Design and Analysis of Reinforcement pad at Nozzle junction on Pressure Vessel

Ajay Teja Denduluri, Sivasai Varunraj Dunna, Harsha Chandan Prasada, Sai Chandu Kuncha and Sundara Ramam Rapeta
Department of Mechanical Engineering
Vignan's Institute of Information Technology, Visakhapatnam, India
ajayteja2000@gmail.com, sivasaivarunrajdunna@gmail.com, harshachandan06@gmail.com, saichandukuncha1@gmail.com, sundar.ramam@gmail.com

Abstract

The goal is to learn more about how a geometric gap between a cylindrical or spherical shell and a reinforcing pad influences stress intensity at the nozzle penetration while the pressure vessel is under internal pressure. In a number of tanks, pressure vessels, and other vessels, nozzles serve as the input and outflow for process and cooling fluid. Various applications such as heat exchangers to make the Nozzle hole in the shell, the shell strength may be lowered. To boost strength and prevent failure, a reinforcement pad is used at the nozzle junction. As a result, the shell nozzle. A multitude of factors are considered when determining whether or not a pad is necessary at the junction. If a pad is needed, the thickness and diameter are estimated using ASME standards. The stress created at the junction is also calculated using a finite element analysis (FEA) of the junction. The magnitudes and distributions of the local stresses induced by the geometric discontinuity and internal pressure loading are thus uncertain. For a variety of reasons, perfect contact between the shell and the pad cannot be maintained, resulting in a gap. The effect of the gap on the stresses in the nozzle reinforcement region is of interest to both designers and manufacturers.

Keywords
Finite Element Analysis, Shell, Pressure Vessel, Reinforcement Pad, Meshing

1. Introduction
Pressure vessels, pipe tees, boilers, and reactors are examples of structural design applications where safe and affordable design criteria must be met. The most common forms of nozzle connections in many technological applications are those subjected to internal pressure and external loads. The application of adequate stress-relieving reinforcements is one of the challenges with nozzle connection design. To ensure the safety of nozzle connections, many types of connections are used. These connections include welded pad reinforcement, self-reinforced nozzles, and internally protruded connectors and toros transitions. A variety of studies have been conducted to examine pressure vessel safety under various loading situations due to the relevance of pressure vessels in engineering applications and the potential of safety concerns in the case of an accident. There are a variety of codes that detail the rules and regulations that must be followed to ensure that equipment is constructed safely. Extensive study has been done on the stresses surrounding nozzle connectors. It is possible to correctly analyze stresses at cylindrical connections in order to ensure a safe and cost-effective design. Traditional pressure vessel design regulations, such as ASME Section VIII, cannot accommodate all design scenarios. The Code, for example, does not address external stresses on nozzles. In such cases, engineers must deviate from the Code and apply recognized design procedures such as finite element analysis (FEA). WRC 107/297, as well as other simplified calculation techniques used in the PVP industry, are based on limited test data and have geometric constraints. The findings become erroneous if certain geometric limitations are not followed. There are no limitations to finite element analysis, and it can provide correct findings in any case.

1.1 Objectives
The main aim of this work is to Design and Analysis of Reinforcement pad at Nozzle junction on Pressure Vessel. To do the Analysis part perfectly, firstly we have to design complete assembly of pressure vessel with reinforcement

pads and without reinforcement pads. After designing both assemblies having with reinforcement pad and without reinforcement pad in NX software. The analysis of both assemblies are done in Ansys software.

2. Literature Review

Y. Bangash (1979) addressed step by step analysis for the direct computation of two-dimensional heat flows and safe pitching of the cooling pipes. Two models of cooling system have been selected and calculations are carried out for an existing vessel. V.N. Skopinsky and A.B. Smetankin (2000) presented the structural modeling and stress analysis of nozzle connections in ellipsoidal head subject to external loading. A parametric study of the effects of geometric parameters on the maximum effective stresses in the ellipsoid-cylinder intersections under loading was performed. The results of the stress analysis and parametric study of the nozzle connections are presented. M.H. Toorani (2003) presented a general approach, based on shearable shell theory, to predict the influence of geometric non-linearities on the natural frequencies of an elastic anisotropic laminated cylindrical shell incorporating large displacement and rotation is presented in this paper. In part two, the modal coefficients are obtained for these displacement functions. You-Hong Liu et al. (2004) elaborates the importance of limit pressures and corresponding membrane stresses. Limit pressure and maximum local membrane stress concentration factor (SCF) are assessed for two orthogonally intersecting thin-walled cylindrical shells subjected to internal pressure. Limit pressures are calculated using inelastic analysis by 3D finite element method. Donald Mackenzie et al. (2008) elaborates the study of tori spherical pressure vessel head. This type of vessel exhibits complex elastic-plastic deformation and buckling behavior under static pressure. Author has assessed both of these behavior modes while specifying the allowable static load. By the direct route in EN code inelastic analysis is used. Chaaba (2010) presented to deal with plastic collapse assessment for thick vessels under internal pressure, thick tubes in plane strain conditions, and thick spheres, taking into consideration various strain hardening effects and large deformation aspect. As a result of this proposal, the limit pressure evolution is obtained, which could cause the plastic collapse of the device for different levels of hardening and also hardening variables such as back-stresses with respect to the geometry change. Shafique M.A. Khan (2010) presented and got the result of stress distribution in a horizontal pressure vessel and the saddle support. The results are obtained from a three dimensional finite element analysis. A quarter of the pressure vessel is modeled with realistic details of saddle support. Physical reasons for favoring of a particular value of ratio of distance of support from the end of the vessel to the length of vessel are outlined. L. Xue et al. (2010) presented with analyzing of burst pressure of cylindrical cell. An elastic plastic large deflection analysis method is used to determine the burst pressure and fracture location of cylindrical shell intersection by use of nonlinear finite element analysis. To verify the accuracy of finite element result author carried out experimental burst test by pressurizing test vessels with nozzles to burst. Sotiria Houliara (2011) elaborates the study of structural response and buckling of long unstiffened thin walled Cylindrical steel shells, subjected to bending moments, with particular emphasis on stability design. Using a nonlinear finite element technique, the bifurcation moment is calculated, the post-buckling response is determined, and the imperfection sensitivity with respect to the governing buckling mode is examined. R.C. Carbonari and Pablo Munoz-Rojas (2011) elaborates the study of shape optimization of axisymmetric pressure vessel considering an integrated approach in which entire pressure vessel model is used in conjunction with a multi-objective function that aims to minimize the von-mises mechanical stress from nozzle to head. The different shapes from usual one are obtained. Even though such different shapes may not be profitable considering present manufacturing processes, they may competitive for future manufacturing technologies and contribute to better understanding of the actual influence of shape in the behavior of pressure vessels. Daniel Vasilikis and Spyros A Karamanos (2011) elaborates the study of using a two-dimensional model with nonlinear finite elements, which accounts for both geometric and material nonlinearities, the structural response of those cylinders is investigated. This investigation is used to develop relevant design guidelines. Bandarupalli Praneeth and T.B.S.Rao (2012) presented the Features of multilayered high pressure vessels, their advantages over monoblack vessels are discussed in this paper. Various parameters of solid pressure vessel are designed and checked according to the principles specified in American Society of Mechanical Engineers (ASME) Section VII Division 1. The stresses developed in solid wall pressure vessel and multilayer pressure vessel is analyzed by using ANSYS. The solid wall thickness is 12mm and of layers thickness is 6mm. Kiran D. Parmar1 and Kiran A. Patel (2013) presented a simple method of estimating limit loads using a sequence of elastic finite element analyses and the lower bound theorem, termed elastic compensation, is demonstrated on the problem of the estimation of the limit behavior of torispherical pressurevessel heads. Author concluded that stress and other parameters are also decreased by changing the weld size of the skirt to dished end joint. S.Sanyasinaidu and K. Chandanarao (2013) has carried out thermo mechanical analysis of high pressure vessel with dished end. Solid, multi cylinder and hemisphere dish end are designed according to ASME. The results are compared with simulation software ANSYS and thermal analysis are performed according to design temperature and results are imported in to structural analysis to find out thermal stresses. Vishal V. saidpatil and Arun Thakare (2014) presented the detailed design & analysis of Pressure vessel used in boiler for optimum thickness, temperature distribution and dynamic behavior using Finite element analysis software. Weight optimization of pressure vessel is carried out by considering thickness as a major factor. The thickness is optimized by changing material and also the composition. Hardik B nayak and R R trivedi (2015) worked on stress analysis of reactor nozzle to head junction subjected to applied to external load, internal pressure and moments under different loading condition, the stress will be occurred at the nozzle to head or shell junction area. The reason for this is the discontinuity of geometry defect will occur and the junction region will be failure source of the whole structure. Patel Nikunj and Ashwin Bhabhor (2019) evaluated the design of reactor pressure vessel designed by "New blue moon engineers" and done new design of some major parts. The design is compared through experimental and analytical base modeling and thermal analysis by using advanced CAE tools. So it gives the best design which is feasible for reactor pressure vessel. This paper gives some of the important information, knowledge and analytical calculation and comparison of existing and new design of vessel to empower the basic fundamentals to carry out work.

3. Methodology

Process of the work:

- The given Pressure Vessel data sheets are extensively checked, and all needed standard dimensions, such as WRNF 150, as well as NPS numbers as reference numbers, are accepted as dimensional references.
- Auto-CAD is used to create parts in 2D (line diagrams) using standard measurements and data from data sheets, and part drawings are combined to create a whole drawing.
- 3D modelling is done on the components in Siemens NX 11.0, and then the parts are joined to finish the assembly.
- The pressure vessel analysis is simulated using Ansys software, and the results are observed.

3.1 Design of Pressure Vessel

With minimal changes to the design parameters, all of the parts were modelled in NX 11.0 using the same approach. Design criteria and design standards are taken into account according to pressure vessel data sheets (Figure 1-17).

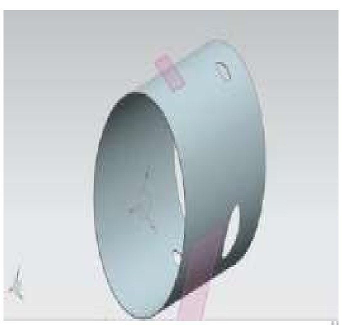

Figure 1: 3D sketch of Shell 1

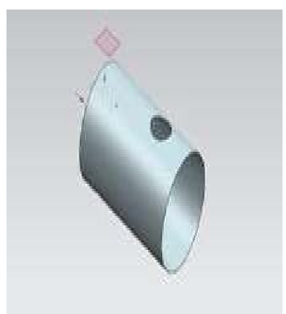

Figure 2: 3D sketch of Shell 2

Figure 3: 3D sketch of Skirt

Figure 4: 3D sketch of Dishend 1

Figure 5: 3D sketch of Nozzle 1 & 2

Figure 6: 3D sketch of Nozzle 4,5,6,7

Figure 7: 3D sketch of Nozzle 8

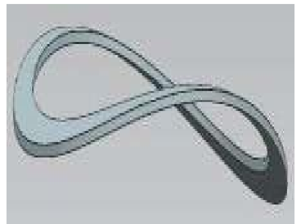

Figure 8: 3D sketch of Reinforcement Pad 1 & 2

Figure 9: 3D sketch of Nozzle 3

Figure 10: 3D sketch of Manhole

Figure 11: 3D sketch of Dishend 2

Figure 12: 3D sketch of Manhole Blank

Figure 13: 3D sketch of MH RF Pad

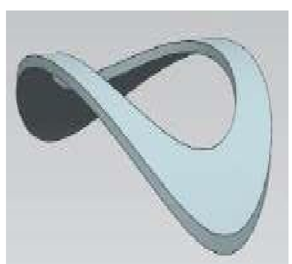

Figure 14: 3D sketch of Reinforcement Pad 8

Figure 15: 3D sketch of RF Pad 1

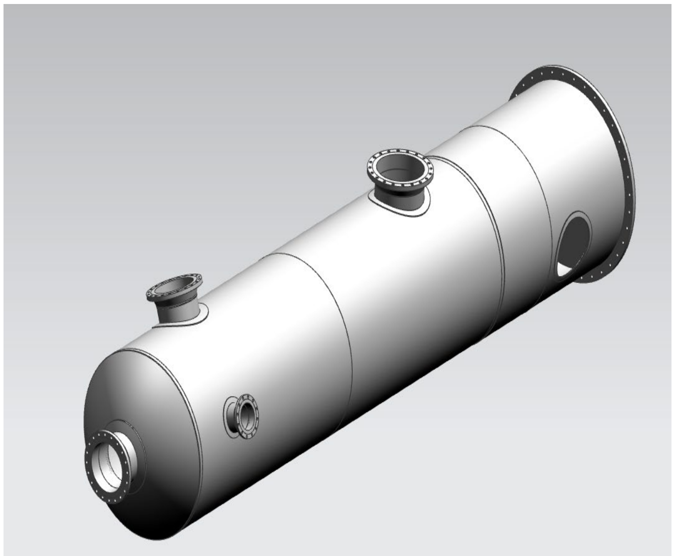

Figure 16: Complete Assembly of Pressure vessel with RF Pads

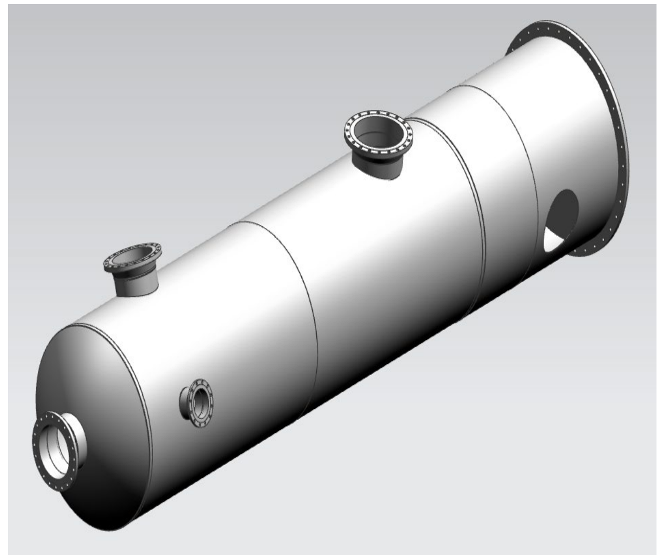

Figure 17: Complete Assembly of Pressure vessel without RF pads

The above Figures shows all the Designed parts of Pressure vessel and Complete Assembly design of Pressure vessel with Reinforcement pads and without Reinforcement pads which all are designed using NX 11.0.

3.2 Formation of a mesh

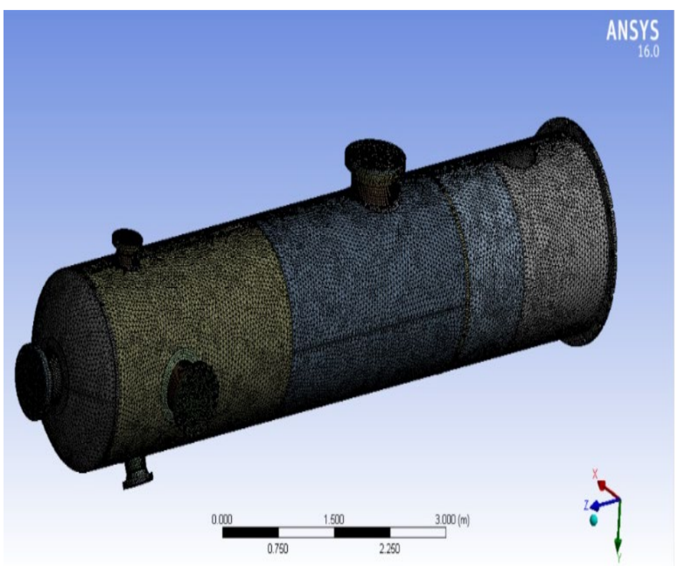

Figure 18: Geometry model of Pressure vessel With RF Pads

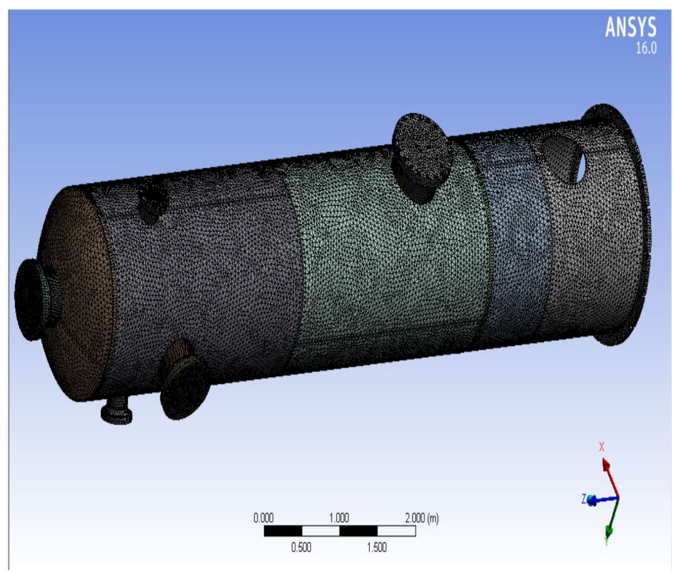

Figure 19: Geometry model of Pressure vessel Without RF Pads

ANSYS workbench was used to do structural analysis of character tool tiers. The assessment is done on a 64-bit operating system with 16GB of RAM and ANSYS 16.0. This arrangement may be ideal for running analytic applications without difficulty. On ANSYS file geometry, the previously prepared IGES file format is loaded. Modal analysis and harmonic analysis are both possible in ANSYS. The IGES Model, which we produced in the NX software application, is imported into ANSYS, with all measurements in millimeters. As stated above in the imported version. Set of tools for the first step. Is broken down into little length parts. To split the combination, the Triangular and Quadrilateral mesh approach is used. Following meshing is done, the model now has 269706 elements and 562802 nodes.

4. Data Collection

The required data is collected from Data sheets

- The Overall length of the Pressure vessel is 9010.65 mm
- Tan to Tan length is 5943.6 mm
- Pad thickness is 16 mm
- Shell thickness is 16 mm
- Head thickness is 20 mm
- Design pressure is 345 kPa
- Design temperature is 343.3 ^{0}C

5. Results and Discussion

5.1 Analysis of Pressure Vessel

All of the parts were examined with ANSYS 16.0 in accordance with the approach. Analysis parameters are compared to design specifications using Pressure Vessel data sheets (Figure 20-30).

5.1.1 Assembly analysis without Reinforcement pad

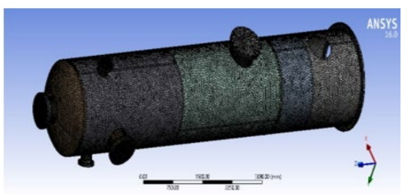

Figure 20: Meshing of total assembly

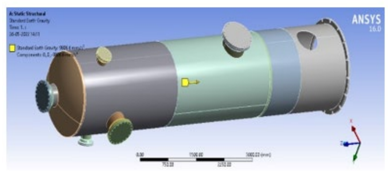

Figure 21: Standard Earth Gravity

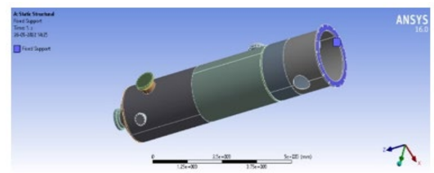

Figure 22: Fixed Support

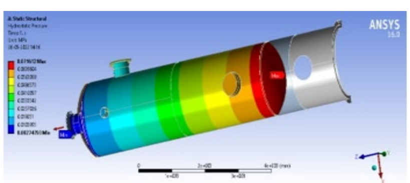

Figure 23: Hydrostatic Pressure

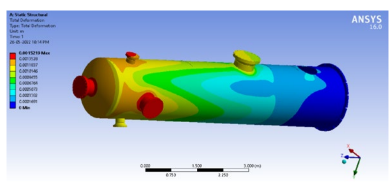

Figure 24: Total Deformation (Static Structural)

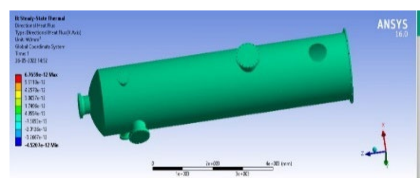

Figure 25: Equivalent Elastic Strain (Static Structural)

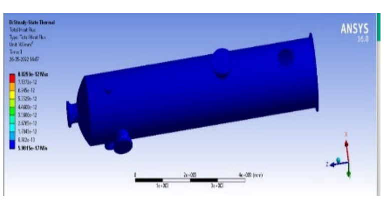

Figure 26: Total Heat Flux (Steady State thermal)

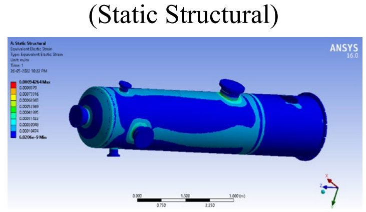

Figure 27: Directional Heat Flux (Steady State thermal)

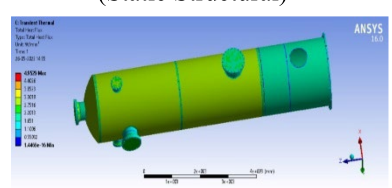

Figure 28: Total Heat Flux (Transient thermal)

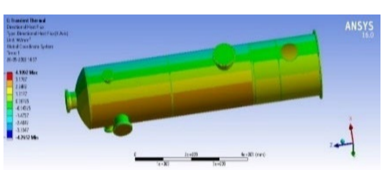

Figure 29: Directional Heat Flux (Transient thermal)

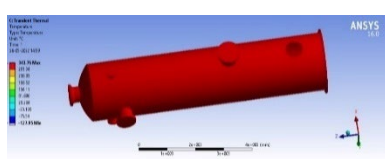

Figure 30: Temperature (Transient thermal)

5.1.2 Assembly analysis with Reinforcement pad

The Figure (31-45) present different meshing and temperature graphs.

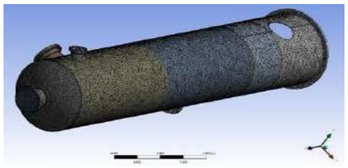

Figure 31: Meshing of total assembly

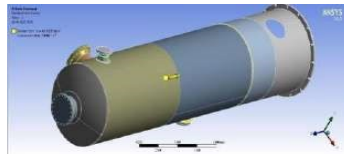

Figure 32: Standard Earth Gravity

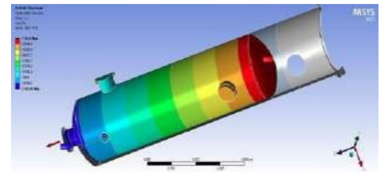

Figure 33: Hydrostatic Pressure

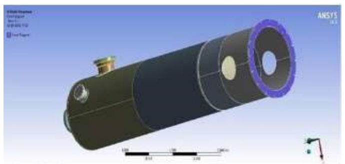

Figure 34: Fixed Support

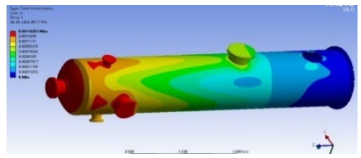

Figure 35: Total Deformation (Static Structural)

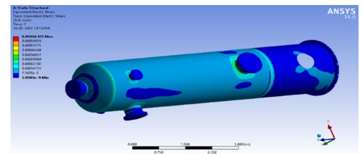

Figure 36: Equivalent Elastic Strain (Static Structural)

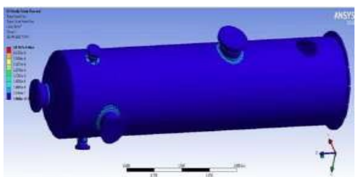

Figure 37: Total Heat Flux (Steady State Thermal)

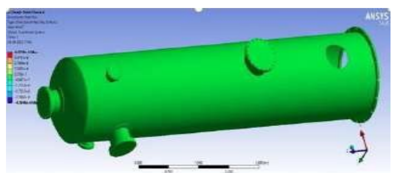

Figure 38: Directional Heat Flux (Steady State Thermal)

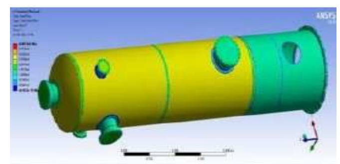

Figure 39: Total Heat Flux (Transient Thermal)

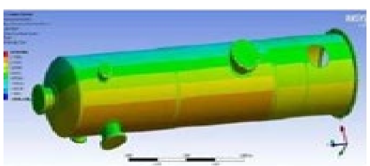

Figure 40: Directional heat flux (Transient thermal)

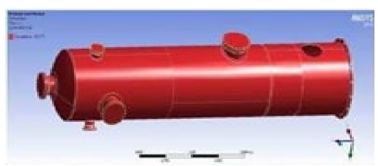

Figure 41: Design temperature (Transient thermal)

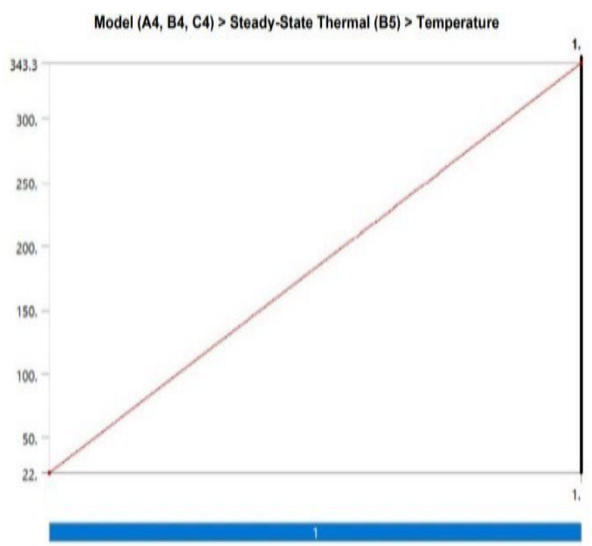

Figure 42: Steady State Thermal Temperature Graph

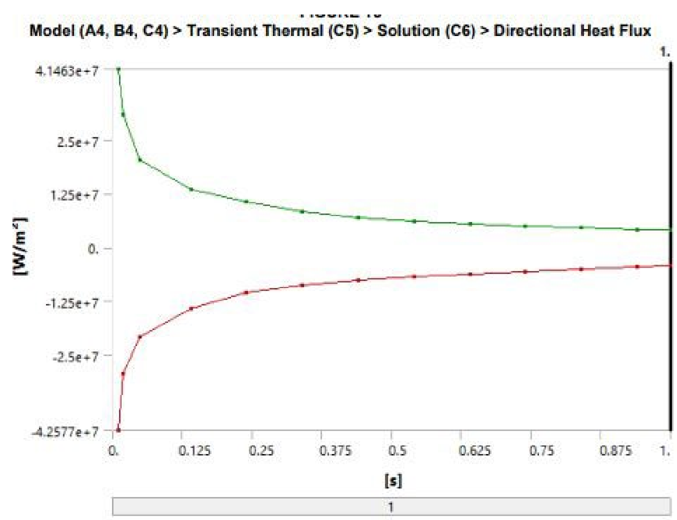

Figure 43: Directional thermal heat flux Graph

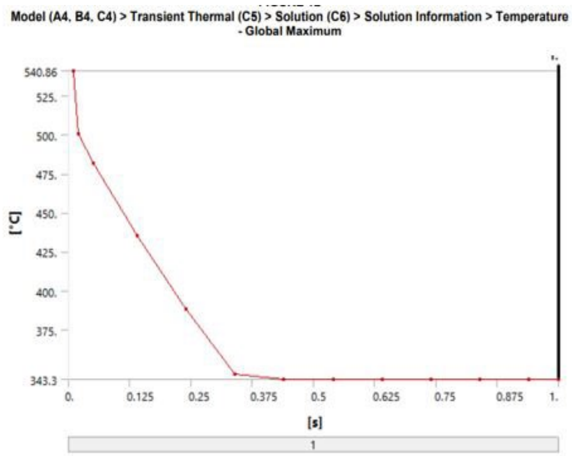

Figure 44: Transient thermal Global maximum Graph

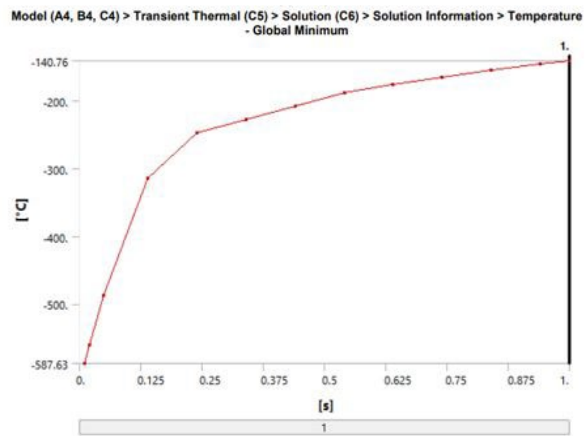

Figure 45: Transient thermal Global minimum Graph

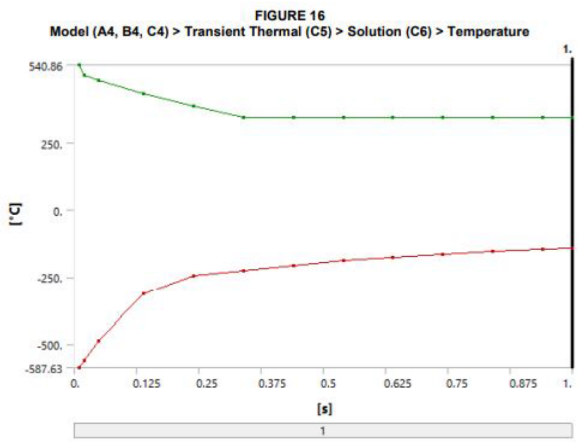

Figure 46: Transient thermal temperature Graph

5.1.3 Deformation of Pressure Vessel

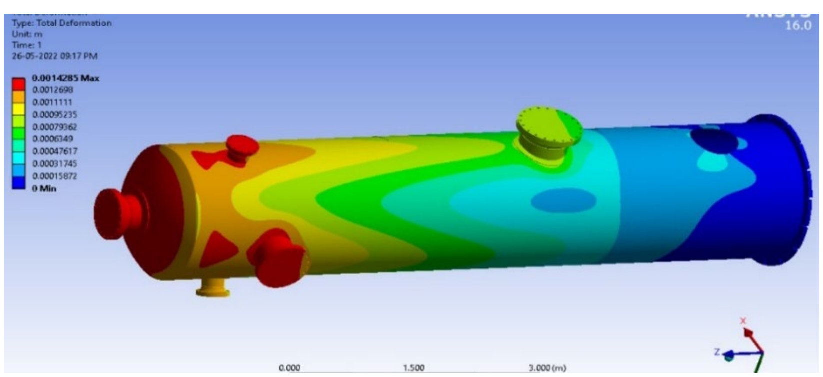

Figure 47: Total Deformation of Pressure vessel With RF Pads

The structural characteristics was evaluated using an Ansys analysis (Figure 46 and 47). The structural solver in ANSYS is used to execute the analysis after the geometry has been created and imported. The Pressure Vessel has been correctly meshed to get best output. Based on this, a structured mesh for the 3D model was created. After meshing, the domain has Triangular and Quadrilateral elements with 562802 nodes and 269706 elements.

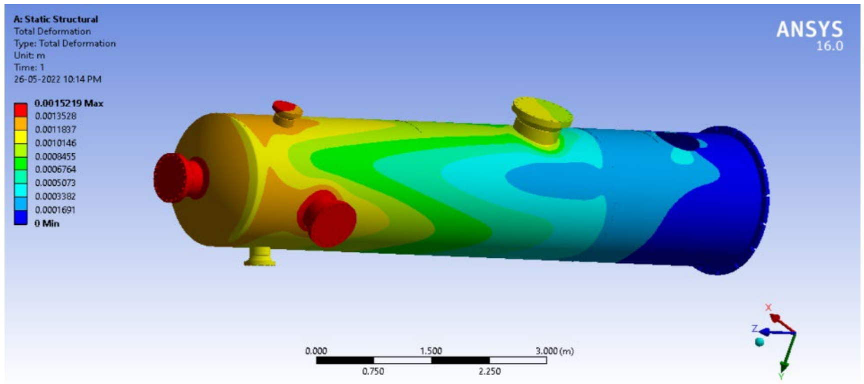

Figure 48: Total Deformation of Pressure vessel Without RF Pads

When pressure is applied to the overall faces of the Pressure vessel, the total deformation of the Pressure vessel is shown in the diagram above; the red region indicates the most deformation, while the blue region represents the least deformation. With RF Pad, For Total Deformation , Less deformation occurs with 1.4285e-003 m Maximum value. Without RF Pad , For Total Deformation , More deformation occurs with 1.5219e-003 m Maximum value. With RF Pad , For Equivalent stress , Less distortion occurs with 1.3207e+008 Pa Maximum value. Without RF Pad , For Equivalent stress , More distortion occurs with 1.8215e+008 Pa Maximum value. As a result, If we use RF Pad we have improvement in strength compared to without RF Pad (Figure 48 and 49).

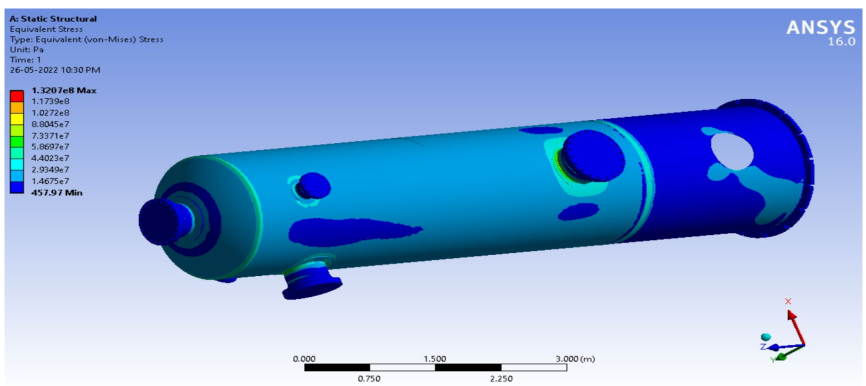

Figure 49: Equivalent Stress on Pressure vessel with RF Pad

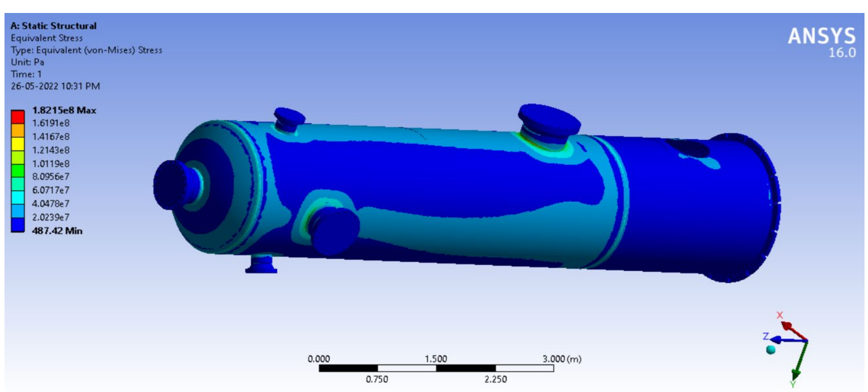

Figure 50: Equivalent Stress on Pressure vessel without RF Pad

The Difference for Pressure Vessel with Reinforcement Pad and without Reinforcement Pad is done only in Static Structural analysis system. The Difference is shown in Percentage representation below for Total Deformation, Equivalent Stress, Equivalent Elastic Strain (Figure 50).

Table 1. Pad and Without RF Pad

S.NO	Solution	With RF Pad	Without RF Pad	Percentage Reduction
1	Total Deformation	1.428	1.5219	6.17%
2	Equivalent Stress	1.3207	1.8215	27.49%
3	Equivalent Elastic Strain	6.6305	9.4264	29.66%

By this above calculations (Table 1) we say that, by keeping the RF Pad at nozzle junction is safe for pressure vessel and by these values we say that, the Total Deformation has been reduced by 6.17% , Equivalent stress has been reduced by 27.49% , Equivalent elastic strain has been reduced by 29.66% by keeping RF Pad.

6. Conclusions

A pressure vessel's Design Pressure, Design Temperature and component dimension are all designed according to ASME pressure vessel specifications. Under various loads, the pressure vessel's Reinforcement pads, shell, Blind Flange and Shell junction area were examined using FEA and ASME methodologies. The material's allowable stress is less than the stress equivalent and pressure vessel component stress classification lines. The analysis conclusions for the normal operating state were within acceptable bounds. As a consequence, the current Reinforcement pads, shell, Blind Flange and Shell junction area designs are capable of sustaining the desired load conditions. The pressure vessels are designed to be safe. The level of safety that we consider acceptable and by which we judge the design to be safe. The bursting pressure is below the design's permissible stress, ensuring that it does not fail. And because the analysis is so near to the analytical design, both the data and the design are regarded safe. In addition, no pressure vessel failures have occurred. By incorporating the RF pad the von-mises stresses are reduced by 27.49%, Total deformation are reduced by 6.17%, Equivalent elastic strain has been reduced by 29.66%.

References

Bangash,Y., Safe Analysis for Cooling Pipes for Prestressed Concrete Reactor Vessels, *Journal of Nuclear Engineering and Design*, Volume 55, Issue 3, December 1979.

Skopinsky,V.N., and Smetankin,A.B., Stress in Ellipsoidal Pressure Vessel Heads with Non-Central Nozzle , *Journal of Nuclear Engineering and Design*, Volume 198, Issue 3, June 2000.

Toorani,M.H., Dynamics of the Geometrically Non-Linear Analysis of Anisotropic Laminated Cylindrical Shells , *International Journal of Non-Linear Mechanics*, Volume 38, Issue 9, November 2003.

You-Hong,Liu., Bing-sheng,Zhang and Ming-De,Xue., Limit Pressure and Design Criterion of Cylindrical Pressure Vessels with Nozzles , *International Journal of Pressure vessel and Piping*, Volume 81, Issue 7, July 2004.

Donald,Mackenzie., Camilleri and Robert Hamilton., Design by Analysis of Ductile Failure and Buckling in Tori-spherical Pressure Vessel Heads, *Journal of Thin Walled Structures*, Volume 46, Issues 7-9, July to September 2008.

Chaaba., Plastic Collapse Assessment Of Thick Vessels under Internal Pressure According to Various Hardening Rules, *Journal of Pressure Vessel Technology*, Volume 132, Issue 5, October 2010.

ShafiqueKhan,M.A., Stress Distributions in A Horizontal Pressure Vessel and the Saddle Supports, *International Journal of Pressure Vessel and Piping*, Volume 87, Issue 5, May 2010.

Xue,L., Widera,G.E.O and sang,Z., Parametric FEA Study of Burst Pressure of Cylindrical Shell Intersections, *Journal of Pressure vessel Technology*, Volume 132, Issue 3, June 2010.

Sotiria Houliara., Buckling of Thin-Walled Long Steel Cylinders Subjected to Bending, *Journal of Pressure Vessel Technology*, Volume 133, Issue 1, February 2011.

Carbonari,R.C. and Pablo Munoz-Rojas., Design of Pressure Vessels using Shape Optimization: An Integrated Approach, *International Journal of Pressure Vessel and Piping*, Volume 88, Issue 5, May 2011.

Daniel,Vasilikis. and Spyros Karamanos,A., Buckling Design of Confined Steel Cylinders under External Pressure, *Journal of Pressure vessel Technology*, Volume 133, Issue 1, February 2011.

Praneeth,Bandarupalli. and Rao,T.B.S., Finite Element Analysis of Pressure Vessel and Piping Design, *International Journal of Engineering Trends and Technology*, Volume 3, Issue 5, 2012.

Kiran Parmar1,D. and Kiran Patel,A.,Thermal Analysis for Skirt Dished End Joint of Pressure Vessel using Finite Element Analysis Approach, *International Journal of Modern Engineering Research*, Volume 3, Issue 4 , July-August 2013.

Sanyasinaidu,S. and Chandanarao,K., Design and Thermo Mechanical Analysis of High Pressure Vessels with Dish End, *International Journal of Advanced Research in Science and Technology*, Volume 2, Issue 2, June 2013.

Vishal Saidpatil,V. and Arun Thakare., Design & Weight Optimization of Pressure Vessel Due to Thickness using Finite Element Analysis, *International Journal of Emerging Engineering Research and Technology*, Volume 2 , Issue 3, June 2014.

Hardik Nayak,B. and Trivedi,R.R., Stress Analysis of Reactor Nozzle to Head Junction, *International Research Journal of Engineering and Technology*, Volume 2, Issue 5, August 2015.

Patel,Nikunj. and Ashwin Bhabhor., Design and Analytical Calculation of Reactor Pressure Vessel, *International Journal of Science and Research*, Volume 8, Issue 5, May 2019.

Biographies

Sundara Ramam Rapeta working as Associate Professor in the Department of Mechanical Engineering in Vignan's Institute of Information Technology. His research field is Design and Composite Materials.

Ajay Teja Denduluri, Sivasai Varunraj Dunna, Harsha Chandan Prasada, Sai Chandu Kuncha are the students who are Completed Under-graduation in B.Tech, Mechanical Engineering from Vignan's Institute of Information Technology, Duvvada, Visakhapatnam from 2018 to 2022.