International Journal of Modern Engineering Research (IJMER)
www.ijmer.com Vol. 2, Issue. 5, Sep.-Oct. 2012 pp-4360-4362 ISSN: 2249-6645

Structural Analysis of Partitioned Vertical Column

Nitin V. Titave[1], Dr. Satishchandra V. Joshi[2] ,Vinay Patil[3]

[1] *(Master of Engineering (Mech. Design Engg.) Student of DYPCOE, Akurdi, University of Pune, India*
[2] *(Principal, DYPCOE,Akurdi, University of Pune,India*
[3] *(Department of CAE, Vaftsy CAE, Pune, India*

Abstract: *Research work in the area of Vertical storage column, used in Ammonium Nitrate pyrolysis Process plant, where in required to design a tall vertical pressure vessel having three partitions, each partition is subjected to different pressures and materials. Evaluation of partitions in partitioned pressure vessel has to be done to know performance of vessel under different pressure condition and ensure structural stability of the system*

This paper focuses on Finite Element Analysis of partitioned vertical storage column subjected with given pressure loading condition.

Keywords: *Finite Element Analysis [FEA], structural analysis, partitioned vertical column*

I. Introduction

A pressure vessel is a closed container designed to hold gases or liquids at a pressure substantially different from the ambient pressure. The pressure differential is dangerous and many fatal accidents have occurred in the history of their development and operation. Consequently, their design, manufacture, and operation are regulated by engineering authorities backed up by laws. For these reasons, the definition of a pressure vessel varies from country to country, but involves parameters such as maximum safe operating pressure and temperature. [1]

1.1 Process Information:

Pyrolysis is a process, in which Nitrous Oxide is obtained by thermo chemical decomposition of Ammonium Nitrate at elevated temperatures in the absence of oxygen. Ammonium Nitrate pyrolysis synthesis is an exothermic reaction occurring at 200 deg C. Nitrous Oxide generated has lot of Impurities like Ammonium Nitrate fumes, Nitrogen, Other oxides of Nitrogen & Steam. These impurities are removed by washing or Scrubbing, which is a 3 stage process. In first stage, steam gets condensed by water scrubbing, In second stage, residual traces of ammonium nitrate are removed by caustic scrubbing. Finally in third stage, Ammonia gas generated by caustic scrubbing is removed by Acid Scrubbing.

1.2 Objectives of study:

Objectives of study are as follows:

i. A detailed understanding of function and configuration of pressure vessel.
ii. Modeling of partitioned pressure vessel shell
iii. Finite Element Analysis of model using Ansys.
iv. Strength enhancement in concern areas of base line design using Ansys.
v. Integration of three stages of scrubbing in a single vessel by using partitions

1.3 Types of Non-Linearities:

Nonlinear structural behavior arises from a number of causes, which can be grouped into three principal categories. Contact Nonlinearities are Situations in which contact occurs are common to many different nonlinear applications. Contact forms a distinctive and important subset to the category of changing-status nonlinearities[2]

Geometric Nonlinearities occurs, if a structure experiences large deformations, its changing geometric configuration can cause the structure to respond nonlinearly.[2] Material Nonlinearities, when Nonlinear stress-strain relationships are a common cause of nonlinear structural behavior. Many factors can influence a material's stress-strain properties environmental conditions, and the amount of time that a load is applied. [3]

II. Design Specifications

2.1 Design Specifications:

Three partitions are equispaced and held in place by use of inner shell. Ratio of Inner radius of outer shell and inner radius of Inner shell to be maintained as per 4.5m/2m (Inner shell radius varies between 0.5 m to 2m).

2.2 Material Properties:

The material used for pressure vessel is structural steel (SA 516 Gr 70) and standard properties are:
E= Modulus of elasticity =2.01 e+11(N/ m^2)
u = Poisson's ratio=0.23
Ultimate Tensile strength: 340 MPa
Yield strength: 240 MPa at 27°C temperature
Factor of safety: 1.5
Working Pressure-0.14 Mpa
Analysis performed for two of three compartments are fully occupied to design pressure conditions.
Table 1 shows basic information of overall dimensions of partitioned pressure vessel and the data for analysis
Component thicknesses shown in Table 1 are derived from calculations using ASME 2007 section-VIII, Div-2. [4], [5]

Table1
Dimensions of partitioned vertical column

III. Finite Element Analysis

Ansys 12 is used to carry out Analysis. The steps involved in Finite Element Analysis of partitioned Vertical Column are Modeling, Meshing, Applying boundary conditions & solving problem with static structural analysis and this is discussed in detailed manner in this section.

3.1 Modeling:

Fig 1 shows the 3-D model of Partitioned vertical Column. It is created in CAD software Pro-E, and then exported in IGES format for further analysis in ANSYS workbench.

Sl. No.	Parameters	Units	Value
1	Diameter of Vessel External cylinder (d1)	mm	9000
2	Diameter of Vessel Internal cylinder (d2)	mm	4000
3	Vessel Vertical Height	mm	21000
4	Vessel Skirt Height	mm	3000
5	Vessel Top (Flat)	mm	132
6	Vessel Base (Flat)	mm	132
7	Vessel Shell Thickness	mm	102
8	Vessel Skirt Support Thickness	mm	102
9	Partition Thickness	mm	102
10	Design Temperature	deg. C	27°C
11	Angle between each partition	deg.	120°
12	Vessel External cylinder thickness	mm	102
13	Vessel Internal cylinder thickness	mm	102

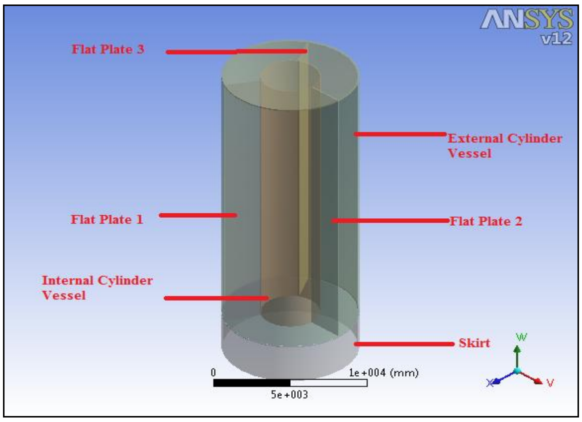

Fig 1: 3-D model of Partitioned vertical column

3.2 Meshing:

Fig.4 shows meshing of model. Assembly model is meshed with 20 node tetrahedron SOLID 186. It is a higher order 3-D 20-node solid element that exhibits quadratic displacement behavior. The element is defined by 20 nodes having three degrees of freedom per node: translations in the nodal x, y, and z directions. The element supports plasticity, hyper elasticity, creep, stress stiffening, large deflection, and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elasto-plastic materials, and fully incompressible hyper elastic materials. [6]

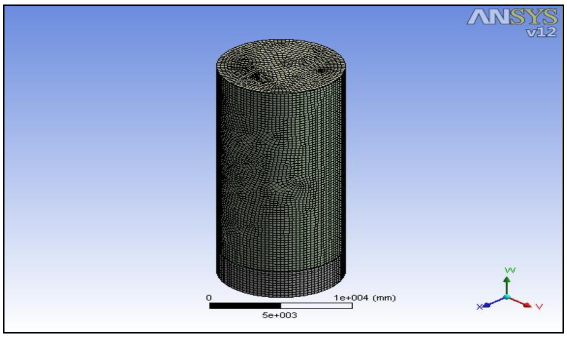

Fig.2: Meshing of Partitioned Vertical Column

3.3 Boundary conditions:

Fig. 3 shows boundary conditions applied to partitioned vertical column, Skirt support is fixed to ground. Internal pressure is acting on vessel wall, partition walls, Flat Top and bottom head and also force due to gravity

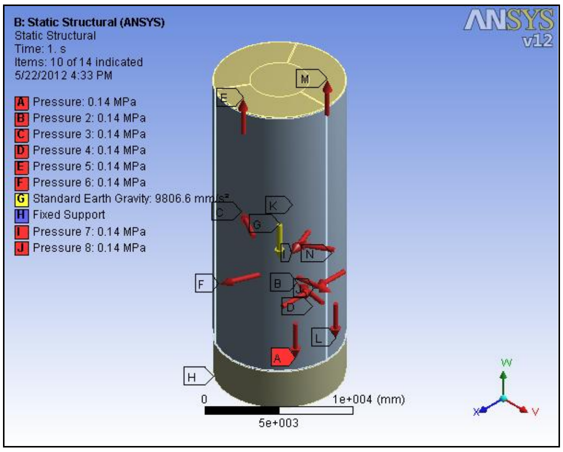

Fig. 3 Boundary Conditions & Constraints

3.4 Analysis Procedure:

Using Ansys workbench, For pressure loading condition of two compartments are fully loaded with designed pressure and one compartment empty, Analysis carried out on single component at a time. Inner cylinder with thickness 102 mm taken for first iteration, with keeping thicknesses of other components constant as per Table 1. Analysis iterations carried out for different element sizes. Plot of No of elements & corresponding maximum von-mises stresses made. In first readings, Von-mises stresses found to be very less as compared to designed allowable stress (Yield strength* Factor of Safety). Then iterations are taken by reducing thickness of inner cylinder until we get safe thickness for inner cylinder. Now keeping Inner cylinder thickness constant, same Analysis procedure carried out for Outer Cylinders, Partitions, Flat top & bottom.

No of Elements	Von-Mises Stress [Mpa]
200267	120.78
264184	106.11
284380	105.27

Fig.4 shows plot of maximum von mises stresses in inner cylinder, Fig. 5 shows maximum von mises stress plot of whole vessel, location of max von mises stress is on partition plate, Fig.6 shows total deformation plot.

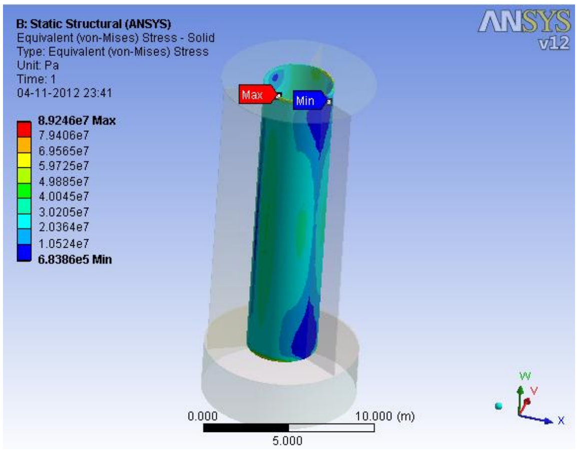

Fig.4: Maximum von mises Stress plot-Inner cylinder

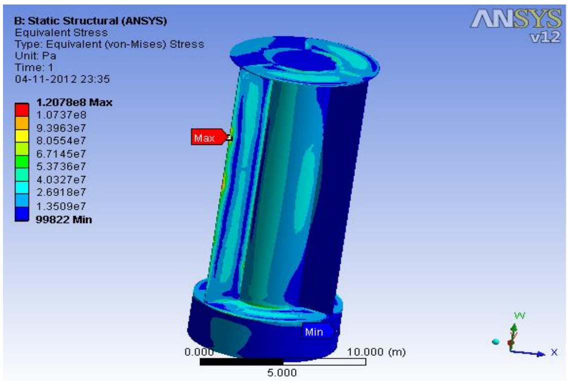

Fig.5: Maximum von mises Stress plot-Partition Plate

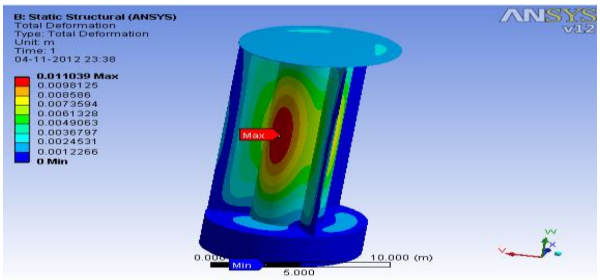

Fig.6: Total Deformation plot (outer shell not shown)

Table 2: Stress Vs. No of Elements

Sr.No.	Dimensions of Components	Von-Mises Stress (MPa)
1	70 mm Vessel Inner Shell thickness	120.78
2	50 mm vessel external shell thickness	91.30
3	80 mm flat partitioned plates	74.00
4	80 mm Top and Bottom covers	66.71

No of Elements Vs Max von mises stresses are shown in Table 2 & Fig. 7 shows plot of number of elements and Von-mises stresses. From graph, it is clear that maximum von-mises stress is approx.upto 120 MPa.

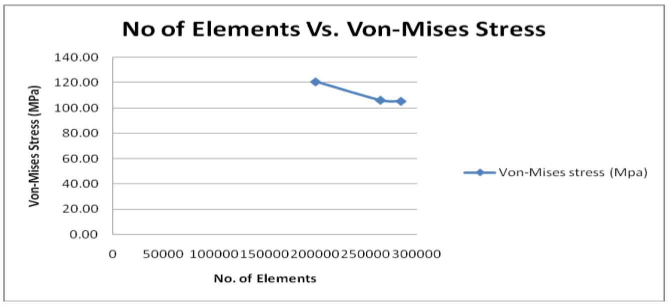

Fig.7: Graph of No of elements Vs Von-Mises stress

IV. RESULTS & DISCUSSIONS:

Results are shown in Table 3:

Table 3: Results of Analysis

V. CONCLUSION

The basic objective was to perform design and analysis of proposed Model of partitioned vertical column. Analytical design of regular pressure vessel is done with reference to ASME 2007 Section VIII, Division-2.

But resultant thicknesses of components seem to be overdesigned. Also calculations of stresses and strains near to end cap not easy to calculate. Finite element analysis is powerful and yet to be simpler approach, which gives accurate results in the field of pressure vessel design. Proposed work is to analyze partition plates critically due to differential pressures, resulting in interference between the vessel and partition and need to analyze in FEA to understand effects on stresses on the Vessel.

VI. ACKNOWLEDGMENT

The Author gratefully thanks to Dr. D.R. Panchagade, PG co-ordinator, Dept. of Mechanical Engg., DYPCOE, Pune for continuous motivation. Authors also gratefully acknowledge the contributions of Vinay Patil of Vaftsy CAE for his guidance for FEA simulation using Ansys Inc. software.

REFERENCES

[1] Stephen Tiernan, Martin Fahy, "Dynamic FEA modelling of ISO tank containers", *Journal of Materials Processing Technology 124 (2002)* 126–132
[2] Proprietary information of POWER GAS INC, PGI USA
[3] Z. Sanal, Nonlinear analysis of pressure vessels: some examples, *International Journal of pressure vessels and piping 77 (2000)* 705-709
[4] Deniss R. Moss, *Pressure Vessel Design manual* 2004 (third edition)
[5] ASME Boiler and Pressure Vessel Code, Section VIII, Divisions 2, 2007, American Society of Mechanical Engineers
[6] Ansys12.0, *Ansys Manual*, ANSYS In