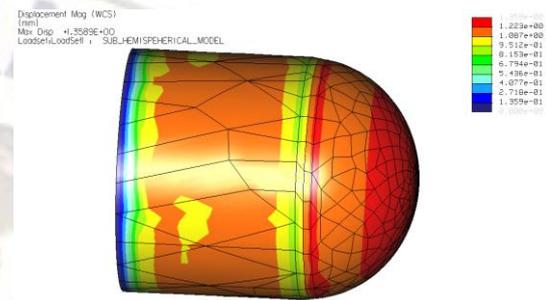


Fig.4.18: Total Displacement for Hemispherical End

The Displacement value from the results can be seen of 1.359 mm which is in the axial direction i.e. in the direction of the axis of vessel.

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.

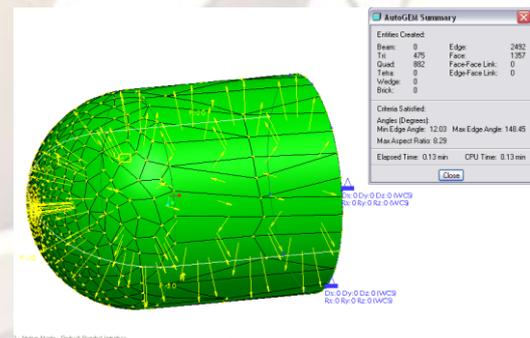


Fig.4.19: Meshed Model for Elliptical End

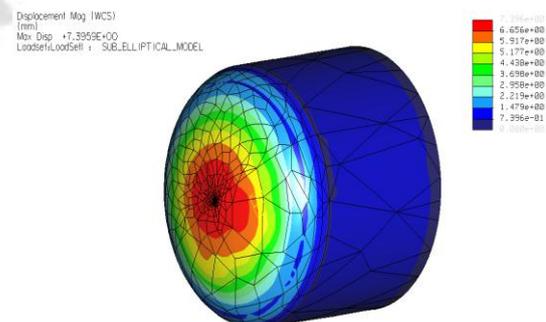


Fig.4.22: Total Displacement for Sub-Structured Elliptical End Type Model

The total displacement of 7.39 mm which is found to be in axial direction of the vessel is shown in Fig 4.22 by the red zone of figure.

The Meshed Model for sub-model of the vessel with Toro spherical end is shown in the Fig.4.24, along with the details of elements used in the mesh.

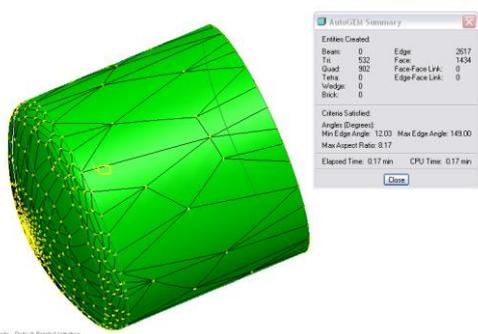


Fig.4.24: Meshed Model of Toro spherical End

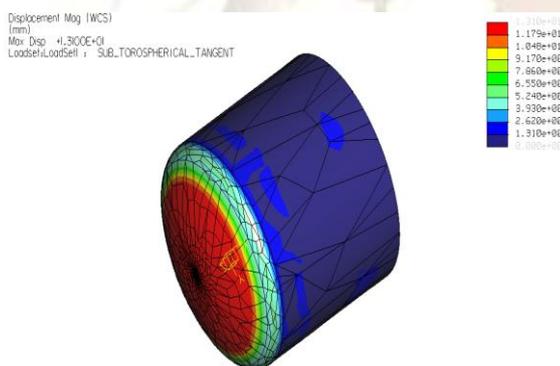


Fig.4.27: Displacement in Toro spherical End

The total displacement of 13.1 mm which is found to be in axial direction of the vessel as shown in red zone.

## V.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 analysed was Toro spherical 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 Fig 5.1 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. Comparative chart regarding the displacement result is shown below.

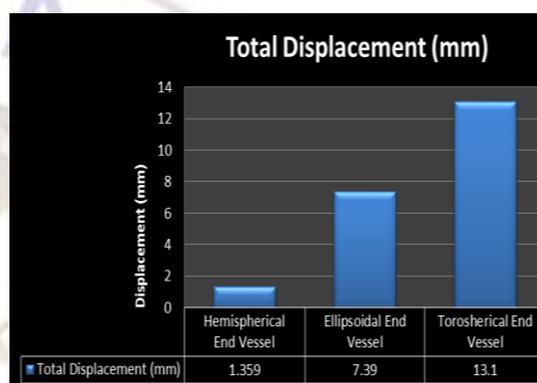


Fig. 5.1: Total Displacement

## VI.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][3][4][5][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.

Three different end connection sub-structured models were analysed 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 Toro spherical 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 Joseph.E Mechanical Engineering Design, 2003, Sixth Edition, Mc GrawHill, Boston, Page 133, Equation 3.52-3.53, Fig 3.28.
- [2] The 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] Watts G.W & Lang H.A, “Stresses in Pressure Vessel with a Conical Head”, ASME Transactions, 207-218, 1952.
- [4] Iyer Mahadevan Shrinivasan, “Analysis of Pressure Vessels Junction by the Finite Element Method”, Civil Engineering Texas Tech University, PhD, 1972, 1-147.
- [5] Ramesh 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] Hwang D. Y. & Stallings J. M, “Finite Element Analysis of Bolted Flange Connections”, Computers & Structures Vol-51, 1992, 521-533.
- [7] Weib 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] Heckman 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.