OPEN ACCESS

International
Journal
Of Modern Engineering Research (IJMER)

Finite Element Analysys of Heat Exchanger

Shubham[1], Shubham Sharma[2], Rohan Singh[3], Manish Kashiv[4]
[1](MECHANICAL,CHANDIGARH UNIVERSITY, INDIA)
[2](MECHANICAL,CHANDIGARH UNIVERSITY, INDIA)
[3](MECHANICAL,CHANDIGARH UNIVERSITY, INDIA)
[4](MECHANICAL,CHANDIGARH UNIVERSITY, INDIA)

ABSTRACT: *In this paper the finite element analysis of aHeat Exchanger is done. The geometry was modelled in CATI A V5 R21 and finite element analysis had been performed in ANSYS12 WB. FE analysis is was used to determine stress analysis at.*

Keywords: *HEAT EXCHANGER, FE ANALYSIS, CATIAV5 R21, ANSYS 12 WB*

I. Introduction

Transfer of heat from one fluid to another is an important operation for most of the chemical industries. The most common application of heat transfer is in designing of heat transfer equipment for exchanging heat from one fluid to another fluid. Such devices for efficient transfer of heat are generally called Heat Exchanger.In Heat Exchanger, there is one most familiar type of exchanger is Shell and Tube Exchanger ,the designers are allow to use wide range of pressures and temperatures because of flexibility and compact. Shell and tube heat exchangers are basically used for the transfer of heat in industrial process applications. They are available in a variety of diameters and lengths. It is manufactured using the most advanced technology available and is specifically engineered for a wide range of applications. Every unit is designed, manufactured and tested to meet the quality requirements of the ASME code: Section VIII Division #1 and ISO 9002. There are two more categories.

II. Model Generation And Boundary Condition Of Heat

Model is created in Catia V5 and Imported in Ansys Workbench as stpfile (figure 1). The meshing of geometry was performed in ANSYS Workbench.

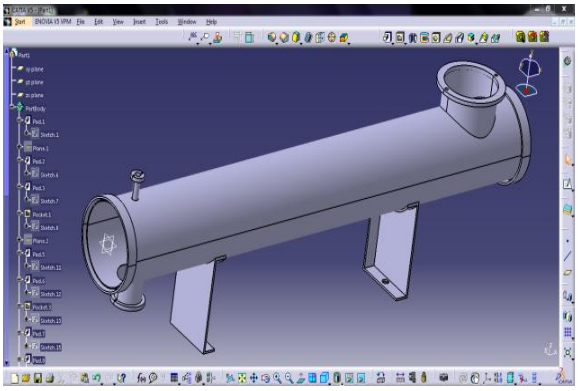


Figure 1 Heat Exchanger Geometry

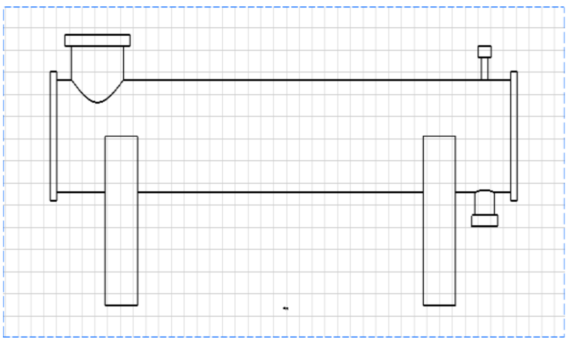


Figure1: Front view

Property of material

Property	Value	Unit
Density	8000	Kgm^-3
Young's modulus	193000	MPa
Poisson's ratio	0.3	
Tensile yield strength	205	MPa
Compressive yield strength	205	MPa
Tensile ultimate strength	515	MPa
Isotropic thermal conductivity	16.2	Wm^-1k^-1

Meshing: Meshing of the Geometry is also done in the ANSYS software, details of which are given below:

- Type of element: Triangular
- Element size: 98
- Physical Preference: Mechanical
- Behavior: Soft
- Transition : Fast

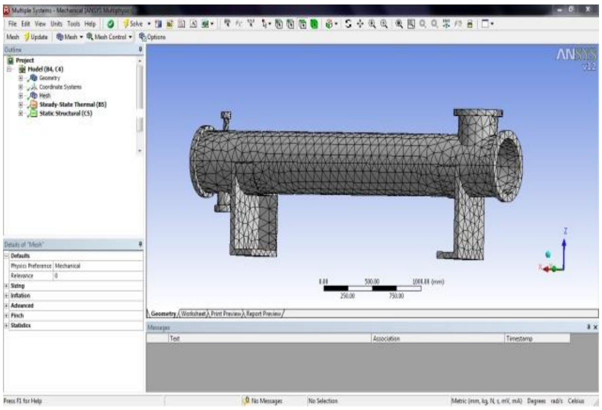


Figure 2 Meshing

METHODOLOGY

In practice, a finite element analysis usually consists of three principal steps:

1. Pre-processing:

The user constructs a model of the part to be analyzed in which the geometry is divided into a number of discrete sub regions, or elements," connected at discrete points called nodes." Certain of these nodes will have fixed displacements, and others will have prescribed loads. These models can be extremely time consuming to prepare, and commercial codes vie with one another to have the most user-friendly graphical \pre-processor" to assist in this rather tedious chore. Some of these pre-processors can overlay a mesh on a pre-existing CAD, so that finite element analysis can be done conveniently as part of the computerized drafting-and-design process.

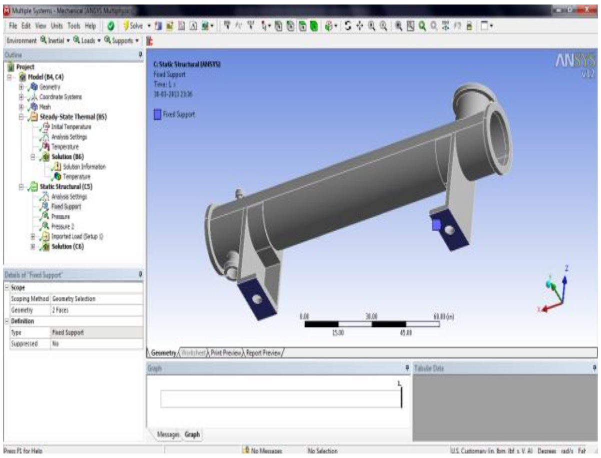
2. Analysis:

The dataset prepared by the pre-processor is used as input to the finite element code itself, which constructs and solves a system of linear or nonlinear algebraic equations. The formation of the K matrix is dependent on the type of problem being attacked, and this module will outline the approach for truss and linear elastic stress analyses. Commercial codes may have very large element libraries, with elements appropriate to a wide range of problem types. One of FEA's principal advantages is that many problem types can be addressed with the same code, merely by specifying the appropriate element types from the library.

3. Post processing:

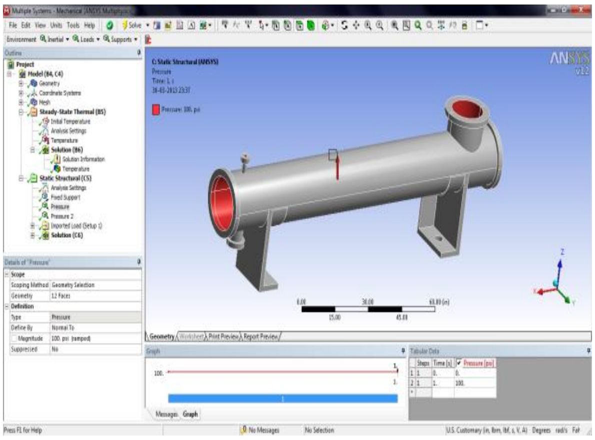
In the earlier days of finite element analysis, the user would pore through reams of numbers generated by the code, listing displacements and stresses at discrete positions within the model. It is easy to miss important trends and hot spots this way, and modern codes use graphical displays to assist in visualizing the results. Typical postprocessor display overlays colored contours representing stress levels on the model, showing a full-field picture similar to that of photo elastic or moiré experimental results.

Fixed part:



Pressure:

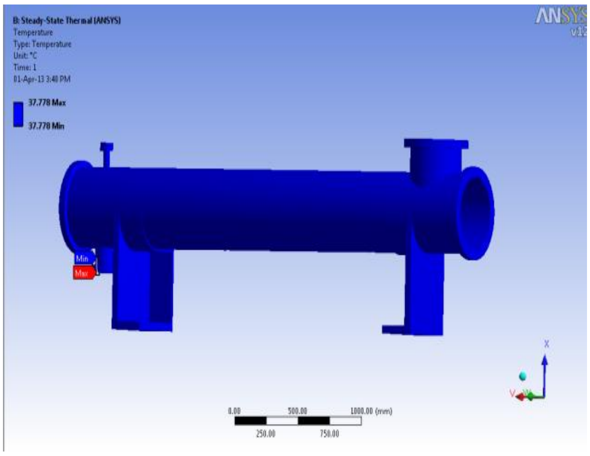
- **100psi in all internal part of shell**
- **130psi pressure in major part of shell**



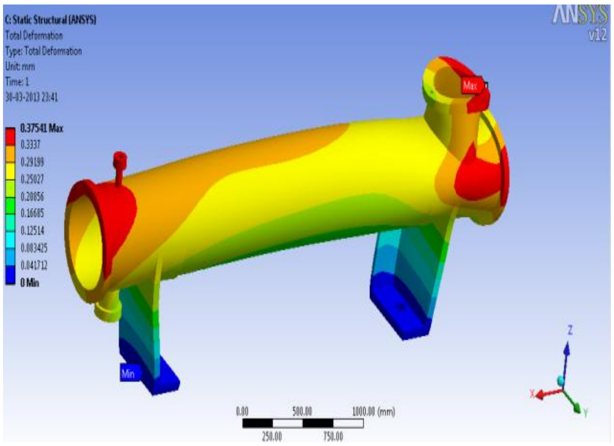
Analysis:
Static structure (ANSYS)
Steady state thermal (ANSYS)
Steady- state thermal to static structure (ANSYS)

III. Solutions

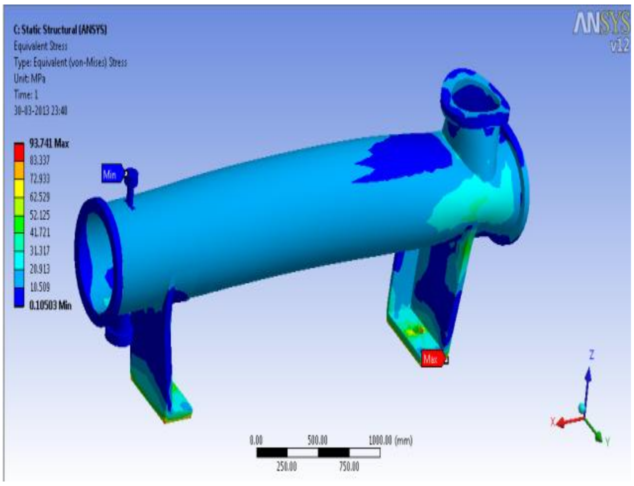
After solving the model in ANSYS, following results are obtained:
Total temperature:



Maximum temperature is 37.778^{0}c
Minimum temperature is 37.778^{0}c
Total deformation:



Maximum deformation is .037541mm
Minimum deformation is 0mm
THERMAL STRESS



Maximum stress is 93.741 Mpa
Minimum stress is 0.10503 Mpa

IV. Conclusions

From the above static structural analysis we found that maximum deformation **.037541mm**, in the thermal to structural analysis the maximum stress is 847.33Mpa, the material yield strength is 1002 Mpa , comparing with this value the stress values are less . From this we conclude that The Axial Turbine Blade is safe.

REFERENCES

[1]. Ackerman R.A. 1997, Cryogenic Regenerative Heat Exchanger, International Cryogenics monograph series, Plenum publishing, New York
[2]. Buchlin J.M. 1991, Industrial Heat Exchangers, Lecture Series 1991-04, Von Karman Institute for Fluid Dynamics, Belgium
[3]. Apblett, W.R.,Jr.,ed.,1982 Shell and Tube Heat Exchangers, American Society for Metals, Metals Park.OH
[4]. DS and DG Wilson 1996 Gas Turbine Regenators, Chapman and Hall, New York
[5]. Yahya S.M Turbines compressors and Fans , ISBN 9780070597709