Optimization of Tubesheet Thickness of Triple Concentric Pipe Heat Exchanger using ASME codes

Somankar Vikas[1]
Post Graduate Student
Department of Mechanical Engineering
Veermata Jijabai Technological Institute, Mumbai, India
vikas.somankar61@gmail.com

Tembhare G.U.[2]
Assistant Professor
Department of Mechanical Engineering
Veermata Jijabai Technological Institute, Mumbai, India
gutembhare@vjti.org.in

Patil V.G.[3]
CAE Consultant VAFTSY CAE
Pune, India
vinaay.patil@vaftsycae.com

Abstract— Heat exchanger is the device used to transfer the heat from hot fluid to cold fluid efficiently. It is made up of various parts and every part has issues regarding its safety. This paper is regarding the tubesheet safety. In this paper, thickness of the tubesheet is optimized for different pressure and thermal loading conditions. Due to the high pressure and high temperature of fluid, tubesheet expands because of which the shell of heat exchanger also expands which generates high stresses and deformations in heat exchanger. To avoid this, analysis of effects of pressure and temperature on tubesheet is necessary. This paper has the objective of analyzing the temperature and stress variation in tubesheet and optimizing its thickness.

Keywords- *Heat exchanger, Tubesheet, FEA, ASME, thermal analysis, structural analysis.*

*****

I. INTRODUCTION

1.1 Heat Exchanger

Heat exchanger is the device which we can see in almost every industry. Heat exchangers are used to transfer the heat from hot fluid to cold fluid. Generally the heat exchangers are used in chemical plants, air conditioners, space heating, refrigeration, power plants, refineries, natural gas processing plants, etc. Because of the various complicated parts and their operations, heat exchangers face many issues regarding their safety. Due to these issues the heat exchangers does not work properly or sometimes it doesn't work at all. The high working pressure and temperature heat exchangers face many problems like distortion of tubes, thermal expansion of tubesheets, etc.

This paper deals with the problem of thermal expansion of tubesheet. Many times because of the high pressure and temperature, tubesheets are heated to very high temperature which causes the expansion of tubesheet in radial direction. This expansion of tubesheet results in high stresses in tubesheet and shell.

1.2 Objective of Project

In many industries, high temperature and high pressure fluids are used. In this project the material used is SA 516 Grade 70 steel. The maximum operating temperature is taken as 250°C and pressure of the fluid is taken as 2 bar in a triple concentric pipe heat exchanger. Because of the high operating temperature, the tubesheet expands which results expansion of shell as well. That causes overall deformation of heat exchanger. Therefore the followings are the objectives of Paper.

- To draw the model in ANSYS 15 software.
- To mesh it to get fine elements at different parts.
- To optimize the thickness of tubesheet from different cases in ANSYS 15.
- To analyze the stresses generated at the shell and tubesheet in ANSYS 15.

II. METHODOLOGY

The project was done on the case study given by the CAE consultant VAFTSY CAE, Pune, India. There are different types of heat exchangers, but this work is done on triple concentric pipe heat exchanger. The heat exchanger is of fixed tube type. Company gave input parameters of model. Design calculations of all parameters, except given parameters, were done by ASME Section VIII codes. Model of the project was made in ANSYS Workbench software. Final model in a sectional view is shown below. Parts in yellow are tubesheets which separates outer and middle sections.

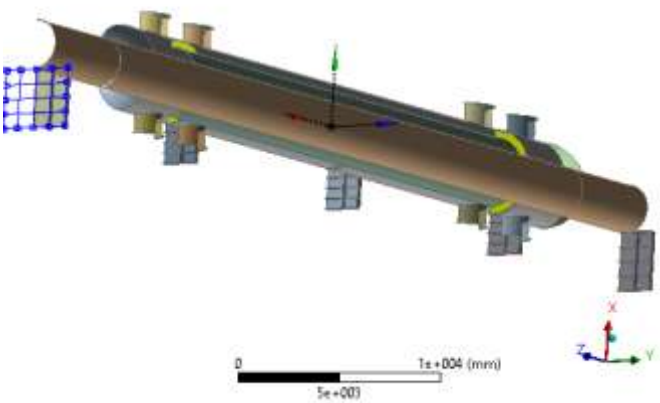

Fig 1: Sectional view of model

III. FLOWCHART OF PROJECT

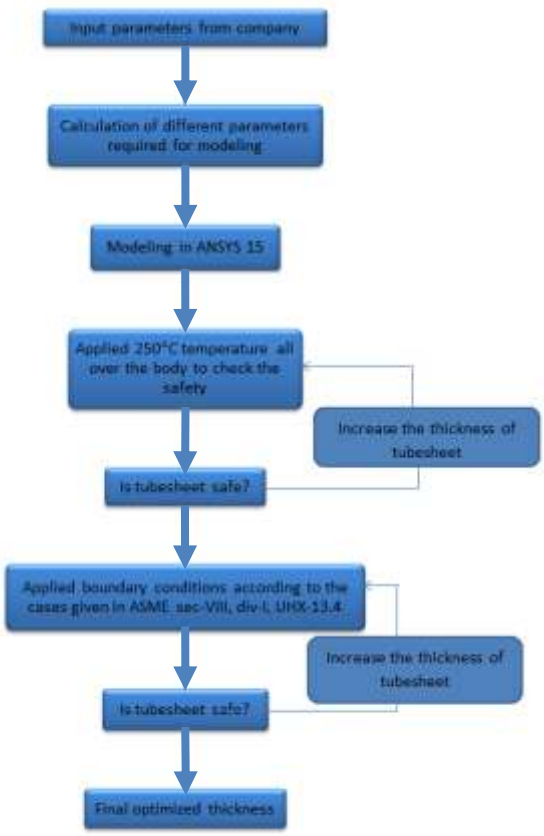

Fig 2: Flowchart

IV. LITERATURE SURVEY

Amey Shirodkar and Sangita Bansode[1] concluded in their research that due to high temperature and pressure, large stresses are generated at shell and tubesheet junction of heat exchanger. Those stresses were validated with Software (PVElite, ANSYS) and Mathematical (ASME Code, Section VIII, Division I) method. They achieved optimized thickness of tubesheet and the stresses produced in the optimize case were also validated with the PVEite and ANSYS software. At the end they concluded that thickness of tubesheet achieved was economical as well as it ensured the safety of heat exchanger.

R. D. Patil, Dr. Bimlesh Kumar[2] while comparing the experimental results with finite element analysis, observed that the percentage variations of experimental values with FEM were well below 10%. They attributed the difference in the experimental and SCF values by FEM to the following reasons:
a) In spite of extreme care taken in manufacturing of models, small amount of machining stresses are likely to be developed in the model, changing the fringe pattern in the model.
b) Fringe order was measured by comparison of fringe colour with colour chart or with colour table. Therefore there is a possibility of misinterpreting the fringe order.
c) There was great amount of friction in the pivots of the loading frame.
They finally concluded that due to limitations of the loading frame, only rectangular plate models that too in uniaxial tension could be tested.

Kalpesh D. Shirode, Dr. S. B. Rane, Mr. Yashwant Naik[3] designed heat exchanger as per ASME Sect. VIII Div. 1 and TEMA design procedure and then analysed it using ANSYS. They found that The factor of safety used for designing the tube sheet in both standards were more which increased dimensions of the heat exchanger components. They concluded that the tube sheet thickness of given heat exchanger is safe according to the codes and analysis. From the result it was observed that design procedure for calculating tubesheet thickness as per TEMA and ASME standards gives tubesheet thickness of 75mm and 60mm respectively. But, FEA gives more optimised tube sheet thickness of 45mm. Based on above it was concluded that FE method for the tube sheet analysis saves 25% of material compare to TEMA design and 20% material compare to ASME design which further reduces the manufacturing cost and time. The induced stresses in the tubesheet for calculated thickness of tube sheet by both codes and analysis were below allowable limits which is acceptable. They finally concluded that from design point of view ASME design method is more realistic than the TEMA methodology.

Ravivarma R., Azhagiri Pon[4] performed various analysis required to assess integrity of Tubesheet. Those analyses were done for operating pressure loads and thermal analysis together with mechanical loads. The analysis was done in two parts; first one was linear Static analysis of conventional equivalent modulus of elasticity & Poisson's ratio method, which was recommended by ASME Sec. VIII, Division-1. Second was a new and realistic approach of linear Static analysis by considering the perforations of tube holes in the tubesheet with pressure acting at inside tubes. This linear Static Analysis was done in ANSYS-14. Based on the results obtained from two different approaches, they concluded that for design of tubesheet for any heat exchanger, analysis done by realistic approach method is better for the selection of optimum required material.

R. D. Patil & Dr. Bimlesh Kumar[5] while comparing the results with finite element analysis found that the percentage variation of experimental values with FEM were about 10-15% and said that this can be attributed to following points.
a) The load applied to the model may not be uniformly distributed over entire cross section of model. Some bending may result due to misalignment of upper and lower clamping plates of the loading frame.
b) In spite of extreme care being taken in manufacturing of models, some small amount of machining stress are developed in the model, which changes the fringe pattern in the model.

V. PROJECT APPROACH

5.1 Input parameters:

Model of the concentric pipe heat exchanger is designed on the basis of input parameters given below in table 1 and other parameters required for modeling were calculated

as per ASME Sec-VIII, div-I. The model is the designed in ANSYS.

Sr.No.	Parameter Description	Notations	Given value
1	Internal Pressure	P	0.2 MPa
2	External Pressure	P_o	1 Atm
3	Process Volume	V_p	300 m^3
4	Expected Stagnant Volume	V_s	30.5 m^3
5	Buffer Volume Requirement	V_b	23.5 m^3
6	Vessel Radius	R	2.1 m
7	Nozzle Diameter	D_n	0.2 m
8	Number of Nozzle	n	8
9	Support Height	h	2 m
10	Heat Transfer Coefficient	h_c	11.3 w/m^2k
11	Material	-	SA 516 Grade 70

Table 1: Input parameters

Pressure	0.195 MPa
Thickness	18 mm
Inside diameter	4200 mm
Temperature	250°C
Material	SA-516 Grade 70

Table 2: Shell (outer pipe section) data

Pressure	0.2 Mpa
Thickness	12 mm
Inside diameter	3442 mm
Temperature	250°C
Material	SA-516 Grade 70

Table 3: Middle pipe section data

Pressure	0.195 Mpa
Thickness	12 mm
Inside diameter	2437 mm
Temperature	250°C
Material	SA-516 Grade 70

Table 4: Inner pipe section data

As per ASME SECTION-VIII, DIV-I, UG-23(e)

$$S_{PS}=3S$$

where,

S = Allowable stress of tubesheet material at design temperature

S_{ps}=Allowable primary plus secondary stress at design temperature (Maximum allowable design stress).

For SA-516 Grade 70,

S=138 MPa

S_{PS}=3×138= 414 MPa

5.2 Meshed model:

After completion of model it was meshed in ANSYS which is shown in fig 2.

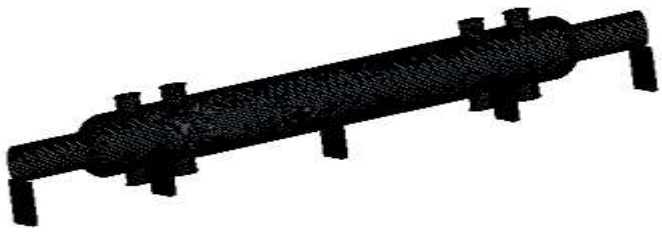

Fig 3: Complete model meshed

Model of the tubesheet in meshed form is shown below. As the tubesheet is inside shell, the shell and some other parts are hidden.

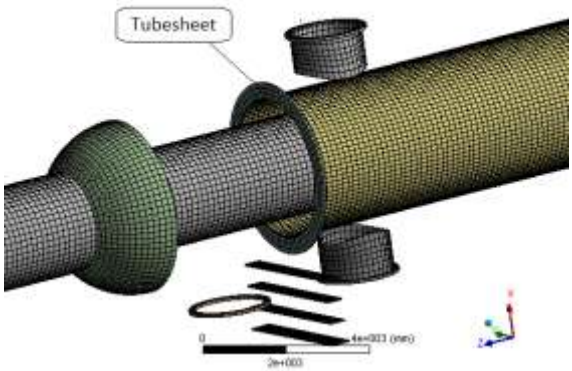

Fig 4: Tubesheet meshed.

5.3 Thermal loading:

In this the thickness of tubesheet was taken as 40mm. 250°C temperature was applied on all the parts and then stress in tubesheet was calculated. It was fond that the stress in tubesheet was above the allowable stress. Then thickness of tubesheet was increased by 10mm steps and stress in each case was compared with allowable stress which is shown in table 2 below.

Tubesheet thickness(mm)	Stress obtained(MPa)	Max. allowable design stress(MPa)
40	454.36	414
50	415.20	414
55	383.52	414
60	353.01	414
70	292.90	414
80	242.96	414

Table 5: Stresses at different thickness of tubesheet.

From the table it is clear that stress obtained at 55mm thickness is below the maximum allowable stress. Hence 55mm thickness was finalized for further analysis.

5.4 Loading cases suggested in ASME Sec-VIII, Div-1, UHX-13.4:

The various loading conditions to be considered shall include the normal operating conditions, the start-up conditions, the shutdown conditions, and the upset conditions, which may govern the design of the main components of the heat exchanger (i.e., tubesheets, tubes, shell and channel).

Loading case	Pressure and thermal condition considered
Case 1	Inner pipe side pressure acting only
Case 2	Middle pipe side pressure acting only
Case 3	Outer pipe side pressure acting only
Case 4	Inner pipe and middle pipe side pressure acting simultaneously
Case 5	Middle pipe and outer pipe side pressure acting simultaneously
Case 6	Inner pipe and outer pipe side pressure acting simultaneously
Case 7	Inner pipe, middle pipe and outer pipe side pressure acting simultaneously
Case 8 – Case 14	Above individual cases with differential thermal expansion

Table 6: Loading cases

5.5 Steps to be followed in ANSYS software

Step 1: Making 3-D model of the heat exchanger in ANSYS workbench 15.0.
Step 2: Considering solving condition as steady state thermal, engineering data like thermal conductivity, Young's modulus, Poisson's ratio, density etc. were applied.
Step 3: The engineering data and geometry in steady state thermal analysis is transferred to static structural.
Step 4: In static structural, model was meshed by using concept of Hex dominant method, mapped face mesh and body sizing.
Step 5: Boundary conditions (i.e. pressure and temperatures) according to different cases were applied to heat exchanger surfaces.
Step 6: Saddle of the heat exchanger was fixed.
Step 7: Model was solved for the applied loadings.
Step 8: Equivalent stresses and linearized stresses were noted down.

5.6 Results obtained in ANSYS

After applying the pressure and temperature accordingly, the model was solved. Stresses obtained in case-1 to case-7 are shown in figures below and stresses in remaining cases i.e. case-8 to case-14 are given in the table.

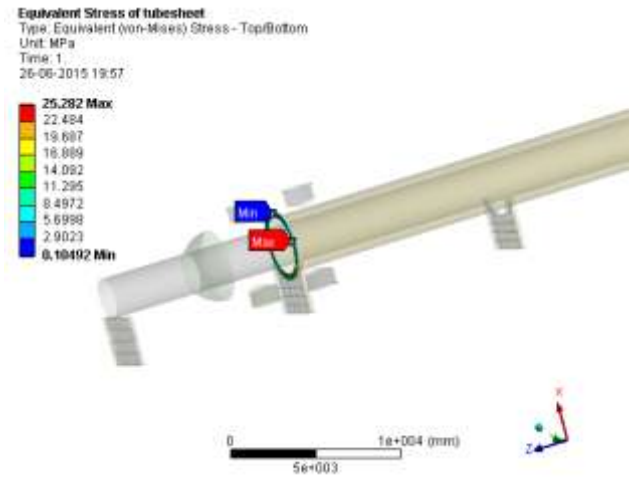

Fig 5: Case 1 stresses in tubesheet

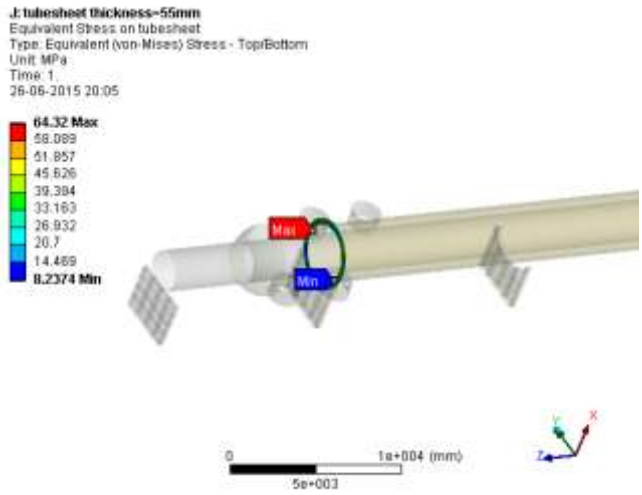

Fig 6: Case 2 stresses in tubesheet

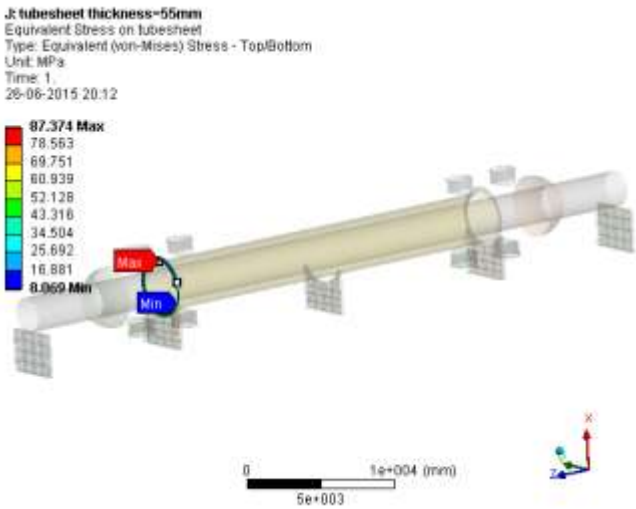

Fig 7: Case 3 stresses on tubesheet

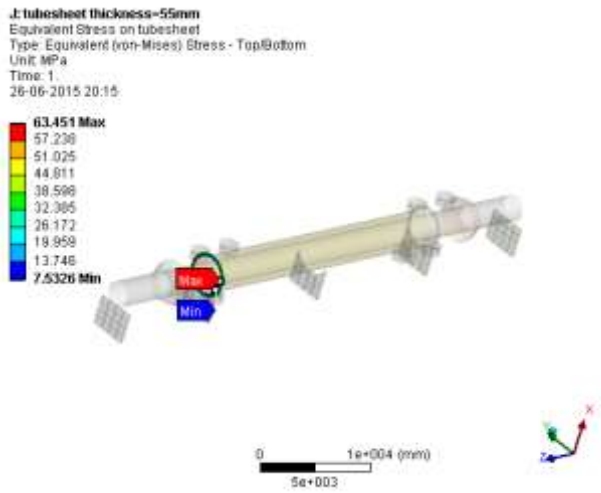

Fig 8: Case 4 stresses on tubesheet

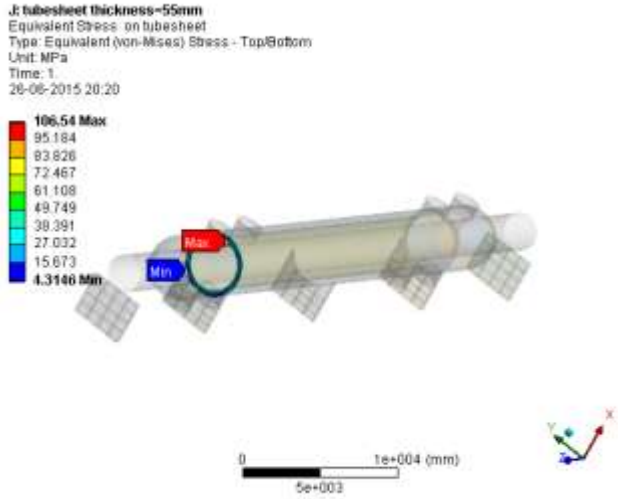

Fig 9: Case 5 stresses in tubesheet

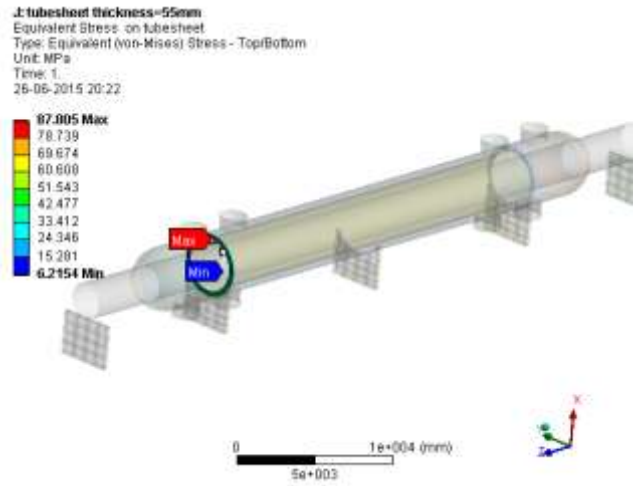

Fig 10: Case 6 stresses in tubesheet

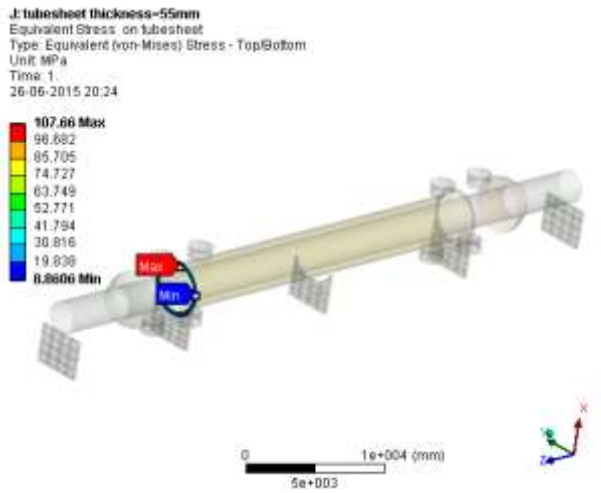

Fig 11: Case 7 stresses in tubesheet

Stresses generated in tubesheet in all 14 cases are given in table below.

Case	Stress obtained(MPa)	Max. allowable design stress(MPa)
Case 1	25.282	414
Case 2	64.32	414
Case 3	87.374	414
Case 4	63.451	414
Case 5	106.54	414
Case 6	87.805	414
Case 7	107.66	414
Case 8	97.463	414
Case 9	116.22	414
Case 10	79.832	414
Case 11	114.33	414
Case 12	97.77	414
Case 13	82.994	414
Case 14	93.961	414

Table 7: Stresses in tubesheet obtained in all cases

VI. CONCLUSION

Due to high pressure and temperature, large amount of stresses are generated in tubesheet. Firstly the thickness of the tubesheet was taken as 40 mm and thermal loading were applied. In thermal loading analysis, 40 mm thick tubesheet was failed. Hence after few more analysis thickness of tubesheet was finalized as 55 mm. After this the tubesheet was to be checked under the loading cases suggested by ASME. Stresses in each case were obtained in ANSYS software. Obtained stresses were compared to allowable stresses and were checked for safety. From the analysis it was found that the stresses generated in tubesheet were well below the safety limit i.e. maximum allowable stress.

Optimized thickness of tubesheet was achieved by using ANSYS software, which is 55mm. So is concluded that the tubesheet of 55mm thickness is safe and economical for the given project.

REFERENCES

[1] Amey Shirodkar and Sangita Bansode, "Optimization of Tubesheet Thickness of Shell and Tube Heat Exchanger", International Journal of Innovative Science, Engineering & Technology, Vol. 1 Issue 6, August 2014.

[2] R. D. Patil, Dr. Bimlesh Kumar, "Stress Analysis of Perforated Tube Sheet Used for Pressure Vessel having Square Pitch Hole Pattern", International Journal of Innovative Research in Science, Engineering and Technology, Vol. 4, Issue 1, January 2015.

[3] Kalpesh D. Shirode, Dr. S. B. Rane, Mr. Yashwant Naik, "comparison of design and analysis of tube sheet thickness by using UHX code of ASME and TEMA standard." International Journal of Mechanical Engineering And Technology, Volume 4, Issue 4, July - August (2013), pp. 105-117.

[4] Ravivarma.R, Azhagiri. Pon, "Finite Element Analysis of a Tubesheet with considering effective geometry properties through design methodology validated by Experiment", International Journal of Computational Engineering Research (IJCER), Vol 04, Issue 4, April – 2014.

[5] R. D. Patil, Dr. Bimlesh Kumar, "An Approach to Finite Element Analysis of Boiler Tube-Sheet", American Journal of Engineering Research (AJER), Volume-02, Issue-08, pp-08-11.

[6] ASME, ASME Boiler & Pressure Vessel Code Section VIII Division 2, New York, American Society of Mechanical Engineers, 1998. (Cited in Zengliang Gao., etal, 2005)