RESEARCH ARTICLE OPEN ACCESS

"Analysis and Weight Optimization of Split Dish Reactor Using Thermo-Structural Coupled FEA"

Yogesh V. Kadam*,S.C.Shilwant** Vinaay Patil***
*,** Mechanical Engineering Dept., Sinhgad academy of engineering, Kondhwa., Pune, India.
***FEA (Finite Element Analysis) Dept., Vaftsy CAE, Pune, India

Abstract—
A vertical split dish reactor with leg supports is modeled using ansys workbench. Thereafter, external loads, such as self –weight, internal pressure and temperature are applied to the model. Pressure and temperature has been continuously a concern which may lead to structural failure if the resulting stresses are severe and excessive. It is a significant study which requires in-depth investigation to understand the structural characteristics. This paper presents and focuses on some Finite Element (FE) analysis of a split dish reactor will be carried out and maximum stresses in the structure will be determined.
Keywords-*FEA;Modal FEA; Non-Linear, Dish Reactor.*

Nomanclatures-
T= Minimum required thickness (in.)
P = Design pressure (psi)
R = Inside radius (in.)
S = Allowable stress (psi)
D = Inside diameter (in.)
E = Weld joint efficiency factor, determined by joint location and degree of examination.
[E=1 for full radiographic examination]

I. INTRODUCTION

Industrial pressure vessels are usually structures with complex geometry containing numerous geometrical discontinuities and are often required to perform under complex loading conditions (internal pressure, external forces, thermal loads, etc.). The design and manufacturing of these products are governed by mandatory national standards, codes and guidelines that ensure high safety performance. Most pressure vessel design codes (e.g. EN13445, BS550, ASME Div III) assume a membrane stress state condition for the determination of the minimum shell thickness and large safety factors at areas of geometric discontinuities such as openings, change of curvatures, nozzle intersections, thickness reduction, etc. It should be noted that large safety factors lead to increasing the material thickness, while safety is not necessarily increased; recall that fracture toughness decreases with increasing thickness, and stress corrosion cracking at components operating in corrosive environments is expected to be higher in thicker parts.

During the last three decades considerable advances have been made in the applications of numerical techniques to analyse pressure vessel problems. Among the numerical procedures, the finite element methods (FEMs) are most frequently used.

In the design/fabrication of pressure vessels, geometric discontinuity (abrupt change in radius of curvature due to misalignment and angular distortion, and/or thickness of the shell) induces additional bending stress which may alter the stress distribution at the regions of the discontinuity. Determination of discontinuity stresses is an important problem. Finite element analysis (FEA) utilizing the commercial software packages (viz., ANSYS, NISA, MARC, etc.) will be more appropriate for shell structures involving elements of arbitrary thickness and curvature to obtain the stress distribution around discontinuities.

In this paper, first, the process and model is explained in a detailed manner. Afterwards, the results of the analysis are presented. Finally, the main conclusions of the investigation are drawn.

II. BRIEF OVERVIEW OF SOME RESEARCH

J.Y. Zheng et al. [1] has done the investigation on bursting pressure of flat steel ribbon wound pressure vessels. The flat steel ribbon wound pressure vessel, invented by Professor Zhu in the People's Republic of China, has shown lots of advantages; namely, flexible design, convenient manufacture, safe use, wide feasibility and easy inspection. The material and manufacturing cost of using the flat ribbon wound technology may be 40% reduced from other methods in use for constructing large pressure vessels. The flat steel ribbon wound pressure vessel may burst either in the circumferential direction or in the longitudinal direction, which depends on the thickness of the inner core and the helical winding angle. By considering the additional strength caused by friction between ribbon layers in the model and ignoring dimensional changes resulting from plastic deformation, the authors deduced equations for the prediction of circumferential and longitudinal bursting pressures.From the comparison between test results and calculated values of longitudinal bursting pressure it can notice that calculation values have a good agreement with test values, maximum relative error is 5.1%, average relative error is 2.5%.It is easy in calculation and convenient to apply in engineering.

T. Aseer Brabinet al. [2] have carried out finite element analysis (FEA) to obtain the elastic stress distribution at cylinder-to-cylinder junction in pressurized shell structures that have applications in space vehicle design. Finite element analysis (FEA) has been carried out on cylindrical pressure vessels having misalignment in a circumferential joint at unfilleted butt joint with equal thickness cylindrical pressure vessel was analyzed with 250 elements and 312 nodes, unfilleted butt joint with unequal thickness cylindrical pressure vessel was analyzed with 350 elements and 416 nodes, filleted butt joint with equal thickness cylindrical pressure vessel was analyzed with 283 elements and 358 nodes. The peak stress values for these configurations obtained from FEA are close to that of test results. The peak stress value is found to reduce due to filleted butt joint and also confirmed through test results.

H. Darijani , R. Naghdabadi et al. [3] was derived an exact elasto-plastic analytical solution for a thick-walled cylindrical vessel made of elastic linear-hardening material By considering the Bauschinger effect and the yield criterion of Tresca. For evaluation purposes, the material behavior was assumed to be a linear strain hardening that obeys Tresca's yield condition with associated flow rule. With the working pressure and geometric dimensions of the vessel, the distribution of the hoop and equivalent stresses are optimized in the way that the distribution of stresses becomes smooth in the vessel wall. Based on two optimizing methods of the hoop and equivalent stresses, the best autofrettage pressure is determined. It shows that this pressure is more than the working pressure and depends on the three following variables: Bauschinger effect, working pressure and geometric dimensions. In the next stage,it determine the wall thickness having the working pressure. For this, two different design criteria namely; optimizing the hoop stress distribution and assuming a suitable percent of yielding in the wall thickness are used. In the last step, for different types of structural materials under different working pressures, a number of different plots are given for the ratio of outer to inner radii and the best autofrettage pressure. It shows that the design of vessels based on the elasto-plastic methods is much more economic than elastic methods. Also, it is seen that for a non-hardening material, the design of vessel is only done for the working pressure less than unit value.

III. BASIC CHEMICAL PROCESS AND OPERATION OF PLANT

German chemist Fritz Haber discovered a process that is still used today. Ammonia was first manufactured using the Haber process on an industrial scale in 1913 in BASF's Oppau plant in Germany. The Haber Process combines nitrogen from the air with hydrogen derived mainly from natural gas (methane) into ammonia. The reaction is reversible and the production of ammonia is exothermic.

$$N_{2(g)} + 3H_{2(g)} \rightleftharpoons 2NH_{3(g)} \quad \Delta H = -92\ kJ\ mol^{-1}$$

Fig (1) shows a flow scheme for the Haber Process.

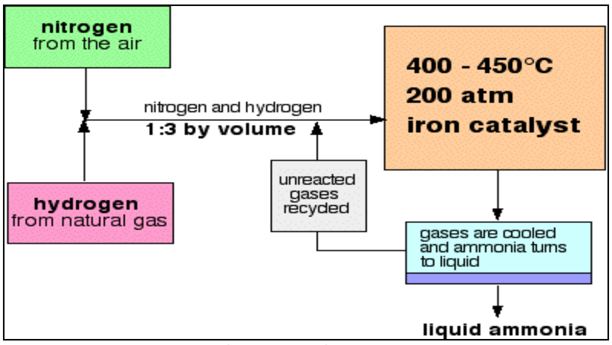

Figure 1:Layout of process of ammonia generation palnt.

By changing the conditions of temperature and pressure alters the position of equilibrium (proportion of nitrogen, hydrogen and ammonia will change). The speed at which equilibrium is reached is made much faster by using an iron catalyst. A reasonable rate of reaction is achieved at 450°c.The gas stream from the reactor is cooled and the ammonia liquefies and can be separated. The unreacted nitrogen and hydrogen gasses are recycled. The table shows the yield of ammonia as a percentage at different temperatures and pressures in the haber process.

Pressure(atm)	100°c	300°c	500°c
25	91.7%	27.4%	2.9%
100	96.7%	52.5%	10.6%
400	99.4%	79.7%	31.9%

The dish wall will take differential pressure of 5MPa. High temperature considerably lowers the yield capacity of the material. Hence while doing FEA, through check is required to see if there is any plastic deformation. Also since the process is reversible one, any alteration in pressure conditions will mean that product will reconvert into reactant causing production loss. Hence dish walls should not deform excessive, that they alter the concentration of reactants and cause the process to reverse. Deformation due to thermal expansion is direct function of distance from heated iron pallets, so it will be our optimization parameter. The thickness of wall will control the plastic deformation hence the thickness will be our another optimization parameter

IV. ANALYTICAL METHOD TO DETERMINE THE THICKNESS OF SHELL

Following is the formula for determining the thickness of cylindrical shell

T = PR / [SE – 0.6P]………….(1)

Following is the formula for determining the thickness of Hemispherical head

T= PR / [2SE – 0.2P]…………(2)

V. THE FEM MODEL AND RESULTS

Finite element analysis (FEA) is one of the most popular engineering analysis methods for Non linear problems. FEA requires a finite element mesh as a geometric input. This mesh can be generated directly from a solid model for the detailed part model designed in a three-dimensional (3D) CAD system. Since the detailed solid model is too complex to analyse efficiently, some simplification with an appropriate idealization process including changing material and reducing mesh size in the FE model is needed to reduce the excessive computation time. The split dish reactor is made of special alloy SP-R4(DNV).

Fig.(3) shows the FEM model of the existing design. It typically is a spherical dish split by a concentric sphere. The existing design is supported on 8 le gs .The material used for FE Analysis is Non Linear. The FEM Model having 6 freedoms: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z-axes.

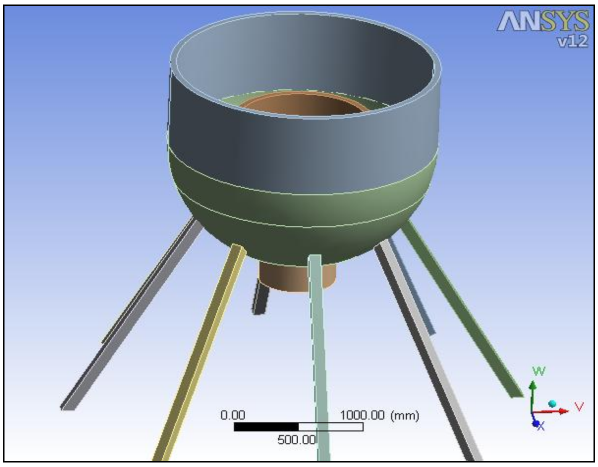

Figure 3: FE Model of Split Dish Reactor.

Material properties applied to the body contains

- Young's Modulus: 201GPa
- Poisson's Ratio: 0.23
- Yield Strength: 550MPa (Room Temperature)
- Allowable Stress: 450MPa
- Ultimate Strength: 650MPa (Room Temperature)

The boundary conditions applied for the body which contains

- The 8 legs are fixed at bottom .
- An internal pressure of 21Mpa was applied on inner face of dish and on outer face of split. Also, an internal pressure of 21.5Mpa was applied on inner face of split. Thus a differential pressure of 0.5Mpa was maintained between dish and split.
- The Conduction temperature of 260°c is applied to inside faces of split, dish and nozzle. Also, a Convection (atmospheric) temperature of 24°c is apllied to outer faces of body including 8legs.
- There is also self weight (g), which was applied as standard gravity in FEA.

At first the FE model of split dish reactor with apllied temperatures as only boundary condition was analysed in FEA as steady state thermal for inducing thermal stresses. Fig(4) shows the FE model of split dish reactor under conduction(500F) and convection(75.2 F) temperatures.

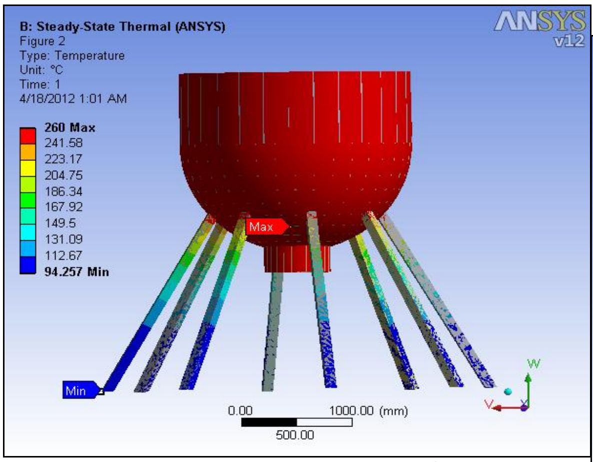

Figure 4: FE Steady- State Thermal Model of Split Dish Reactor.

A different type of meshing sizing is made for different parts of split dish reactor. For split dish reactor, we model and meshed only the middle and lower split dish reactor portion using Hex Dominant Quadrilateral and Triangular elements. Fig(5) shows FE Hexdominant Mesh Model of Split Dish Reactor.

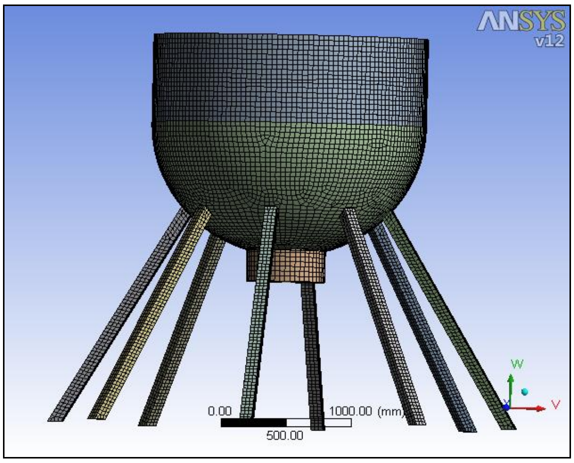

Figure 5: FE Hexdominant Mesh Model of Split Dish Reactor.

In next stage, the solution obtained from thermal analysis is incorporated in setup of static structural analysis to analyze FE model of split dish reactor with applying all remaining boundary conditions except temperature, Fig (6) shows FE model with applied boundary conditions.

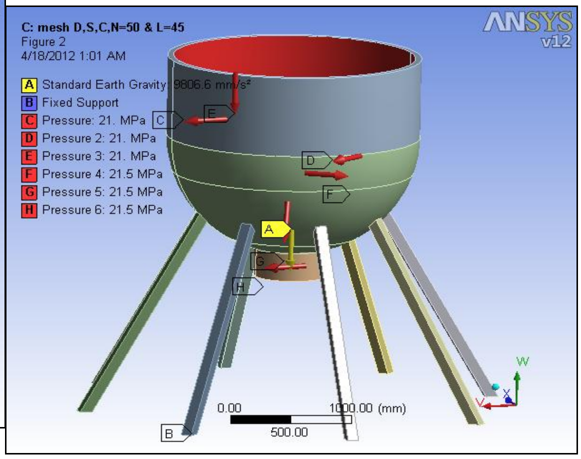

Figure 6: FE Boundry Conditioned Model of Split Dish Reactor.

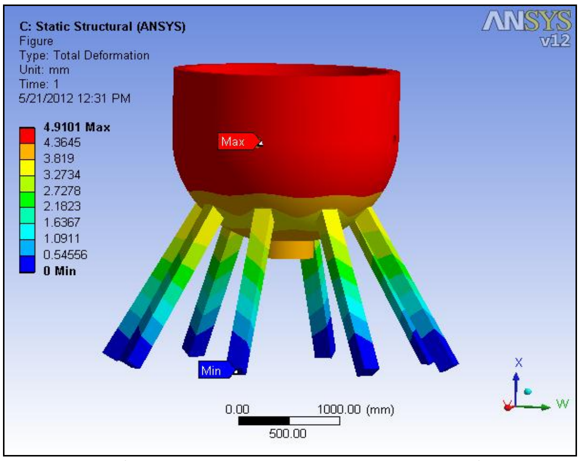

Figure 7: Deformation with Hexdominant mesh of Split Dish Reactor.

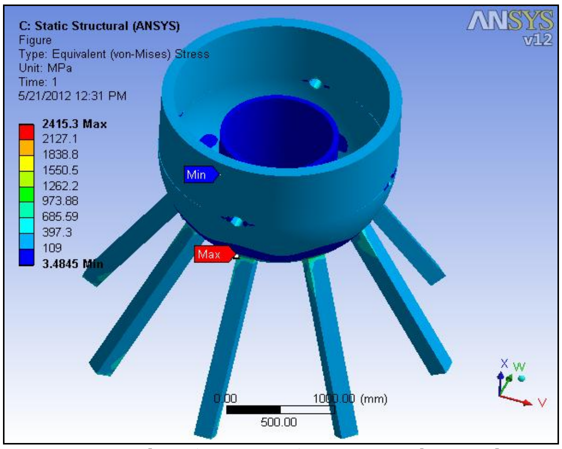

Figure 8: Equivalent (von misses) stresses with Hexdominant mesh of Split Dish Reactor.

Fig(7) and Fig(8) shows deformation and equivalent (von misses) stresses with hex dominant mesh of split dish reactor.

After doing no. of iterations we get different values of maximum stress corresponding deformations maximum load that can be sustain by the model. The results of analysis yields

No. of Nodes=115008; No. of Elements=22442; No. of Steps=5

Sr. No	Part	Thickness (mm)	Mesh Size (mm)	Max. Deformation (mm)	Average Stress Over part (MPa)	Max. Stress (MPa)	Max. Stress at Jnt. Of dish & leg (MPa)
1	Half Cylinder	60	45	4.91	360.32	661.96	2415.3
2	Lower Dish	60	45	4.36	301.21	688.71	
3	Inner Split	50	30	3.27	139.27	236.42	
4	Lower Nozzel	50	30	3.27	169.54	284.28	
5	Legs	110	24.25	3.81	555.49	2415.3	

Result Table

Conclusion

The analysis of split dish reactor brought a number of inadequacies in design. It is necessary that the dish wall will not deform excessive unless it alters the concentration of reactants and cause the process to reverse, results in production loss. So the dish wall and leg will not be so thin.

The optimum thickness of split dish reactor also changes with operating conditions such as teperatures and pressures used. From above analysis, it is found that the maximum stress concentration will be at the joint of leg and dish wall so it is required to optimize the the leg thickness along with the other parts of the model.

Also it is found that the lower nozzle and inner split experiences very less stress than yield stress. So there is chance to minimize the thickness and in turns lowers weight of body.

Also it reduces the distance between dish wall and inner split that in turns reduces the deformation due to thermal expansion.

REFERENCES

[1.] J.Y. Zheng, P. Xu, C. Chen ”*Investigation on bursting pressure of flat steel ribbon wound pressure vessels*” International Journal of Pressure Vessels and Piping 75 (1998) 581–587.

[2.] T. Aseer Brabin, T. Christopher, B. Nageswara Rao,*”*Finite element analysis of cylindrical pressure vessels having a misalignment in a circumferential joint*” International Journal of Pressure Vessels and Piping 87 (2010) 197-201.

[3.] H. Darijani , M.H. Kargarnovin , R. Naghdabadi “*Design of thick-walled cylindrical vessels under internal pressure based on elasto-plastic approach*” Materials and Design 30 (2009) 3537–3544.

[4.] Jaroslav Mackerle ”*Finite elements in the analysis of pressure vessels and piping-a bibliography (1976-1996)*” Int. J. Press-. Vess. & Piping 69 (1996) 279-339

[5.] Z. Sanal “*Nonlinear analysis of pressure vessels: some examples*” International Journal of Pressure Vessels and Piping 77 (2000)705-709.

[6.] A.Th. Diamantoudis, Th. Kermanidis ”*Design by analysis versus design by formula of high strength steel pressure vessels: a comparative study*” International Journal of Pressure Vessels and Piping 82 (2005) 43–50.

[7.] W. Payten, M. Law ”*Estimating the plastic collapse of pressure vessels using plasticity contours*” International Journal of Pressure Vessels and Piping 75 (1998) 529–536.

[8.] Myung Jo Jhung, Youn Won Park, Changheui Jang ”*Pressurized thermal shock analyses of a reactor pressure vessel using critical crack depth diagrams*” International Journal of Pressure Vessels and Piping 76 (1999) 813–823.

[9.] T. Hyakudome, S. Ishibashi, Y. Watanabe, H. Yoshida, S. Tsukioka, T.Aoki” *Application to Pressure Vessels for Underwater Vehicle of Magnesium Alloys*” JAMSTEC, 2-15 Natsushima Yokosuka Kanagawa 237-0061 JAPAN.

[10.] Shinichi Takagawa” *Pressure Test of a Pressure Vessel with Glass Dome and Cylinder of Other Material*” Institute of Industrial Science ,The University of Tokyo.