IOSR Journal of Engineering (IOSRJEN) **www.iosrjen.org**
ISSN (e): 2250-3021, ISSN (p): 2278-8719
PP 18-22

Finite Element Analysis of a Pressure Vessel and Effect of Reinforcement Pad

R. R.Jain[1], G. S. Kanase[2], G. A. Babel[3], S. S. Dalvi[4], B.R.Pujari[5]
[1, 2, 3, 4,5]*(Department of Mechanical Engineering, M E S College of Engineering Pune ,S P Pune University India)*

Abstract : *Pressure vessels are used for storing inflammable fluids like petrol, diesel, propylene, LPG, ammonia, propane, butane, etc. In Pressure Vessels, Nozzles are required for inlet and outlet purposes. These nozzles create discontinuities on the geometry which results in stress concentration around these nozzles. The junction might fail because of these high stresses; therefore, anelaborate analysis is required. It is quite difficult to perform stress calculations using theoretical formulae of such complex geometries, hence finite element method is used to perform and simulate static structural analysis of the pressure vessel. Specifically, elastic stress analysis, limit load analysis, and elastic-plastic stress analysis are the methods used to check for acceptance criteria and protection against plastic collapse, and computational results for a given load case are compared. Reinforcement pad is a method used for reducing stress concentration around a nozzle-vessel junction. Comparative study of stresses with and without reinforcement pad is performed.*
Keywords: *Pressure Vessel, Finite Element Analysis, Reinforcement pad, Static structural analysis*

I. Introduction

Pressure vessels are broadly connected in numerous fields of mechanical engineering, for example, compound and oil machine-building, atomic and control designing, gas, oil and oil-refining ventures, aviation systems, and so on. As the name infers these are vital parts of handling gear. Nozzles or opening are important in the pressure vessels to fulfill certain necessities, for example, bay or outlet association, sewer vents, vents and so forth. Geometrical parameters of nozzle associations may differ significantly in even a solitary vessel. These nozzle cause geometric intermittence on the vessel shell. So a pressure fixation is made around the nozzle vessel intersection. This may come about is disappointment of the intersection. Subsequently a point by point examination of this intersection is required. [1, 2]

Because of various loads connected to these structures, a neighborhood high stress condition occurs on the crossing point of the nozzle and the vessel. Interior pressure is essential load condition utilized as a part of the structure examination for assurance of principle vessel-nozzle associations. However the impact of outer forces and moments connected to nozzle ought to be contemplated notwithstanding along with the internal pressure. Outside loading for the most part happens because of a channeling framework connected to the nozzles.PravinNarale et.al [1] have performed static structural analysis of a pressure vessel with nozzles along the periphery of the vessel. They concluded that discontinuities around the vessel lead to increased stress concentration around the nozzle-vessel junction.,J. Fang et.al [2] has done a comparative study of usefulness for pad reinforcement in cylindrical vessels under external load on nozzle. The main purpose of this paper is to perform a comparative study of strength behavior for vessel shell intersections with and without pad reinforcement under out-of-plane moment loading on nozzle.V. N. Skopinsky et.al [3] presented a paper on solid modelling and static structural stress analysis of a nozzle junction on an ellipsoidal dish end of a pressure vessel. The paper presents a detailed analytical procedure used and the features of the finite element model undertaken for finding the solution of the problem. Steven R. Massey et.al [4] proposed results of finite element analysis of a number of of nozzle shell dimensional configuration in an effort to determine whether the FE results may be used to establish the critical variables necessary to construct a standard allowable piping load basis.

II. Need of FEA

Finite Element Method (FEM) is a computer based numerical method of solving engineering problems by simulating the actual test conditions. FEA is widely accepted in almost all engineering disciplines. The technique is frequently utilized as another option to the test strategy set out in numerous models. The system depends on the preface that an inexact answer for any engineering issue can be obtained by subdividing the structure/segment into littler more reasonable (finite) components. The Finite Element Model (FEM) is analyzed with an inherently greater precision than would otherwise be possible using conventional hand analyses, since the actual shape, load and constraints, as well material property combinations can be specified with much greater accuracy than that used in classical hand calculations. [3]

III. Meshing

Meshing is discretization of a model into smaller elements. It is one the most integral step of any finite element analysis. Mesh size should be selected in such a way that a fine balance is maintained between computation time and accuracy of the results obtained.

Fig. 1 Meshed Geometry

3.1 Number of Elements.

Optimum mesh size should be chosen for any analysis. After a certain point when we go on increasing the number of elements, there occurs a condition where any further increase in number of elements leads to no significant change in the analysis results of the problem. At this Convergence is said to have occurred i.e. Optimum mesh size is obtained. Fig. 2 shows the Mesh convergence for this particular problem.

IV. Static Structural Analysis

Static structural analysis is one in which the load/field conditions does not fluctuate with time and the presumption here is that the load or field conditions are applied gradually. From a formal perspective, three conditions must be met in any pressure analysis, balance of forces (or stress), similarity of displacements and fulfillment of the condition of stress at continuum limits. Different sorts of loads like pressure, weight, inertia, thermal, or indicated loads can be connected. An imperative point to consider is that no less than one of the displacement must be known before the rest can be resolved. The referred to displacement are alluded to as limit conditions. [1]

4.1 Inputs for finite Element Analysis

<u>A. Design Data</u>

- Design Code: ASME Section 8 div. 2 Part 5
- Design Pressure: 6 MPa
- Design Temperature: 200 Celsius
- Material: Stainless Steel SA-240

IOSR Journal of Engineering (IOSRJEN) **www.iosrjen.org**
ISSN (e): 2250-3021, ISSN (p): 2278-8719
PP 18-22

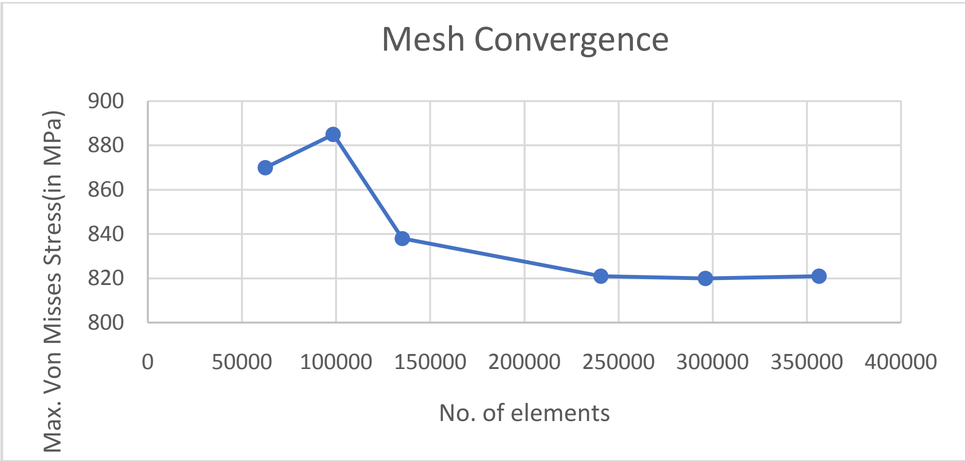

Fig. 2 Mesh convergence

B. Geometry

- ID: 2156 mm
- OD: 2236 mm
- No. of Nozzles: 13
- Manhole ID: 570 mm
- Manhole thickness: 20 mm
- Pad Inner Diameter: 570 mm
- Pad Outer Diameter: 650 mm
- Pad thickness: 40 mm

C. Material Data

- Composition: Cr-22%, Ni-5%, Mo-3%
- Properties:
  - Density – 782 kg/m3
  - Poisson's ratio – 0.3
  - Young's Modulus – 190 GPa
  - Min. Yield strength – 448.15 MPa

D. Forces and boundary conditions:

Table no 1: Forces and Boundary Conditions

Nozzle thrust	A	116244.7311N
Nozzle thrust	B	116244.7311N
Nozzle thrust	C	116244.7311N
Nozzle thrust	D	116244.7311N
Nozzle thrust	E	116244.7311N
Nozzle thrust	F	116244.7311N
Nozzle thrust	G	1530279N
Nozzle thrust	H	429286N
Nozzle thrust	I	197744.68N
Nozzle thrust	J	109393.32N
Nozzle thrust	K	776377.56N
Nozzle thrust	L	108819.84N
Nozzle thrust	M (motor weight)	0, 0, -14000N
Internal pressure	P	6 MPA
Displacement	LUGS	FREE,0,0
Std. earth gravity		0,0, -9.81 m/s^2
Moment due to motor		0 , 4496000N-m, 6366000

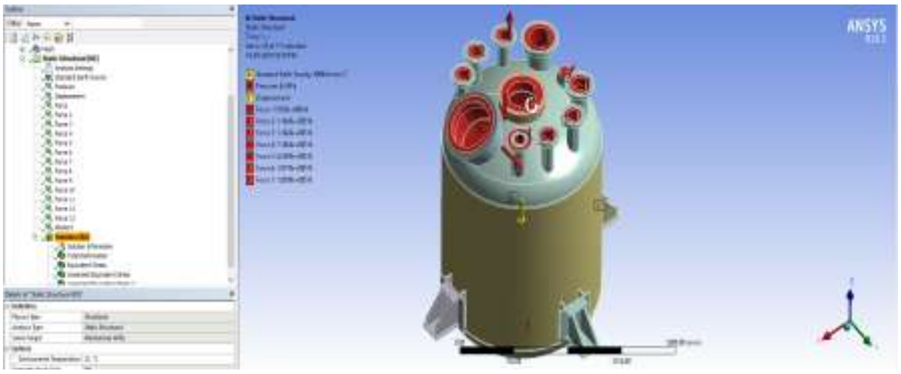

Fig. 3 Forces and Boundary Conditions

V. Results

The stress values are calculated for two conditions. First without reinforcement pad and then with reinforcement pad. Equivalent von-misses stresses are calculated using ANSYS software along with the maximum displacements

1. **Without Reinforcement Pad.**

- Overall max. Deformation: - 821.83 MPa
- Max. Von misses stress value: - 6.2615 mm

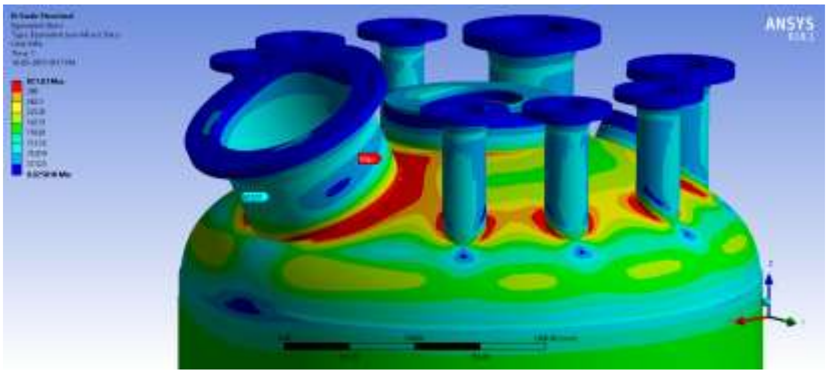

Fig. 4 Stress distribution for vessel without Reinforcement Pad

2. **With reinforcement pad**.

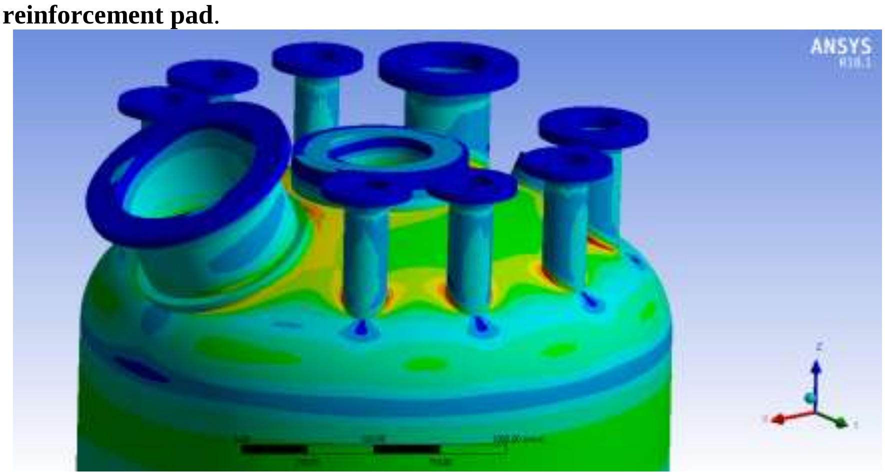

Fig. 5 Stress distribution for vessel with Reinforcement Pad

- Overall max. deformation :- 356.24 MPa
- Max. Von misses stress value :- 4.7779 mm

RF pad not only decreases the maximum stresses induced but as can be seen from the above images it also improves the stress concentration around the junction.

VI. Conclusion

1. The accuracy of the Finite Element Analysis is extremely dependent on the type and method of mesh employed, particularly if higher order mesh (cubic, quadratic and so on.) components are not utilized. For most parts, a better mesh will deliver more accurate outcomes than a coarser mesh. Sooner or later, one achieves the state of convergence.
2. Type of mesh employed also affects the results in a very significant manner. Hex mesh is generally preferred since the computation times are fairly less for hex mesh.
3. Maximum Equivalent Stresses are calculated for the two cases. It can be seen that the maximum stress always occurs at the junction of the nozzle and the pressure vessel. The primary reason for such high stress concentration is the irregular geometry and discontinuities on the shell.
4. The Reinforcement pad significantly improves the strength of the junction without having to increase the dish end thickness.

References

[1] PravinNarale, Prof. P. S. Kachare ,Structural Analysis of Nozzle Attachment on Pressure Vessel Design, International Journal of
[2] Engineering Research and Applications (IJERA),1353-1358, 2012.
[3] Fang, J., Tang, Q.H., and Sang Z.F., A comparative study of usefulness for pad reinforcement in cylindrical vessels under external load on
[4] nozzle, International journal of pressure vessels & piping, 273-279, 2009.
[5] Skopinsky, V.N. and Smetankin A.B. ,Modelling and stress analysis of nozzle connections in ellipsoidal heads of pressure vessels under
[6] external loading, Int. Journals of Applied Mechanics and Engineering, vol.11, No.4, pp.965-979, 2006.
[7] Steven R. Massey, Charlie S. Hsieh, Results of FEA analyses at nozzle/shell junctions subjected to external loads.