IJESRT

INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY

FINITE ELEMENT ANALYSIS OF STACKED VESSELS

Vidyadhar Biswal*, Rohit Kumar, Shubham Singhmar, Mohit Kumar
* Asstt. Professor, Mechanical Engineering, Chandigarh University India.
Asstt. Professor, Mechanical Engineering, Chandigarh University India.
Student, Department Of Mechanical Engineering, Chandigarh University, India.
Student, Department Of Mechanical Engineering, Chandigarh University, India.

ABSTRACT

The stacked vessels will experience seismic conditions for San Francisco California. These conditions will be analyzed using finite element analysis and the UB-97 code. ASME section IID allowed stresses will be used with VIII-2 rules to determine the acceptability of the design. All internal surfaces are pressurized and openings have exit pressure forces applied.

KEYWORDS: Stacked vessels, Catia, Ansys, FEA.

INTRODUCTION

CAD, or computer-aided design and drafting (CADD), is the use of computer technology for design and design documentation. CAD software replaces manual drafting with an automated process. CADD describes the purpose of streamlining design processes, drafting, documentation, and manufacturing processes.CAD is mainly used for detailed engineering of 3D models and/or 2D drawings of physical components, but it is also used throughout the engineering process from conceptual design and layout of products, through strength and dynamic analysis of assemblies to definition of manufacturing methods of components. It can also be used to design objects.

Introduction to Catia

Initially, CATIA name is an abbreviation for Computer Aided Three-dimensional Interactive Application. Catia is the standard in 3D product design, featuring industry-leading productivity tools that promote best practices in design while ensuring compliance with your industry and company standards.

Introduction to FEA

FEA consists of a computer model of a material or design that is stressed and analyzed for specific results. It is used in new product design, and existing product refinement. A company is able to verify a proposed design will be able to perform to the client's specifications prior to manufacturing or construction. Modifying an existing product or structure is utilized to qualify the product or structure for a new service condition. In case of structural failure, FEA may be used to help determine the design modifications to meet the new condition. FEA uses a complex system of points called nodes which make a grid called a mesh. This mesh is programmed to contain the material and structural properties which define how the structure will react to certain loading conditions. Nodes are assigned at a certain density throughout the material depending on the anticipated stress levels of a particular area. In practice, a finite element analysis usually consists of three principal steps.

- **Preprocessing** - The user constructs a model of the part to be analyzed in which the geometry is divided into a number of discrete sub regions, or elements," connected at discrete points called nodes." Certain of these nodes will have fixed displacements, and others will have prescribed loads. These models can be extremely time consuming to prepare, and commercial codes vie with one another to have the most user-friendly graphical “preprocessor" to assist in this rather tedious chore. Some of these preprocessors can overlay a mesh on a preexisting CAD file, so that finite element analysis can be done conveniently as part of the computerized drafting-and-design process.

- **Analysis** - The dataset prepared by the preprocessor is used as input to the finite element code itself, which constructs and solves a system of linear or nonlinear algebraic equations [K][U]=[F] where u and f are the displacements and externally applied forces at the nodal points. The formation of the K matrix is dependent on the type of problem being attacked, and this module will outline the approach for truss and linear elastic stress analyses. Commercial codes may have very large element libraries, with elements appropriate to a wide range of problem types. One of FEA's principal advantages is that many problem types can be addressed with the same code, merely by specifying the appropriate element types from the library.
- **Post processing** - In the earlier days of finite element analysis, the user would pore through reams of numbers generated by the code, listing displacements and stresses at discrete positions within the model. It is easy to miss important trends and hot spots this way, and modern codes use graphical displays to assist in visualizing the results. Typical postprocessor display overlays colored contours representing stress levels on the model, showing a full field picture similar to that of photo elastic or moiré experimental results.

Introduction to ANSYS

ANSYS is general-purpose finite element analysis (FEA) software package. Finite Element Analysis is a numerical method of deconstructing a complex system into very small pieces (of user-designated size) called elements. The software implements equations that govern the behavior of these elements and solves them all; creating a comprehensive explanation of how the system acts as a whole. These results then can be presented in tabulated, or graphical forms. This type of analysis is typically used for the design and optimization of a system far too complex to analyze by hand. Systems that may fit into this category are too complex due to their geometry, scale, or governing equations.

THE MODEL DESIGN AND BRAKING CONDITIONS

The model has been created in CATIA and imported to Ansys workbench in 'stp' format.

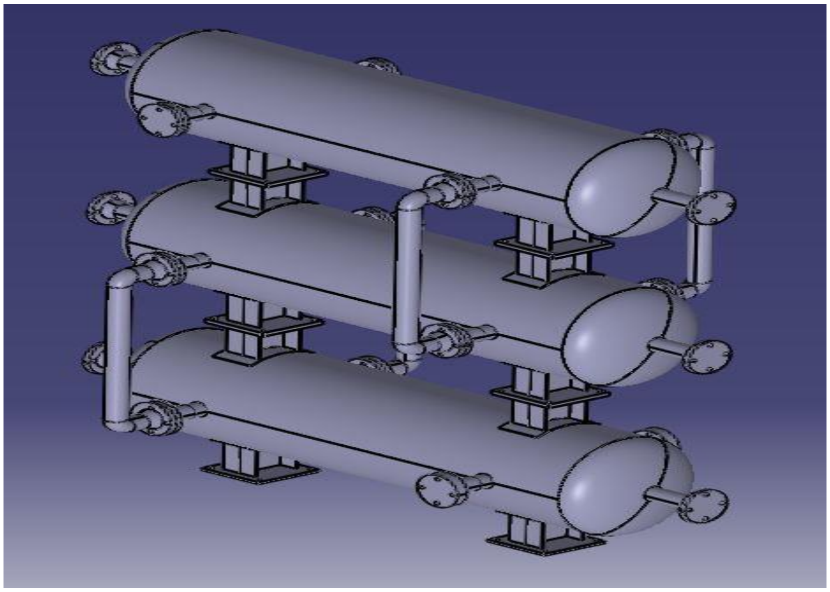

Fig.1-Catia model of the stacked vessels.

Meshing

The meshing of geometry was performed in ANSYS and using 3D tetrahedral elements having global mesh size of 12 inch, corresponding mesh details are as discussed below

MESH DETAILS	
Physical reference	Mechanical
Relevance centre	Fine

Element size	Default
Smoothing	Medium
Transition	Fast
Span angle centre	Coarse
Nodes	187199
Elements	96078

Table 1. Details of meshing

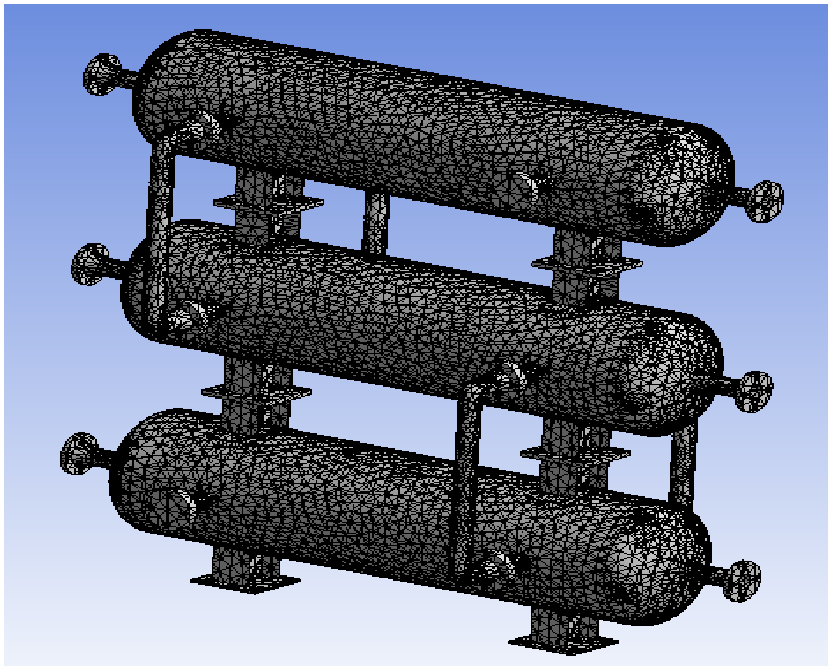

Fig.2.Tetrahedral meshing of the geometry

PROPERTIES

Following two materials were used for the construction of pressure vessel.

	Material 1	Material 2	Material 3	Material 4
Material =	SA-516-70	SA-105	SA-106 B	SA-36
Application =	Shells / Heads	Flanges	Pipe	Supports
Sm [psi] =	20,000	20,000	17,100	16,600
Sy [psi] =				
E1 =	1.0	1.0	1.0	1.0
E2 =	1.0	1.0	1.0	1.0
E [psi] =	29,400,000	29,400,000	29,400,000	29,400,000
v =				
Coef =				

Table2. List of materials used for stacked vessels.

ISSN: 2277-9655
(I2OR), Publication Impact Factor: 3.785

APPLICATION OF LOAD AND BOUNDARY CONDITIONS

The bottom set of saddles are fixed to prevent rigid body motion. The vessel experiences a pressure of 200 psi at all the internal surfaces and a force of 674.372 lbf is applied at the opening.

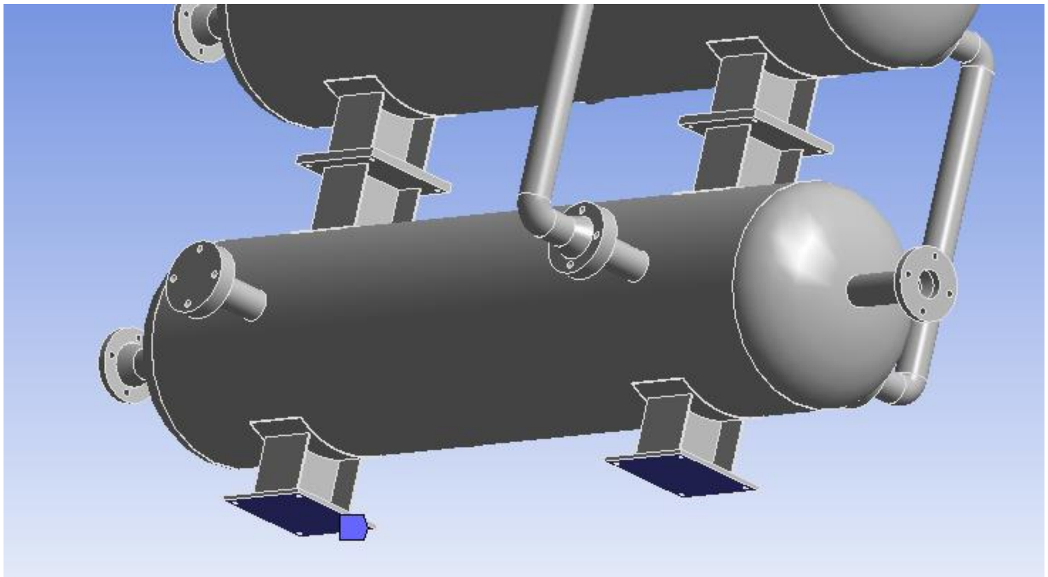

Fig3. Fixed supportat bottom of saddle

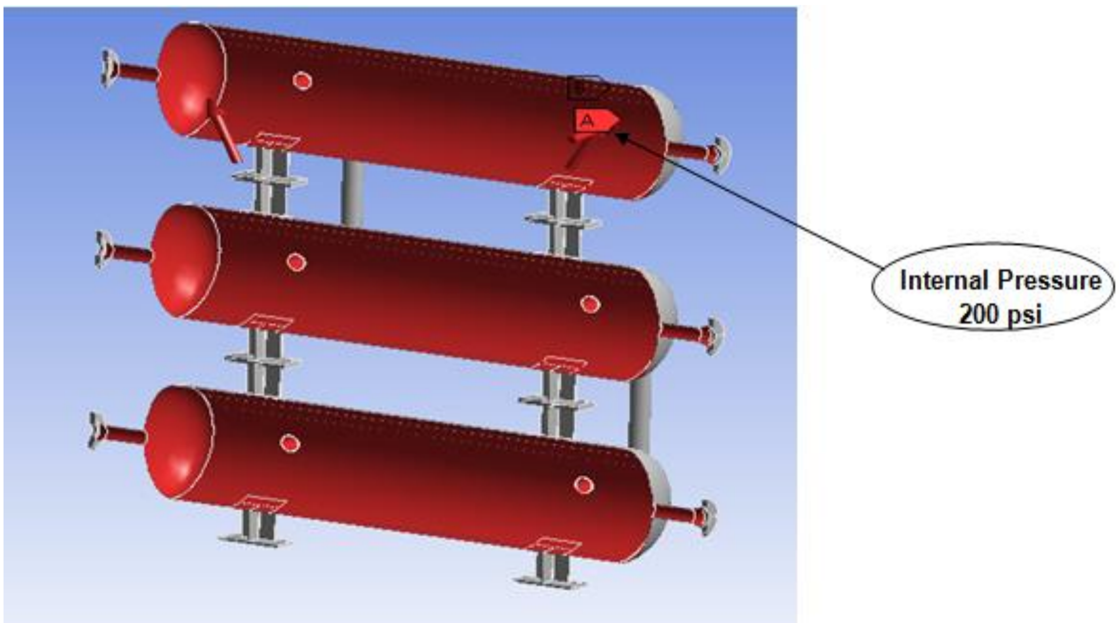

Fig.4 Internal view of stacked vessels.

Standard earth gravity and horizontal 155.26 in/s^2accelerations applied to the model. The horizontal acceleration is used to simulate seismic loads.

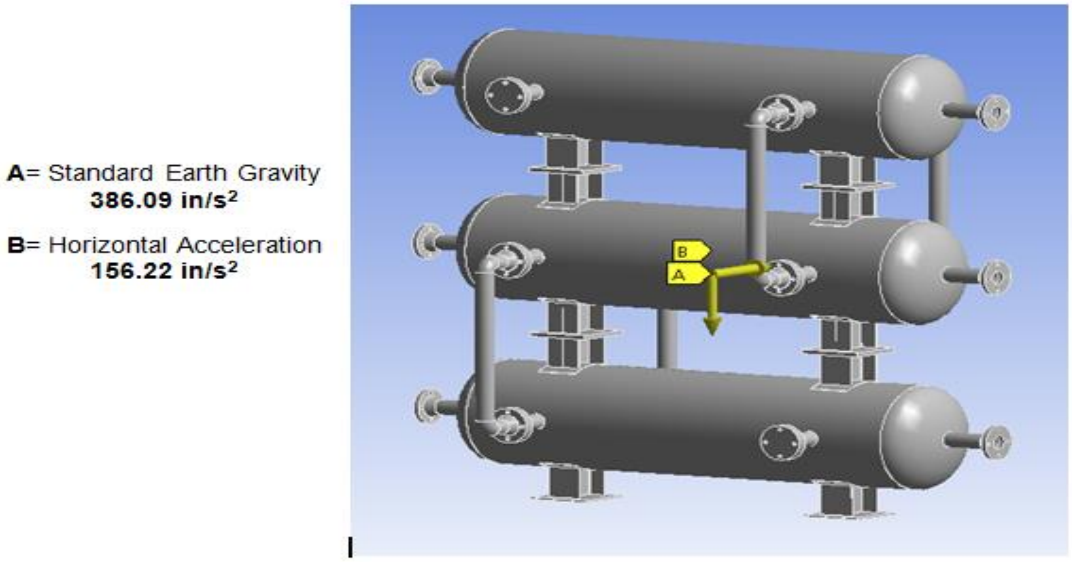

Fig 5. View of vertical and horizontal accelerations

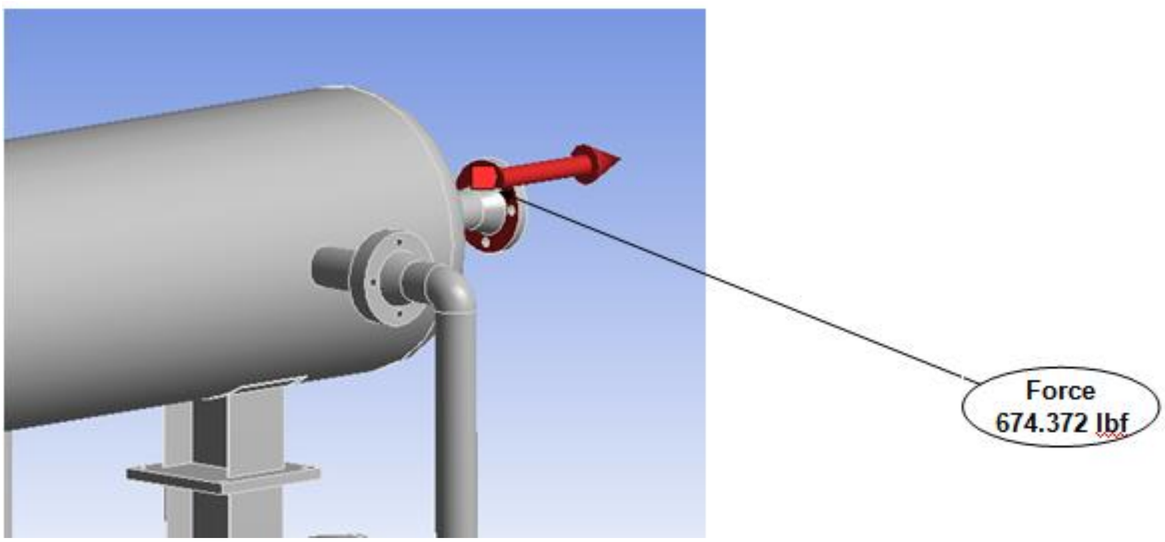

Fig 6.Exit pressure force applied at the opening

RESULT OF STATIC STRUCTURAL ANALYSIS

After solving the above mentioned load in Ansys 16 Workbench following Results obtained in Static Structure.

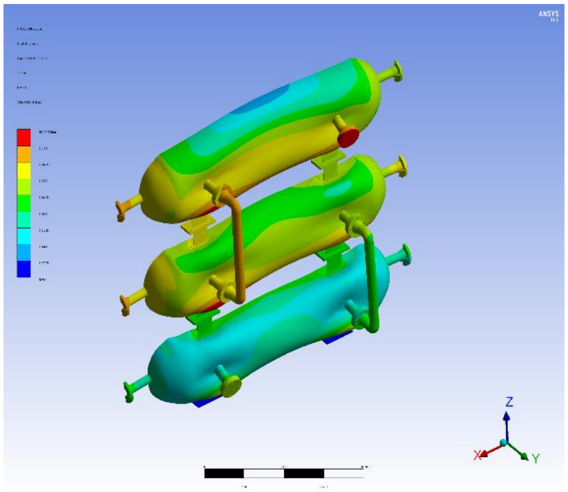

Fig7: Total Deformation

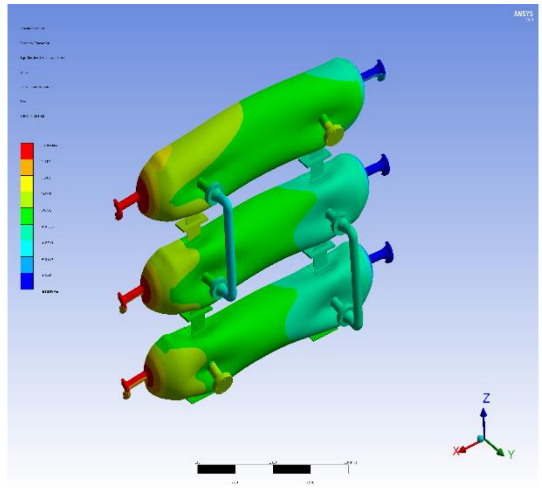

Fig7: Directional Deformation

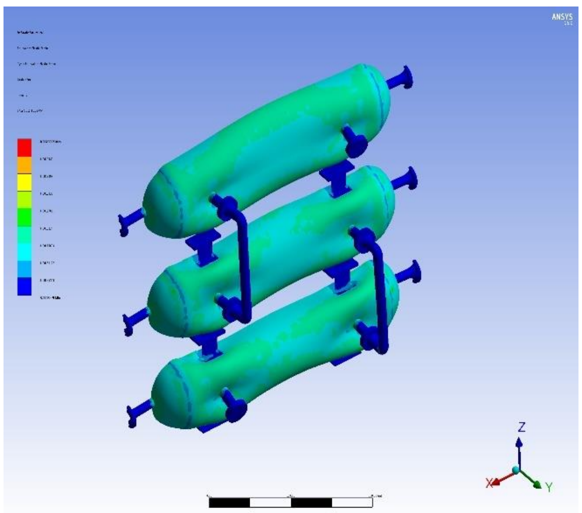

Fig9: Equivalent Elastic Strain

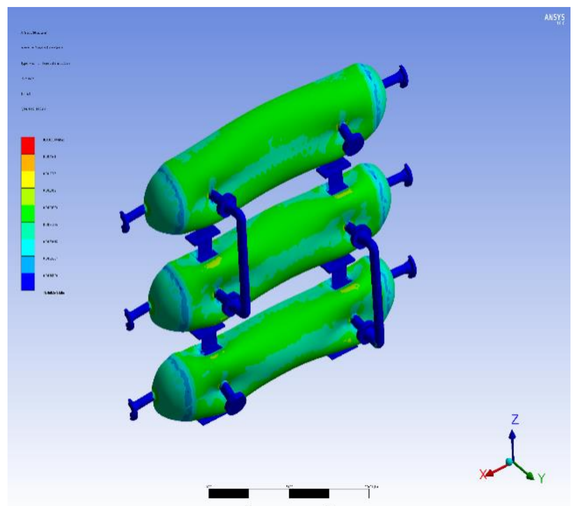

Fig10: Maximum Plastic Elastic Strain

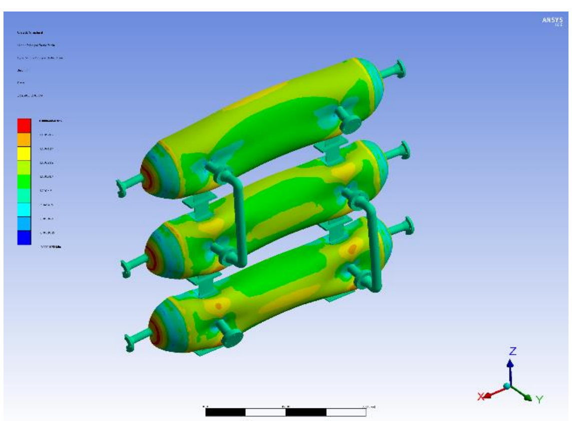

Fig11: Middle Principal Elastic Strain

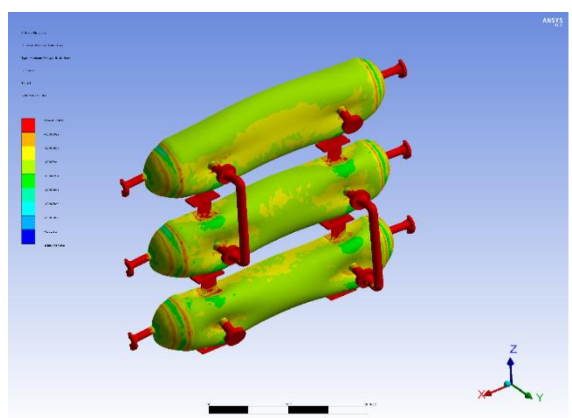

Fig12: Minimum Principal Elastic Strain

ISSN: 2277-9655
(I2OR), Publication Impact Factor: 3.785

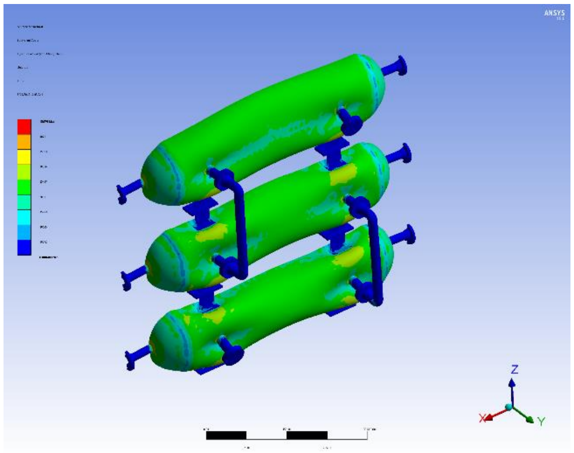

Fig13: Equivalent Stress

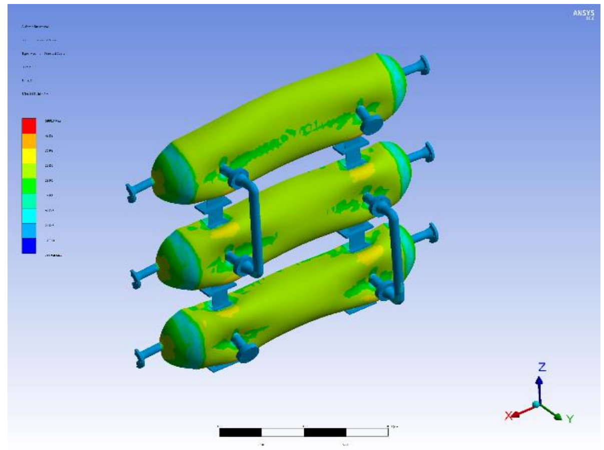

Fig14: Maximum Principal Stress

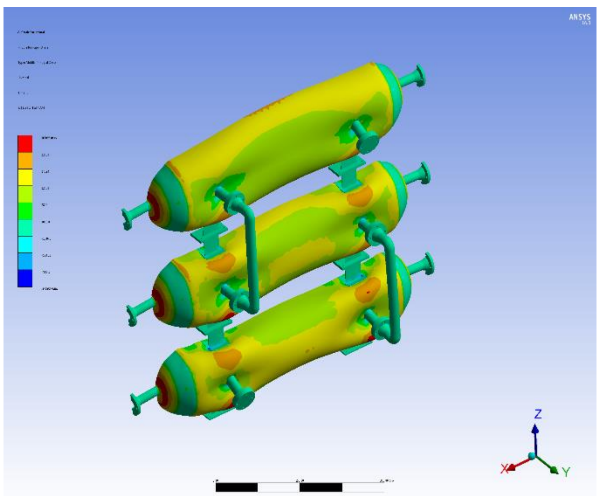

Fig15: Middle Principal Stress

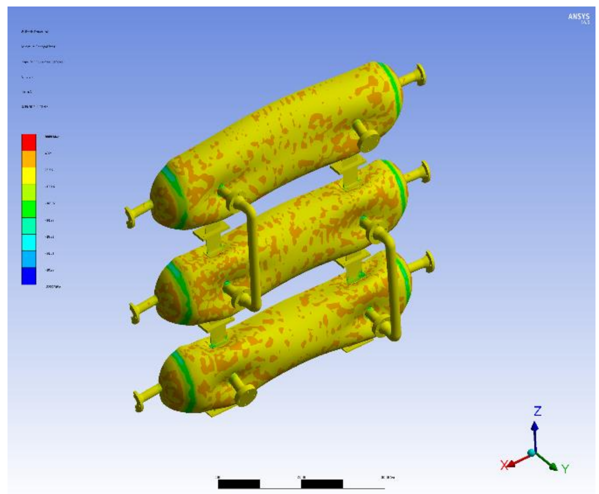

Fig16: Minimum Principal Stress

CONCLUSION

During the process the static load of 200 psi was applied on the inner surfaces. And the fixed support on the on the bottom body and the analysis was carried out on the Stacked Vessel. Results so obtained showed the maximum Von-Misses stress of 45879. And the deformation observed was only 0.64845 From the above static structural it is clear that all checkpoints are satisfied and following conclusion can be drawn. The Stacked-Vessels meets the design rule and the allowable stress is acceptable.

REFERENCES

[1] Joseph C. Slater , Wright Finite Element method: An easily extensible research-oriented finite element code, April 5,2004.
[2] O.C. Zienkiewicz , R.L. Taylor , The Finite Element Method, Fifth Edition, ISBN: 07506 50559
[3] S.S. Rao, The Finite Element Method in Engineering, fourth Edition, ISBN: 0750678283
[4] George S., Kollar, Laszloa P. Mechanics of Composite Structures.Cambridge University Press, New York, 2003
[5] Design Data book, “The McGraw-Hill Publication”.
[6] ASME VIII-1 Code
[7] C. S. Krishnamoorty, Finite Element Analysis, Tata McGraw-Hill
[8] David V. Hutton, Fundamentals of Finite Element Analysis, McGraw Hill

ISSN: 2277-9655
(I2OR), Publication Impact Factor: 3.785

[9] D. Maity, Computer Analysis of Framed Structures, I. K. International Pvt. Ltd. New DelhiErik G. Thompson, Introduction to the Finite Element Method: Theory, Programming and Applications, John Wiley
[10] H. C. Martin and G. F. Carey, Introduction to Finite Element Analysis - Theory and Application, NewYork, McGraw-Hill
[11] Irving H. Shames, Clive L. Dym, Energy and Finite Element Methods in Structural Mechanics; New Age International