IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
ISSN(e) : 2278-1684, *ISSN(p) :* 2320–334X, *PP : 44-49*
www.iosrjournals.org

Analysis of STHE to Identify Low Stress Area for Locating Nozzle Opening

M. A. Nalawade, S. H. Dwivedi, K. R. Kapadni, A.V.Waikar, S. H. Gawande, R. N. Yerrawar, U.B. Ubarhande
(Department of Mechanical Engineering, M.E.S. College of Engineering, Pune, India)
(PEM, Alfa Laval (India) Pvt. Ltd, Pune, India)

Abstract : *Conventionally heat exchangers are designed using either ASME code or TEMA standards for the given design conditions. This design ensure the safety, however the stresses at typical locations, such as a Kettle type conical shell and in the vicinity of nozzle reinforcement pads, are not easy to locate and evaluate. Hence, in this work a simplified approach to locate the position of nozzle opening for kettle type STHE is presented. This paper focuses on the analysis of eccentric cone which is welded with the shell on both sides with the nozzle. During analysis it is observed that stress peak is lying in the close vicinity of the nozzle opening. The detailed design, analysis, fabrication and testing work was carried out at Alfa Laval (India) Pvt. Ltd., Pune-12.*

Keywords *- Shell and tube heat exchanger, TEMA, finite element analysis, Nozzle openings*

I. INTRODUCTION

Heat exchangers is a pressure vessel are widely applied in industries such as chemical and petroleum machine-building, nuclear and power engineering, gas, oil and oil-refining industries, aerospace techniques, etc. There are many important components of processing equipment like nozzles or opening are necessary in the pressure vessels to satisfy certain requirements such as inlet or outlet connection. Nozzles are connected to a pressure vessel placed both on the cylindrical shell and the heads of the vessel. Geometrical parameters of nozzle connections may significantly vary even in one pressure vessel and it also cause geometric discontinuity of the vessel wall. Hence a stress concentration is created around the opening and the joints may fail due to these high stresses. Hence a detailed analysis is required. There are many codes and standards available for theoretical calculation but they are not helpful to calculate place of maximum stresses, position of critical component, etc. So, there is need to conduct a detailed finite element analysis of the joints to calculate stresses at the joints both in the vessel & nozzle. ANSYS package is used for analysis. The basics of heat exchanger thermal design is explained in [1] which covers STHE components; classification of STHEs according to construction and according to service, tube Side design, shell side design, including tube layout, baffling, and shell side pressure drop and mean temperature difference. A simplified approach to optimize the design of Shell and Tube Heat Exchanger [STHE] by flow induced vibration analysis [FVA] is presented in [2]. It is necessary to superimpose the stress systems due to internal pressure and various external load components and only then determine the maximum stress intensity [3]. Because of its simple form, convenient manufacture, low cost, and rich application experience a nozzle reinforcement pad structure is considered as important local reinforcement method in pressure vessels [4].The presence of nozzle in the pressure vessel creates a region of increased localized stress in the vicinity of opening[5].It gives the plastic limit load solutions for thin walled branch junctions under internal pressure in plane bending, based on detailed three dimensional (3D) finite element (FE) limit analyses using elastic – perfectly plastic materials [6].To overcome difficulty of mathematical similarity and to accurately represent the weld area at the joint, a smooth transition was inserted at the intersection of the two surfaces [7]. The ability to discretize the regular domains with finite elements makes the method a worthy and important analysis tool for the of boundary, initial, and Eigen value problem arising in various engineering discipline solutions[8].A compressive study of local pressure stresses at the junction of pipe nozzle could be find out[9].

II. PROBLEM STATEMENT AND OBJECTIVE

To design and develop a Kettle Type shell and Tube heat exchanger to meet the functional requirements as per Table1, Table 2 and Table 3 to satisfy company requirement as per ASME code & TEMA standards. The scope was limited to analyze the stress variation at conical shell. The objective of this work is to locate the suitable nozzle position by performing the static structural analysis of heat exchanger.

Table 1: Design Input Data

Design Parameter	Value	Unit
Internal design Pressure of shell	20	Kg/cm^2
Internal design Pressure of shell	15	Kg/cm^2
Design Temperature of Shell	90	˚C
Design Temperature of Tube	160	˚C
Corrosion Allowance	3	Mm

Table 2: Geometrical Input Data

Parameter	Value (mm)
Shell ID	500
Conical Shell Thickness	14
Tube OD	19.05
Tube Thickness	2.769
Tube Length	2000

Table3 Nozzle position

Sr. No.	Model No.	Diameter (mm)	Thickness (mm)
1.	A	44.85	8
2.	B	44.85	8
3.	C	44.85	12
4.	D	72.64	12
5.	E	44.85	12
6.	F	44.85	12
7.	L1,L2,L3,L4	44.85	12

III. DESIGN AND SOLID MODELING

Thorough study of 2-D drawing is performed. With help of manual design and computer aided design the detailed 3-D model of kettle STHE according to customer's requirement is developed. After studying the details of 2D drawing, 3-D parts are created as shown in Fig.1 Tube sheet and Baffle Assembly, Fig.2 Tube bundle Assembly then assembled together into the solid model Kettle STHE Fig.3 is generated with the help of CATIA V5 software.

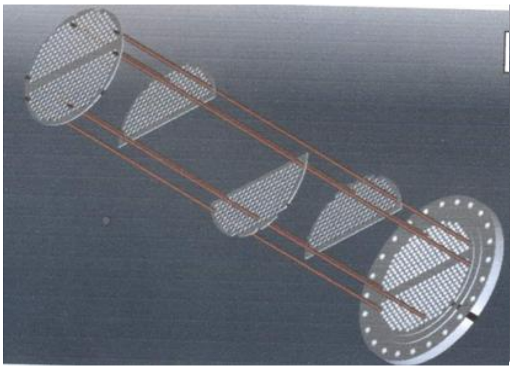

Fig.1 Tube sheet and baffle assembly

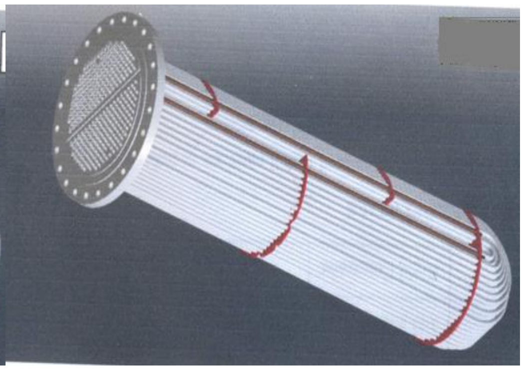

Fig. 2 Tube bundle assembly

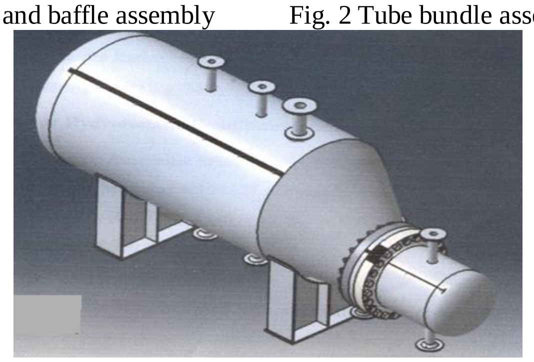

Fig. 3 Kettle STHE assembly

IV. STRUCTURAL ANALYSIS & DISCUSSION

The objective to build a solid model is to mesh that model with nodes and elements. Once the creation of solid model completed, set the element attributes and establishing meshing controls, which turn the ANSYS program to generate the finite elements. For defining the elements attributes, the user has to select the correct element type. This is most important task in finite element analysis because it decides the accuracy and computational time of analysis.

Table 4: Element type used

Element Type No	Element	ANSYS Element	Parts Modeled
1.	3-D Elastic Shell	8 node Shell 93	Shell plate, Cone plate, saddle, nozzle and pad etc.
2.	Rigid Constraint	MPC-184	Rigid Element

In this work, 3-D Elastic Shell (8 node Shell 93) and Rigid Constraint (MPC-184) elements are used as element type as shown in Table 4. It is well suited to modeling irregular meshes. The element may have any work orientation. Here 3-D Elastic Shell (8 node Shell 93) is used for meshing of Shell plate, Cone plate, nozzle and pad etc and MPC -184 is used for meshing of Rigid Element. 3-D Elastic Shell (8 node Shell 93) is used for particularly well suited to model curved shell. The element has six degree of freedom at each node. The element has plasticity, stress stiffening large deflection and large strain capabilities. A MPC-184 element is a Rigid link or beam element can be used to model rigid constraint between two deformable bodies or as rigid component used to transmit forces and moment in engineering application. It also used in application that calls for thermal expansion on an otherwise rigid structure. The direct elimination cannot be used for thermal expansion problem.

4.1 Finite Element Analysis Procedure

The 3D model of conical shell and Nozzle was made on CATIA software. The model is then transferred in IGES format and exported into the Analysis software ANSYS 13.0. The Cone is analyzed in ANSYS in three steps. First is preprocessing which involves modeling, geometric clean up, element property definition and meshing. Next comes, solution which involves imposing boundary conditions and applying loads on the model and then solution runs. Next in sequence comes post processing, which involves analyzing the results plotting different parameters like stress, strain. The type of meshing used for conical shell is MAPPED mesh. The meshed model is shown in Fig.4.

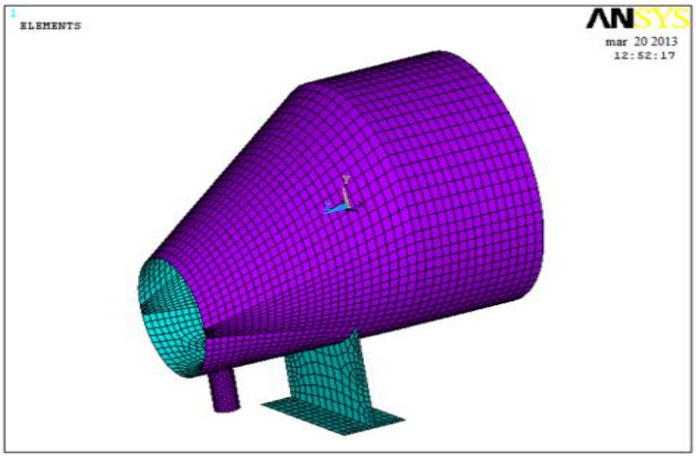

Fig.4: F.E.A. mesh model for design case

4.2 Loading and Boundary Conditions

Load is given as per specification sheet as shown in Table 5. Loading and boundary condition shown in Fig.5

Table 5: Loading Conditions

Parameter	Values	Unit
Internal Design pressure	1.96	N / mm^2
External Design Pressure	0.1013	N / mm^2
Design Temperature	90	˚C
Hydrostatic Pressure	2.54	N / mm^2
Hydrostatic Temperature	Ambient	˚C
Forces on Nozzle	Fx=4500, Fy=3375, Fz=4500	N
Moments on Nozzle	Mx=675000, My=878000, Mz=1013000	N-mm,

4.3 Boundary Conditions

The boundary conditions applied on the model are as shown in the enclosed Fig.6. The model is fixed in all three directions at the end of the cone (Fx=0, Fz=0, Fy =0), and free to move horizontally at other end (Fx=0, Fz=0, Fy free to move). Also model is fixed at saddle.

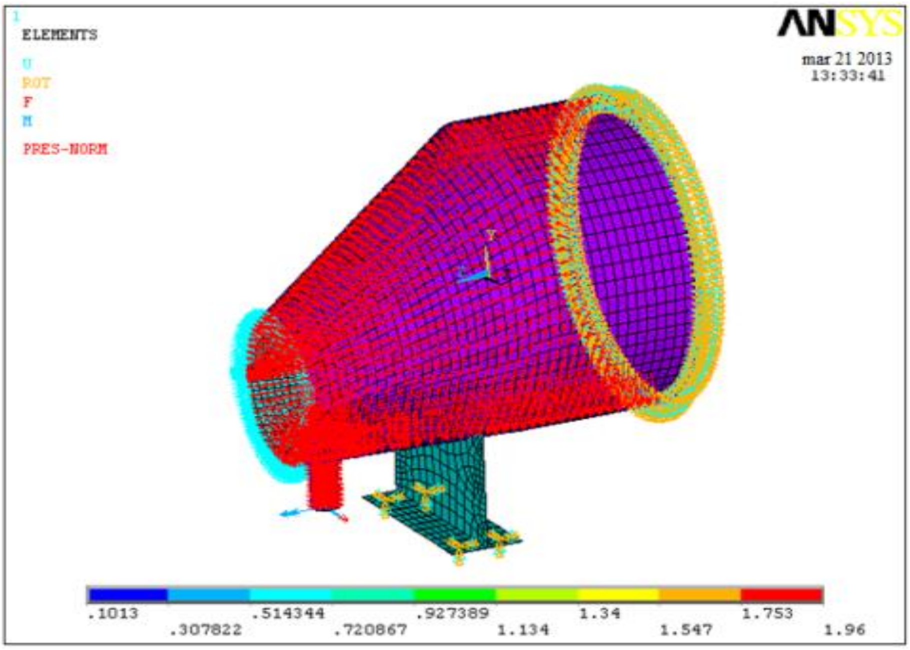

Fig.5 Boundary Conditions

4.4 Conical Shell Analysis

The contour of stress intensity at design conditions is shown in the Fig.6 to Fig.11.

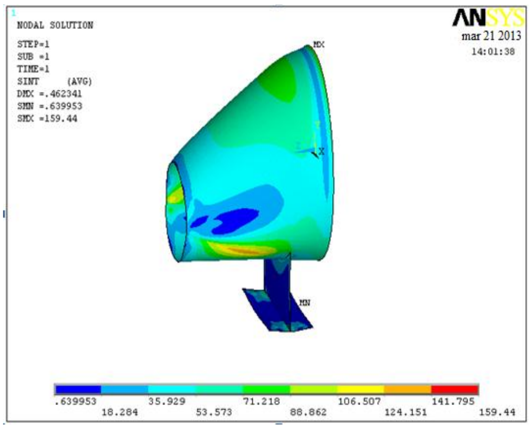

Fig.6 Cone with saddle top layer stress intensity plot at design condition

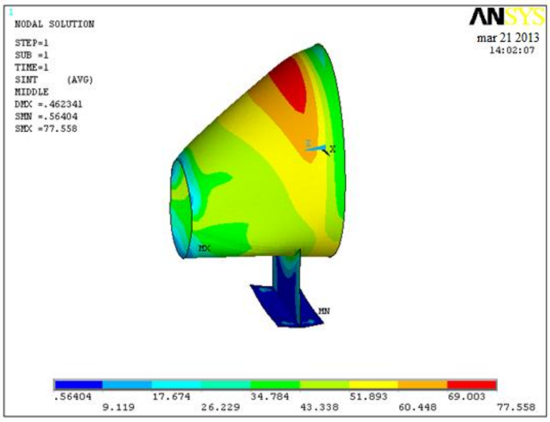

Fig.7 Cone with saddle middle layer stress intensity plot at design condition

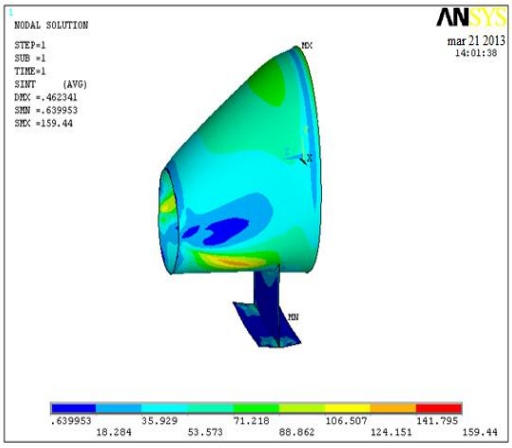

Fig. 8: Cone with saddle bottom layer stress intensity plot at design condition

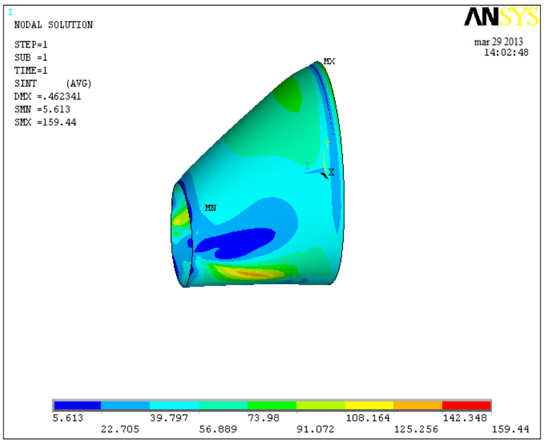

Fig. 9: Cone without saddle top layer stress Intensity plot at design condition

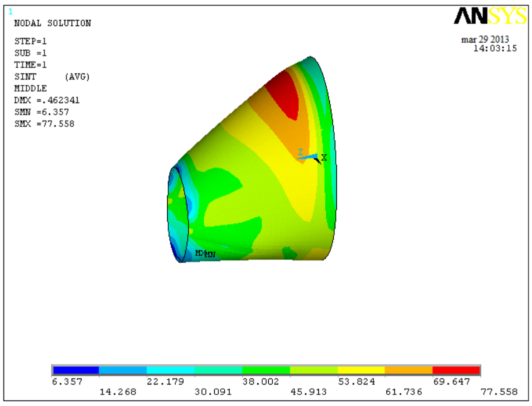

Fig.10 Cone without saddle middle layer stress intensity plot at design condition

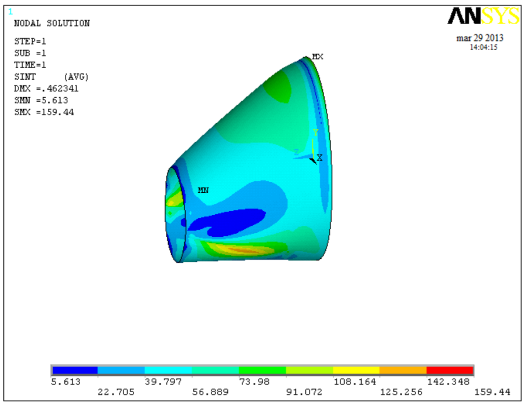

Fig.11Cone without saddle bottom layer stress intensity plot at design condition

Table 6 shows summary of FEA stresses at different layers obtained from Fig. 6, Fig.7, Fig.8, Fig. 9, Fig. 10 and Fig. 11.

Table 6 Summary of FEA Stresses at different layers

Stress at different layers	Cone with saddle	Cone without saddle
Top layer		
Minimum stress	0.6399	5.613
Maximum stress	159.44	159.44
Allowable stress	172.71	172.71
Middle layer		
Minimum stress	0.5640	6.357
Maximum stress	77.558	77.558
Allowable stress	115.14	115.14
Bottom layer		
Minimum stress	0.6399	5.613
Maximum stress	159.44	159.44
Allowable stress	172.71	172.71

From Table 6 it is seen that as the maximum stress intensity for conical shell is less than allowable stress intensity, hence the performed design is safe. From analysis, the minimum stress intensity for cone with saddle is 0.5640N/mm^2 at middle layer that this is safe region to locate nozzle opening. For cone without saddle the minimum value of stress intensity is 5.613 N/mm2 at both top layer and bottom layer, that this is safe region for nozzle.

V. CONCLUSION

The design of STHE i.e. thermal and mechanical design was carried out using the TEMA standard/ASME code both manually and using software. From Table 6 FEA Stresses at different layers showing analysis of minimum and maximum stress intensity in conical shell with saddle, without saddle, it shows that all maximum stresses by FEA analysis are less than the allowable stresses in a conical shell, hence the design is safe. So the nozzle can be placed at the region where the stress intensity is minimum. It was found that locating nozzle as per the input data provided is relevantly satisfies the given condition. The method used is very simple, easy, advance, and the less time consuming as comparing to existing method use in a different Indian industries

REFERENCE

1. R. Mukherjee, Effectively Design Shell-And-Tube Heat Exchangers, *Chemical Engineering Progress*, Feb. 1998.
2. S. H. Gawande, A. A. Keste, L.G. Navale, M. N. Nandgaonkar, V. J. Sonawane and U. B. Ubarhande, "Design Optimization of Shell and Tube Heat Exchanger by Vibration Analysis," *Modern Mechanical Engineering, 1(1), 2011,6-11, doi: 10.4236/mme.2011.11002.*
3. C. J. Dekker, H. J. Boss, Nozzles-on external loads and internal pressure, *International Journal of pressure vessel and piping, 72, 1997, I-18.*
4. J. Fang, Q.H. Tang, Z.F.Sang, A comparative study of usefulness for pad reinforcement in cylindrical vessels under external load on nozzle, *International journal of pressure vessels* & piping, *86 (4), 2009, 273-279.*
5. D. E. Hardenbergh, Stresses at nozzles connections of pressure vessels, SESA Spring Meetting, Washington, 1(5), 1959, 52-158.
6. Y.J.Kim, K.H.Lee, C.Y.Park, Limit loads for thin-walled piping branch junctions under internal pressure and in-plane bending, *International Journal of Pressure* Vessels and Piping, 83 (9), 2006, 645–653.
7. M. Qadir, D. Redekop, SCF analysis of a pressurized vessel–nozzle intersection with wall thinning damage, *International Journal of Pressure Vessels and* Piping, 86 (8), 2009, 541–549.
8. E. Madenci, I.Guven, The Finite Element Method and Applications in Engineering Using Ansys, (Springer publication, USA) 2005, ISBN-10: 0-387-28289-0.
9. J. L. Ha, B. C. Sun, B. Koplik, Local Stress Factors of a Pipe-Nozzle under Internal Pressure, *Nuclear Engineering and Design, 157(1), 1995, 81-91.*