Design Development and Analysis of Catbari/Chocolate Wax Melting Tank

Prof. Dhananjay C. Kadlag[1], Prof. Vipul R. Navale[2]
Prof. Amit G. Hejib[3], Prof. Arvind L. Dhobale [4]
Prof. Ram A. Gadekar [5] , Prof. Kumar D. Kolhe [6]

[1,2,3,4,5,6]*Asst.Professors, Department of Mechanical Engineering, Jaihind College of Engineering ,Kuran, India.*

[1]dckadlag@gmail.com
[2]vipulnavale@rediffmail.com
[3]amithejib84@gmail.com
[4]dhobalearvind1@gmail.com
[5]rgadekar2009@gmail.com
[6]kumarkolhe@gmail.com

Abstract:- Wax melting is the most important process in industry in order to prepare a final product. While preparing that product it is necessary that to melt a wax in proper amount as well as the required condition. During melting of wax or pulp because of high temperature more amount of thermal stresses are developed and when that thermal stresses are exceeds certain limits then the welding section get weak and because of that there will be leakages problem at joint so that loss of thermal energy through joints. But if we design the tank for wax melting by applying the seamless welding process we are easily avoid those leakages at the joint. So there will be a need to design a tank by seamless welding process to avoid the thermal loss and reduce the thermal stresses. So in this paper we try to complete the design according to actual design dimensions and try to prepare the designed model with the help of catia software. After preparing that tank with that software I try to complete whole analysis with the help of ansys software . This analysis will be carried in order to get equivalent stresses, maximum principle stresses and the total deformation of the assembly. Also from the design and analysis it is clear that it is clear that the selected wall thickness of 5mm will be on safe side so there will be a optimization of thickness.

Keywords: Wax Melting, Analysis with Ansys, Thermal stresses. Principle stresses.

I. INTRODUCTION

In all over the world, food is an essential for human in day to day life. Catbari, chocklet and many other foods. While preparing such food the basic raw material is wax and it is very important to prepare a final product. For converting that raw material in to final product the device required is that melting tank. Melting tank is the device which is used to melt the wax under high temperature. Now in industry to melt the wax a pressure vessel are used. but the drawback of pressure vessel is the high thermal stresses are developed inside the vessel and the leakage problem at the joint, and because of that there will be a loss of thermal. So to avoid that we try to design and developed a wax melting tank for melting the wax. First we are try to check the design and then developed a tank according to the requirements of end users. Now my aim is to design a tank with some software like catia, pro-e, hyper mesh etc. because of my simplicity I select catia to design wax melting tank. In this design I try to complete design of tank. This tank include the different ports like inlet, outlet, inspection, manhole and drain along with left and right hand flange. As the seamless welding is provided so there will be no any leakages problem at the joints

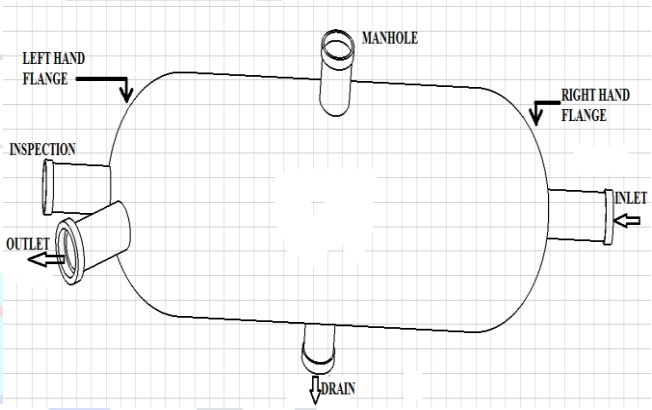


Fig.1. Wax melting tank

Firstly, the wax passed through the inlet to inside the tank after heating it will be collect through outlet section. Drain will be provided in order to remove unwanted material along with them manhole is provided for inspection purpose.

Problem Statement-Melting a wax is a serious task in any food industry. Now in industry they referred a cylindrical Pressure vessel for wax melting but the problem is that during wax melting high temperature are developed inside and due to that high thermal stresses are developed . Because of that there should be leakages of wax through welding joint and there will be a loss of thermal. It create serious problems at the time of working in site, to remove this we must assure about vessel design .

Methodology

In order to design a wax melting tank we try to replace it with the help of pressure vessel. Methodology consists of application of scientific principles, technical information and imagination for development of new or improvised wax melting tank to perform a specific function with maximum economy and efficiency. This project work will relate to design of tank, Optimization of stresses, and selection of proper method at joint to avoids leakage at the joints including:

1. Problem Definition
2. Literature Review
3. Finding out Design Parameters of Tank
4. Design and Calculations of Tank
5. Design of tank in catia software (2D and 3D)
6. Ansys analysis
7. Analysis by FEM Software
8. Result and Discussion

9. Conclusion.

Objectives-The main objectives are

1. To design the tank with ASME code same as the pressure vessel.
2. To optimise the thickness of tank so that material cost saving.
3. To replace the welding method by seamless welding to avoid leakages at the joints.
4. To do the analysis of tank with ansys software.

II. LITERATURE SURVEY

Various researchers have worked for the development of wax melting with pressure vessel

Sumit V Duplet. [May-2014], conducted a study of "Review on Stresses in Cylindrical Pressure Vessel and its Design as per ASME Code".They found that different stresses which are exerted on the pressure vessel. The total design will be done on the basis of ASME code this analysis will give the exact values of the different stresses like maximum principle stresses, Equivalent stresses based on American society of mechanical engineering.[1]

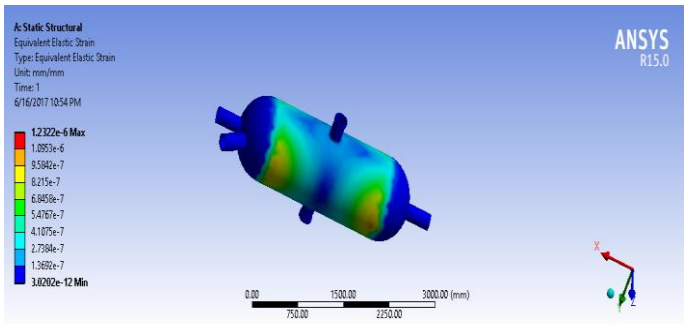
AntonioRamos [2014]"The melting process of storage materials with relatively high phase change temperatures in partially filled spherical Vessels". In this paper they studied that the different melting processes of storage material with relatively high phase change material when temperatures in partially filled spherical Vessels.[2]

S Ravinderet.[Feb.-2013], "Design and analysis of pressure vessel assembly for testing of missile canister sections under differential pressure". This paper give the information about Design and analysis of pressure vessel assembly during the working on site for testing of missile canister sections under differential pressure and this testing will be carried out for different pressure conditions.[3]

Apurva R. Pendbhaje [March-2012], "Design and analysis of pressure vessel". This paper states that to carry the design of pressure vessel to melt the wax and total analysis will be carried out with the help of analysis.[4]

M.Rahimi[Jun-2012],"A combine experimental and computational study on the melting behavior of a medium temperature phase changes to rage material inside shell and tube heat exchanger".this gives the information about experimental and computational study on the melting behavior of a medium temperature phase changes to rage material inside shell and tube heat exchanger"[5]

III. FINITE ELEMENT ANALYSIS



. Fig. 2 Equivalent elastic strain.

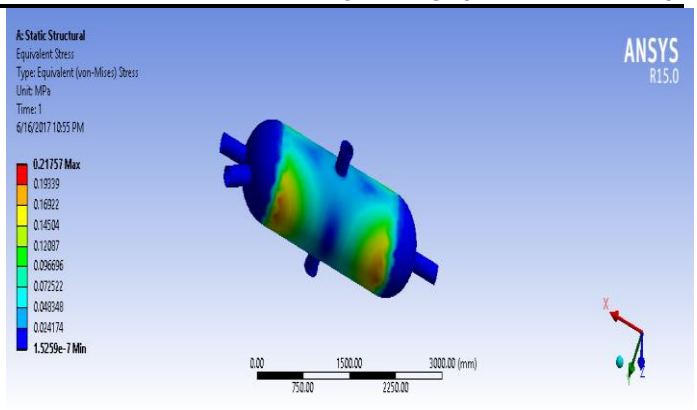


Fig. 3. Principal stress diagram.

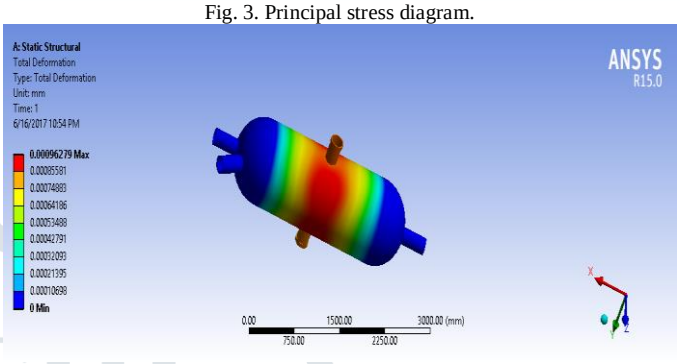


Fig.4 .Total Deformation.

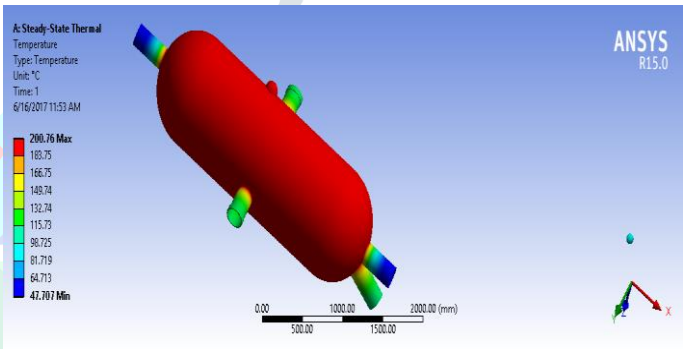


Fig.5.Temperature at Steady State

With the help of FEA analysis it gives the information about equivalent stress, maximum principle stress and the total deformation of the tank. For the equivalent stress we get the maximum and minimum stress values along with them we get the maximum and minimum principle stress value. But with the analysis of total deformation it should be clear that without application of pressure the minimum deformation will be 0 and by the application of pressure we get the maximum deformation 0.75285.

Meshing

One of the purposes of meshing is to actually make the problem solvable using Finite Element. By meshing, you break up the domain into pieces, each piece representing an element. You need these elements to be able to apply Finite Element since Finite Element is all about having a basis local to an Element and stitching a bunch of local solutions together to build the global one. If you did not mesh and just assumed some basis that covered the whole domain, that would be a Spectral Method.

In case of tank body having 49 faces and it is divided in to 133658 elements from that 268217 nodes are formed.

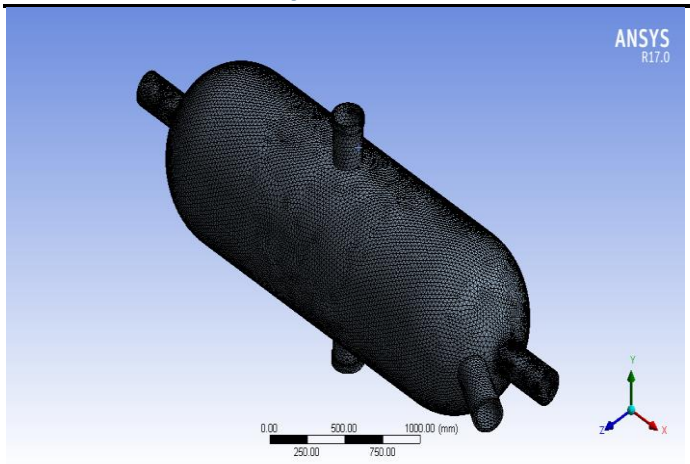


Fig.5. Meshing

Boundary Condition

IV. EXPERIMENTAL VERIFICATION

TANK DESIGN

Fig.6. Actual Tank Model

Formulae:
Cylindrical Shell Thickness
Vessel Height = Shell OD
Shell OD = 650 mm
So vessel height also 650 mm
Height for static Head calculation
Height for static Head = vessel Height. + Top Nozzle projection + Bottom Nozzle Projection
=650+150+150
=950 mm
Maximum Possible Static Head,H (mm) = 1500 mm
(rounded , considering all (Max. Distance Between Topmost and possible Tolerance) Bottom Most Pressure Parts.)

Design Internal Pressure including Static Head for Calculations:
Density of Contents, 1000(Kg/m^3)
Static Head pressure (P)

$P = \rho * g * H$
$=1000*9.81*1500*10^{-6}$
= 0.01471 MPa
= 0.015 MPa
Design Pressure = P + Pressure due to Static Head
= 0.491 + 0.015
= 0.505 MPa
Hydrostatic Test Pressure = (MAWP x Ratio x 1.5)
= 0.491 x 1.000 x 1.50
= 0.736 MPa
= 7.508 Kg/cm^2
As per L & T Datasheet Hydro test to be carried out at 5.25 Kg/cm2 in shop in Vertical position only

TABLE XIII
SPECIFICATIONS OF TANK

Melting Capacity	200 Kg.
Size of Tank	Φ 1470 x 975mm.
Tank Material	S.S.304
Heating Element	Uniformly heated by electric flat heaters.
Control	Temperature controlled by Digital A' meter Phase indicates by lamps, Auto controls by Thermo states
Electrical Parts	Standard CE Marked.
Rating	6 K.W.

1. Hydro test body metal Temperature =17°C above MDMT & need not Exceed 48°C [Ref.UG-99 (h)].
2. MAWP is assumed same as Design Pressure as per UG-99, (Note: 34, Page:74 of code).
3. Service Classifications is normal (non-Lethal).
4. Overpressure Protection as per UG-125 is in Client's Scope

$$t = \frac{P*R}{S*E-0.6*P}$$
$$t = \frac{0.49*44.5}{(815.2*1)-(0.6*0.49)}$$
t = 0.27 cm
t = 2.7 mm
$$P = \frac{S*E*t}{R+0.6*T}$$
$$P = \frac{0.49*1*0.27}{44.5*0.6*0.27}$$
P = 4.92 Kg/ cm^2
Hence Internal Design Pressure (Pi) = 4.92 Kg/ cm^2 = 0.483 MPa
Maximum Allowable Stress(S)
$$S = \frac{P*R*0.6*t}{E*t}$$
$$s = \frac{0.495*44.5+(0.6*0.27)}{1*0.27}$$
S = 814.75 Kg/ cm^2
S = 79.95 MPa
Inside Radius (Corroded) (R) = R+ CA = 495.000 + 0.000 = 495.00 mm
Provided Thickness (Nominal) = 5.00 mm
$$t = \frac{P*R}{S*E-0.6*P}$$
$$t = \frac{4.99*44.5}{(815.2*1)-(0.6*4.99)}$$
t = 0.27 cm
t = 2.7 mm
$$P \frac{S*E*t}{R+0.6*t}$$
$$P = \frac{0.49*1*0.27}{44.5+0.6+0.27}$$
P= 4.92 Kg/ cm^2
Joint Efficiency (E) = 1.00

JointEfficiencyFactor=0.385SE =85 * 80 * 1 = 30.80 MPa
Minimum Required Thickness =

$$t = \frac{4.99*44.5}{(815.2*1)-(0.6*4.99)}$$

t = 0.37 cm
t = 3.7 mm
Longitudinal Stress (Circumferential Joints) when the effect of supplementary loads as per UG-22 is absent
Joint Efficiency (E) = 1.00
Joint Efficiency Factor =1.25SE
= 1.25 * 80 * 1 = 100 MPa
Minimum Required Thickness =

$$t = \frac{P*R}{2*S*E+0.4*P}$$

$$t = \frac{4.99*44.5}{(2*815.2*1)+(0.4*4.99)}$$

t = 0.137 cm
t = 1.37 mm
Minimum required thickness shall be > 2.5 mm (3/32 in.) excluding Corrosion Allowance is 2.50 mm.
t = Greater of (3.70 , 1.37 , 2.50)
Governing thickness + Corrosion Allowance = 3.70 + 0.00 = **3.70 mm**

CHECK: Required Thickness= 3.139 mm< 5.000 mm (Provided) Thickness is Optimum

External Pressure Calculation
Corroded thickness (t) = 5.00 mm
Total Length between stiffing Ring (L) = 1750.00 mm
Outside Diameter of Cylindrical shell (Do) = 1000 mm
L/Do Ratio (L/Do) = 1.750
Do /t Ratio (Do /t) = 200
Factor A from Fig G (A) = 0.00125
Factor B from chart CS-2 (B) = 2250
Pa = 4B/3(Do/t)(x)

$$Pa = \frac{4*2250}{3*(200)}$$

Pa = 15 MPa
Maximum Allowable External Pressure [MAEP] (Pa) = 15 MPa
Required thickness under external pressure (t)
t = (3PDo/4B)+CA(xi)
t = (3 * 4.99* 1000 / 4 * 2250) + 1.5
t = 3.16 mm
t_f= 3.16 + 1.5 = 4.66 mm

Hence shell thickness is safe at 5.00 MM

External Pressure Maximum Allowable Working Pressure at given thickness, corroded [MAWP]

$$P = \frac{2*S*E*t}{L*M+0.2t}$$

But M = 1.54

$$P = \frac{2*815*1*0.05}{99*1.54+0.2*0.05}$$

P = 0.53 Kg/ cm^2
Maximum Allowable Pressure at Cold & New Condition [MAP]:

$$P = \frac{2*S*E*t}{L*M+0.2t}$$

Crown Radius (L) = 990.00 mm
Knuckle Radius (r) = 99.00 mm
But M = 1.54

$$P = \frac{2*815*1*0.05}{99*1.54+0.2*0.05}$$

P = 0.53 Kg/ cm^2

SF required thickness
Minimum Required Thickness

$$t = \frac{P \times R}{(s \times E - 0.6 \times P)} + CA$$

$$t = \frac{0.53 \times 44.5}{(815 \times 1 - 0.6 \times 4.99)} + 0.00$$

t = 0.29 cm t = 2.9 mm

External Pressure Calculation
P = 1.67 x External Design Pressure
= 1.67 * 6.114
= 10.21 Kg/ cm^2
= 1.002 MPa
Required thickness, t = P * L*M / (2 * S * E - 0.2 * P)
t = 10.21 * 99 * 1.54 / (2 * 815 * 1 - 0.2 * 10.21)
t = 0.95 cm
t = 9.5 mm
Requirement for Cold Forming As Per Ucs- 79

From the above table III, we have proved that the strain values observed in experimental result with FEA result with marginal acceptable error.

V. RESULTS AND DISCUSSIONS

From all the calculation it is clear that whatever thickness we provided that should be on the safe side. So there should be the avoid of the leakages problems. Along with them we found that there will be the optimization of thickness. Also we take the hydrostatic test and take the result.

TABLE XIII
ANSYS RESULTS

	Equivalent elastic strain(Mpa)	Von misses stress (Mpa)	Total deformation (mm)
Min.	1.2322	1.5259	0
Max.	3.022	0.21757	0.00096379

VI. CONCLUSION

From this Design calculation we can conclude that there will be a Required Thickness= 3.139 mm< 5.000 mm (Provided) Thickness is Optimum External Pressure Calculation i.e. it is in safe zone .Also From the external pressure calculation we made the conclusion that there will be Requirement for Cold Forming As Per Ucs- 79 as the calculated thickness will be 9.5mm.
From the FEA analysisit should be clear that all the design should be on the safe side.

.

REFERENCES

[1] Sumit V Duplet. [May-2014], "Review on Stresses in Cylindrical Pressure Vessel and its Design as per ASME Code".

[2] AntonioRamosArchiboldetJoséGonzalezAguilar, MuhammadM.Rahman,D.YogiGoswami, ManuelRomero,EliasK. Stefanakos[2014]"The melting process of storage materials with relatively high phase change temperatures in partially filled spherical Vessels".

[3] S Ravinderet.[Feb.-2013], "Design and analysis of pressure vessel assembly for testing of missile canister sections under differential pressure".

[4] Apurva R. Pendbhaje [March-2012], "Design and analysis of pressure vessel".

[5] M.J.HosseinietA.A.Ranjbar,K.Sedighi, M.Rahimi[Jun-2012],"A combine experimental and computational study on the melting behavior of a medium temperature phase changes to rage material inside shell and tube heat exchanger".

[6] E.S.BrbozaNetoM.Chludzinski,P.B.Roese,J. S.O.fonseca [April 2011] "Experimental and numerical analysis of a LLDPE/HDPE liner for a composite pressure vessel".

[7] M.Maleki, B. Haghpanahjahromi.E. Hosseinian [Aprial 2010] " Residual stress analysis of autofretaged thick-walled spherical pressure vessel.

[8] N.H.S.TayF.Bruno, M.Belusko[Dec.2012] "Experimental investigation of dynamic melting in a tube-in-tank PCM system".