IJISET - International Journal of Innovative Science, Engineering & Technology, Vol. 1 Issue 6, August 2014.
www.ijiset.com
ISSN 2348 – 7968

Optimization of Tubesheet Thickness of Shell and Tube Heat Exchanger

Amey Shirodkar[1] and Sangita Bansode[2]

[1] Mechanical Department, K.J.Somaiya College of Engineering, Mumbai University
Vidyavihar, Maharashtra, India

[2] Mechanical Department, K.J.Somaiya College of Engineering, Mumbai University
Vidyavihar, Maharashtra, India

Abstract

A heat exchanger is a device built for efficient heat transfer from one medium to another. Many a times some issues occurred in the heat exchanger. Out of which this paper is concerned with the thermo-mechanical issue that is thermal expansion of tubesheet due to high temperature. It is necessary to make a optimize design which is safe, economical and accurate. Due to high temperature and high pressure fluids tubesheet of heat exchanger expands which results expansion of shell which causes deformation of heat exchanger. To avoid this deformation, analysis of effect of temperature variation and associated stresses in the tubesheet is necessary. Objective of this paper is to analyse the temperature variation at the junction of shell to tubesheet junction in shell and tube heat exchanger and optimization of tubesheet thickness.

Keywords: *Heat Exchanger, Thermal Analysis, Structural Deformation, Optimization*

1. Introduction

1.1 Heat exchanger

Heat exchanger is one of the equipment found in all industry. Heat exchangers are components that allow the transfer of heat from one fluid to another fluid. Heat exchangers are widely used in space heating, refrigeration, air conditioning, power plants, chemical plants, petrochemical plants, petroleum refineries, natural gas processing, and sewage treatment. Heat exchangers can have a range of problems which can cause them to perform poorly or stop working all together. Due to a high pressure and temperature many problems occurred in heat exchanger like shrinkage of tubes, vibration issues, distortion of tubes due to high flow rate, thermal expansion of tubesheet due to high temperature. Out of which this work is based on the problem regarding thermal expansion of tubesheet. Sometime high temperature fluid is used in heat exchanger due to which thermal expansion of tubesheet occurs that leads to cracks on surface of heat exchanger. In this project radial differential thermal expansion occurred.

1.2 Objective of Paper

In most of the refineries high temperature and high pressure fluids are used. In this project the material used is alloy of chromium and nickel also high temperature (698 ^{0}C) and high pressure (8 bar) fluid is used in shell and tube heat exchanger, due to high temperature tubesheet of heat exchanger expands which results expansion of shell and tubesheet. That causes deformation of heat exchanger. Therefore the following is the objective of Paper.

- The validation for different stresses in PVElite (software method) and ASME code (mathematical method).
- To obtained the optimized thickness of tubesheet from different cases in PVElite.
- To separate the model into different parts to get the fine meshing at the high stress location.
- To analyze the stresses produced at the shell, tubesheet and channel in ANSYS 14.

2. Methodology

The work is done on the case study given by Aker Solutions Pvt Ltd., Kanjurmarg, Mumbai. This work is based on the fixed tubesheet shell and tube heat exchanger. Company gave the details of current shell and tube heat exchanger. The flow chart for methodology is given in fig. 2.1. This methodology shows the flow of project. The model of fixed tubesheet heat exchanger is made in PVElite software. PVElite is customized software for vessels like heat exchangers, boilers, tanks etc. In PVElite the model of component like shell, channel, tubesheet etc. are already available. The result of stresses at the tubesheet, shell and channel are calculated from this software. Thus results are then validated with ASME code calculation and by ANSYS software for general case. The 3 D model of shell and tube heat exchanger is designed in Ansys workbench14.0 software. Then PVElite software is used to obtain the stresses for different thickness. This stresses are

ISSN 2348 – 7968

compared with the allowable stress. If it is safe then for different thickness of tubesheet, shell and channel the table is prepared from which optimum case is selected for validation. This optimized case is validated in PVElite and ANSYS software. If results are compared then this case is consider as final optimum case else consider second case for validation.

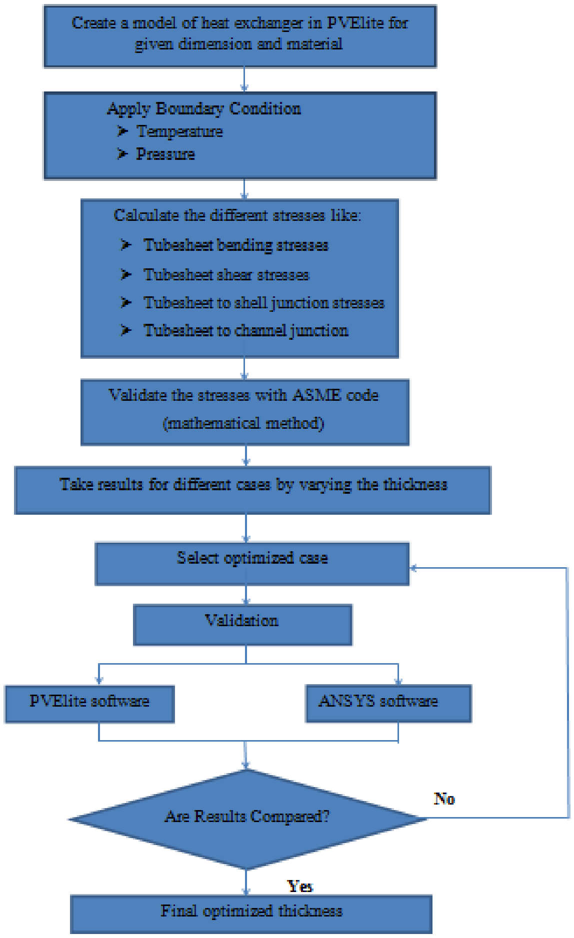


Fig.2.1: Flow Chart for methodology

3. Project Approach

Softwares are used to generate the stresses on the structure. PVElite is one of the customized software for vessels like heat exchanger, boiler etc. to calculate stresses. This stresses are then validate with the results obtained from the mathematical calculation. i.e. ASME code, Section VIII, Division I. This results are also validate with the other software i.e. ANSYS.

3.1 Software Approach (PVElite Software)

PVElite is customized software for vessels like heat exchangers, boilers, tanks etc. The model of fixed tubesheet heat exchanger is made in PVElite software. The result of stresses at the tubesheet, shell and channel are calculated from this software. Fig. 3.1 shows the model made in PVElite software. PVElite software generates the results in the form of different stresses as output. The stresses generated are tubesheet bending stresses, tubesheet shear stresses, shell to tubesheet junction, channel to tubesheet junction and axial membrane stress in shell

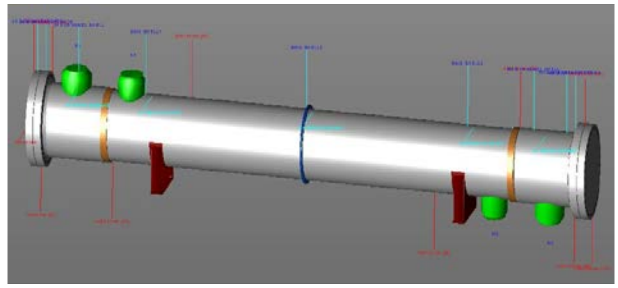


Fig. 3.1 Model in PVElite Software

3.2 Mathematical Approach (ASME Code)

It is necessary to do mathematical calculation to obtained results because the mathematical calculations are the base of any results. In American Standard for Mechanical Engineering (ASME) code section VIII, Div. I the rules for constructions of pressure vessels are given. In that UHX-13 is the procedure for fixed tubesheet shell and tube heat exchanger. By using this ASME code (mathematical method) the different stresses generated at STHE are calculated and validated with PVElite (software method).

3.2.1 Engineering Data

Table 3.1, 3.2, 3.3 and 3.4 shows the engineering data of shell, channel, tubesheet and tube. It shows its pressure, diameter, thickness, material, temperature respectively.

Table 3.1: Shell Data

Shell Data		
Pressure	P_s	0.8 mpa
Thickness	t_s	36 mm
Inside Dia.	D_s	910 mm
Temperature	T_s	654^0C
Material	SA-240 304H	

Table 3.2: Channel Data

Channel Data		
Pressure	P_t	0.8 mpa
Thickness	t_c	36 mm
Inside Dia.	D_c	910 mm
Temperature	T_c	698^0C
Material	SA-240 304H	

ISSN 2348 – 7968

Table 3.3 : Tubesheet Data

Tubesheet Data		
Tubesheet Type	Fixed	Conf. A
Thickness	h	250 mm
Outside Dia.	A	982 mm
Area Of The Untubed Lanes	AL	0 mm^2
Material	SA-213 TP304H	

Table 3.4 : Tube Data

Tube Data		
No. Of Tube Holes	N_t	736
Tube wall thickness	e_t	1.24 mm
Tube outside Dia.	D	19.05 mm
Tube Pitch	P	23.812 mm
Material	SA-213 TP304H	

3.2.2 Tubesheet Diagram

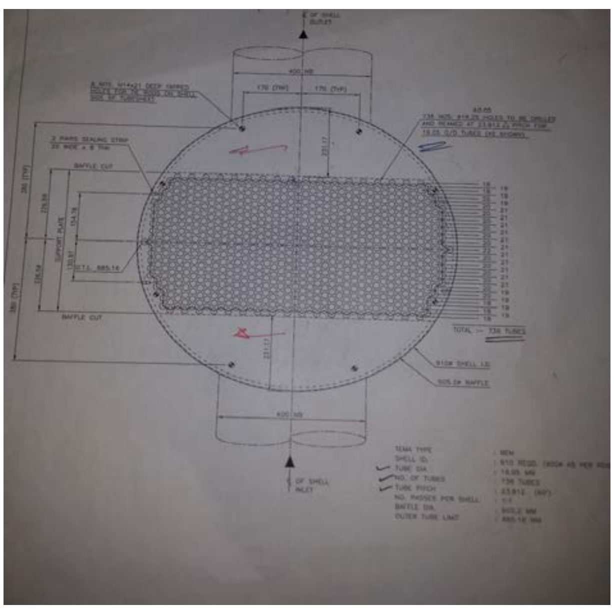


Fig. 3.5 Tubesheet Design

The fig. 3.5 shows the No Tubes in Window (NTW) layout. This NTW layout is used in this project for tubesheet. In this layout there are no tube holes in upper and lower portion of the tubesheet. This layout is used according to the requirement of space in heat exchanger. The dimensions are given in this figure. Also the required parameters are mentioned. The outer tube limit is also given.

3.2.3 Different Steps of UHX-13 of ASME Code to Design Tubesheet

Some of the important steps of UHX-13 are given below.

The Tubesheet Bending Stress [Sigma]:
= (1.5 * Fm / mu*) * (2 * ao/(h - h'g))2 * Pe

The Tubesheet Average Shear Stress [Tau]:
= (1 / (2*mu)) * (ao/h) * Pe

The Shell Bending Stress due to Joint Interaction:
= 6*ks/ts1^2 {betas[deltaS*Ps + as^2*PstarS/(Esb*ts1)] + 6(1 - nu*^{2})/(E*)(ao/h)^3(1 + h*betas/2)[Pe(Zv + Zm*Q1) + 2/ao^2 * Zm * Q2]}

Shell Stress Summation
|Sigmasm| + |Sigmasb| =<SPSsb

Channel Bending Stress due to Joint Interaction:
= 6kc/tc^2{betac[deltaC * Pt + ac^2/(Ec*tc)* P*c] - 6(1 - nu*^{2})/(E*)(ao/h)^3(1 + h*betac/2) * [Pe(Zv + Zm*Q1) + 2/ao^2*Zm*Q2] }

Channel Stress Summation
|Sigmacm| + |Sigmacb| =<SPSc

3.3 ANSYS Calculation (Software Approach)

ANSYS is engineering simulation software. ANSYS Mechanical software is a comprehensive FEA analysis (finite element) tool for structural analysis, including linear, nonlinear and dynamic studies.

3.3.1 Steps to do analysis in ANSYS software

Step 1: Make a 3 D model in ANSYS workbench 14.0 for section of shell and tube heat exchanger.
Step 2: In steady state thermal apply engineering data includes thermal conductivity, Young's modulus, Poisson's ratio, density etc.
Step 3: The engineering data and geometry is transformed to static structural
Step 4: In model part of static structural do meshing by using concept of Hexa dominant method, mapped face mesh and body sizing.
Step 5: Apply effective pressure at the inner surface of the heat exchanger
Step 6: Apply fixed constraint at the one end of the channel
Step 7: Solve for equivalent stresses obtained at static structural part
This steps shows the procedure to do analysis of this paper in the ANSYS software.

ISSN 2348 – 7968

3.3.2 Overview of ANSYS in This Project

ANSYS 14.0 has a facility to make a 3 D model in workbench. The Half section of shell and tube heat exchanger is made in this ANSYS 14.0 workbench and applied symmetry to this model. The engineering data then applied to the model. The given material is alloy of chromium and nickel.

The engineering data and geometry of the steady state thermal is then linked to the static structural part as shown in fig.3.6. The meshing is done in the model part as shown in fig. 3.7. Meshing is done by using hex dominant method, body sizing and mapped face meshing. Then effective pressure of 1.1549 MPa which is obtained from the mathematical calculation based on ASME code is applied to the inner surface of the heat exchanger as shown in the fig.3.8.That effective pressure incorporates the effect of shell and channel side temperature as well as the rim temperature. The fixed constrained is applied to the one end of the channel as shown in fig. 3.9

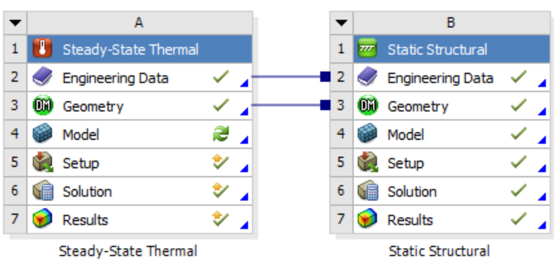


Fig. 3.6 Link in ANSYS 14.0

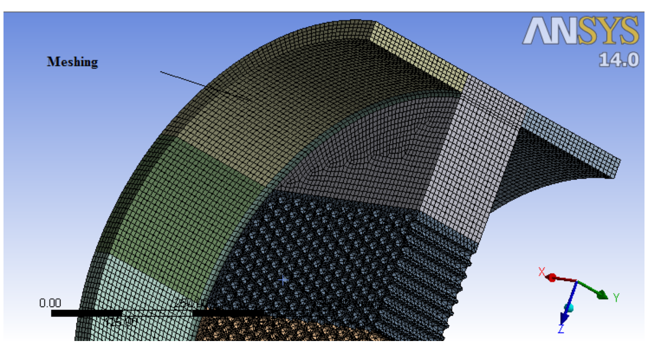
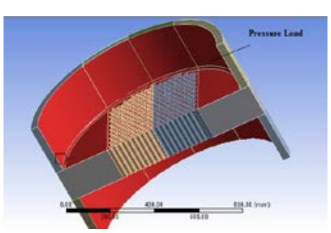


Fig. 3.7 Meshing on HE



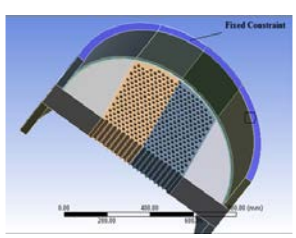


Fig.3.8 Pressure Load

Fig.3.9 Fixed Constraint

4 Results and Validation

Results obtained for different stresses from PVElite and ASME method are shown below:

4.1 Results

Table 4.1: Different stresses

SR. NO.	Method / Stresses	ASME Code (Mathematical Method) N/mm^2	PVElite (Software Method) N/mm^2
1	Tubesheet bending Stresses	-23.5506	-23.449
2	Tubesheet Shear Stresses	-5.1117	-5.1117
3	Shell to tubesheet Junction	30.775	31.937
4	Channel To Tubesheet junction	63.569	60.12
5	Axial Membrane Stress in Shell	-1.7	-1.7792

Table 4.1 shows the different stresses generated by PVElite software and their validation with ASME code. Different stresses produced at different region. As the result of high temperature and pressure the stresses produced at the junction of shell and tubesheet is more due to which expansion of tubesheet occurs. Therefore this project is concerned with the stresses produced at the junction of shell and tubesheet. The main area of interest of this project is the stresses produced at the junction of the shell and tubesheet. Due to the high temperature and pressure high stresses are generated at the junction of shell and tubesheet. The model is designed in ANSYS and boundary conditions are applied to the model. The results obtained form it shown in fig. 4.1.

Fig. 4.1 shows the stresses generated at the shell and tubesheet junction. The maximum stresses generated are 25.736 N/mm2 which shows by red color in the fig. 4.1. The minimum stress generated is 8.6186 N/mm2.

Table 4.2 shows the validation for stress generated for the general case having tubesheet thickness 250 mm and shell thickness 36 mm.

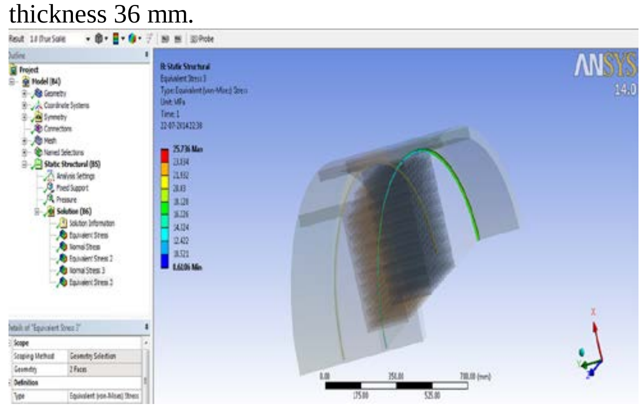


Fig. 4.1 Generated Stresses

ISSN 2348 – 7968

Table 4.2 Validation for junction stress

SR. NO.	Method / Stresses	ASME Code (Mathematical Method) N/mm^2	PVElite (Software Method) N/mm^2	ANSYS (Software Method) N/mm^2
1	Shell to tubesheet Junction	33.0775	31.937	25.736

4.2 Algorithm to achieve optimized safe design from PVElite:

Step 1: Make a model of shell and tube heat exchanger in PVElite.
Step 2: Apply input to model like material, temperature, pressure, dimensions etc.
Step 3: Assume tubesheet thickness (X1)
Step 4: Calculate different stresses
Step 5: If stresses within allowable then go to step 7 else go to step 3
Step 6: Reduced tubesheet thickness and go to step 4
Step 7: If (induced stresses- allowed stresses) <= 10% then go to step 8 else go to step 6.
Step 8: Optimized safe design

As from the methodology the different cases are studied to obtain optimized thickness by varying the thickness as shown in table 4.3.
The result pass and fail is made on the stresses generated on the heat exchanger in PVElite software.
Case 1 is having the details that are given by the Aker Solutions Company. In case 2, 3, 4 varied the tubesheet thickness (T1 & T2) and checked the results whereas in case 5 & 6 varied the shell and channel thickness (C1, C2, S1 & S2). From the table 4 it is conclude that the case 4 is optimized case for selection of shell and tube heat exchanger as weight is directly proportional to the cost.

4.3 Validation

The case 4 is selected as optimized case in this project from PVElite analysis. It is necessary to validate the stress produced at the shell to tubesheet junction. The changes made in the design of the previous general case in ANSYS 14.0 according to the case 4.After solving the model in the ANSYS it shows the result as shown in fig. 4.2 for the junction of shell and tubesheet.

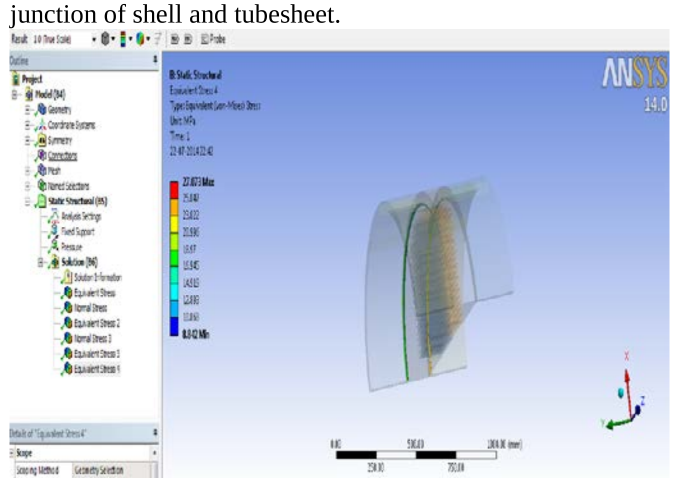


Fig. 4.2 Generated stress for optimized case

The maximum stresses produced at the junction of shell and tubesheet is 27.073 N/mm2. It is shown by the red color in the above fig. and minimum stresses with the blue color. This stresses are validated with the stresses produced by the PVElite software showing 29.39 N/mm2 shown in table 4.4.

Table 4.4 Validation for stresses for optimized case

SR. NO.	Method / Stresses	PVElite (Software Method) N/mm^2	ANSYS (Software Method) N/mm^2
1	Shell to tubesheet Junction	29.39	27.073

Table 4.3 Optimization based on weight

	Case 1		Case 2		Case 3		Case 4		Case 5		Case 6	
	t(m)	Wt(kg)	t(m)	Wt(kg)	t(m)	Wt(kg)	t(m)	Wt(kg)	t(m)	Wt(kg)	t(m)	Wt(kg)
C1	0.036	1070.04	0.036	1070.04	0.036	1070.04	0.036	1070.04	0.036	1070.04	0.034	1008.46
C2	0.036	1070.04	0.036	1070.04	0.036	1070.04	0.036	1070.04	0.036	1070.04	0.034	1008.46
S1	0.036	1284.05	0.036	1284.05	0.036	1284.05	0.036	1284.047	0.036	1284.047	0.034	1210.15
S2	0.036	1284.05	0.036	1284.05	0.036	1284.05	0.036	1284.047	0.036	1284.047	0.034	1210.15
SB	0.014	1461.27	0.014	1461.27	0.014	1461.27	0.014	1461.266	0.014	1461.266	0.014	1461.27
T1	0.25	1095.34	0.12	525.766	0.14	613.393	0.15	657.2069	0.16	701.0207	0.25	1095.34
T2	0.25	1095.34	0.12	525.766	0.14	613.393	0.15	657.2069	0.16	701.0207	0.25	1095.34
TW (Kg)		8360.13		7220.97		7396.23		7483.854		7571.481		8089.16
Result		Pass		Fail		Fail		Pass		Pass		Pass

ISSN 2348 – 7968

5 Conclusions

1. Due to high temperature and pressure large stresses are generated at the junction of shell and tubesheet junction of heat exchanger. This stresses are validated with Software (PVElite, ANSYS) and Mathematical (ASME Code, Section VIII, Division I) method.
2. Optimized thickness of tubesheet is achieved from table 4.3 and the stresses produced in the optimize case is also validated with the PVEite and ANSYS software. So it is economical as well as it ensures safety of heat exchanger.

Appendix

The Tubesheet Bending Stress - Original Thickness [Sigma]:
$= (1.5 * Fm / mu^*) * (2 * ao/(h - h'g))^2 * Pe$
$= (1.5 * 0.2169 / 0.2000) * (2 * 442.5800 / (250.000 - 0.000))^2 * -1.15$
= -23.5506 MPa

The Tubesheet Average Shear Stress - Original Thickness [Tau]:
= (1 / (2*mu)) * (ao/h) * Pe
= (1 / (2*0.200)) * (442.5800 /250.000) * -1.155
= -5.1117 MPa

Shell Stress Summation
|Sigmasm| + |Sigmasb| =<SPSsb
|-1.8 | + |-28.975 | =< 120.90 MPa
30.775 must be < or = 120.90 MPa

Channel Stress Summation
|Sigmacm| + |Sigmacb| =<SPSc
|4.9 | + |58.6690 | =< 81.38 MPa
63.569 must be < or = 81.38 MPa

Acknowledgments

Although it is a stimulating and motivating experience, completing a project takes a lot of effort, time and energies. And apart from the efforts of one's own, the success of any project largely depends upon the guidelines and encouragement of many others. I would like to take this opportunity to express my deepest gratitude to the people who have been instrumental in the success of this project.

References

[1] Feng Qing-xiao, "Discussions for the Integral Joint Structure of Tubesheet to Shell on Tubular Heat Exchanger." SINOPEC Engineering Incorporation, Beijing 100101, China
[2] KhosrowBehseta, Donald Mackenzie and Robert Hamilton, "Ratcheting Assessment of a Fixed Tube Sheet Heat Exchanger Subject to In Phase Pressure and Temperature Cycles." Journal of pressure vessel technology, volume 133, issue 4.
[3] ASME, ASME Boiler & Pressure Vessel Code Section VIII Division 2, New York, American Society of Mechanical Engineers, 1998. (Cited in Zengliang Gao.,et al,2005)
[4] Vijay Kumar and Pardeep Kumar, "Mechanical design of pressure vessel by using PVELITE software." International Journal of Scientific and Research Publications, Volume 4, Issue 4, April 2014