IJESRT
INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY

DESIGN AND ANALYSIS OF TUBULAR HEAT EXCHANGER FOR VARYING TEMPERATURE PROFILES USING ALTAIR HYPERMESH

Nitesh More*, A.B. Jayant
* M.Tech. Scholar (Mechanical Engg.) Laxmi Narayan College of Technology & Science, Indore.
Asst. Professor, L.N.C. Technology & Science, Indore.

ABSTRACT

This Paper deals with the heat transfer of shell and tube or simply tubular type heat exchanger for different no of tubes with varying temperatures. It provides a linear steady state thermal analysis which further be analyzed in analogy of structural parameters. The thermal stress and heat transfer induced is studied using Altair HYPERMESH HyperworkV13.0 simulating software at steady state boundary conditions. Heat exchanger is designed for analyzing heat transfer rate using air as a working medium for parallel flow justifying the phenomenon of convection within the shell and tubes used. Tubular exchangers are used primarily for liquid-to-liquid. An attempt is made in this paper is to the design of shell and tube heat exchangers by modeling & CFD analysis in Altair HYPERMESH using tube material as Aluminum, which is the best source of heat transfer and modeling procedure assembly of Shell and Tube is done using air as working medium. The thermal analysis of Shell and Tube heat exchangers is carried out by varying the temperature of inletting air. The geometrical modeling covers a shell containing five numbers of tubes with it. With the help of the simulating results, maximum & minimum temperature gradient ranges from 2.931 to 14.23kW/mm2 & 0.0 to 2.384E-07 kW/mm2 respectively. The maximum & minimum grid temperature ranges from 3.008E+15 to 15.04E+15 kW/ mm2 & -0.005157 to -0.02579kW/mm2respectively. The maximum & minimum element flux ranges from 6.947E+02 to 33.72E+02kW/mm2& 0.00 to 5.651E-05kW/mm2respectively. The pictorial representations of simulated results reveals towards the steady state linear thermal analysis.

KEYWORDS: HYPER-MESH, Shell & Tube type heat exchanger, Thermal stress, Tubular exchanger.

INTRODUCTION

The term heat transfer is use for thermal energy between a hot to a colder body. When a physical body such as an object or fluid or gases, is at a different temperature than its surroundings or another body, where temperature difference between object is proximity, heat transfer will occurs in such a way that the body and the surroundings reach thermal equilibrium. The transfer of thermal energy can occur in three different ways: Conduction; Convection and Radiation or any combination of these. In this paper, transfer of thermal energy is focused to be carried out using convection between the geometry of the heat exchanger design. This phenomenon of heat transfer is utilized in many of industrial applications and automations industries. The device called heat exchanger is employed to utilize transfer of thermal energy (enthalpy) between two or more fluids or gases (air), between a solid surface and a fluid or gases (air), or between solid particulates and a fluid or gases (air), at different temperatures and in thermal contact. The general function of heat Exchanger is to transfer heat from one fluid or gas to another fluid or gas. The basic component of heat exchanger involves tubes with one fluid or gas running through it and another fluid or gas circulating on outer surface. Thus three heat transfer operations are needed to be described:

1. Convective heat transfer from fluid to the inner wall of the tube,
2. Conductive heat transfer through the tube wall, and
3. Convective heat transfer from the outer tube wall to the outside fluid

The basic relationship for heat transfer by convection is:

$$q = hA(T_a - T_b) \quad \ldots\ldots\ldots\ldots (1)$$

Where q is the heat transferred per unit time, h is the heat transfer coefficient, T_a is the object's surface temperature and T_b is the fluid temperature. Convection is generally the dominant form of heat transfer in liquids and gas. Convective heat transfer involves the combined processes of conduction (heat diffusion) and advection (heat transfer by bulk fluid flow). Typical applications involve heating or cooling of a fluid stream of concern and evaporation or condensation of single- or multi-component fluid streams. In other applications, the objective may be to recover or reject heat, or sterilize, pasteurize, fractionate, distil, concentrate, crystallize, or control a process fluid [5].

In present heat exchangers are classified & available in many configurations, depending upon their application, process fluids, and mode of heat transfer and flow arrangements, relative direction of fluid flow, physical state of fluids and design-constructional features heat exchangers can be classified. Heat exchangers can transfer heat through direct contact with the fluid or through indirect ways. They can also be classified on the basis of shell and tube passes, types of baffles, arrangement of tubes (Triangular, square etc.) and smooth or baffled surfaces. The selection of a particular heat exchanger configuration depends on several factors which may include the area requirements, maintenance, flow rates, and fluid phase. In order to meet these widely varying applications, several types of heat exchanger have been developed [14].

NECESSARY FEATURES FOR A HEAT EXCHANGER

In order to work upon the featured design of the heat exchanger, the two important aspects are taken into consideration. These features would include maximum ratio of heat transfer to pressure drop along with least operating cost without compromising with reliability of designing system.

Higher heat transfer coefficient & area:

The heat transfer surface which uses local turbulence for single phase flow or have some important features for two phase flow will raise the heat transfer coefficient. Secondly to increase the heat transfer area one can use larger exchangers, but the more cost effective way is to use a heat exchanger having a large area density per unit exchanger volume, maintaining the integrity of the Specifications.

Lowering the Pressure drop

By using segmental baffles in a heat exchanger it will results in undesirable high pressure drop and also pumping cost as pumping cost is directly proportional to the pressure drop within the heat exchanger [1].

MODELING DETAILS

In this research, a compact heat exchanger is selected in order to raise the details of model and to make observation solid regards to the flow inside the heat exchanger. Design parameter and geometric parameter are presented in the table below. The geometric model with five numbers of pipes is inserted inside the heat exchanger. The working fluid used is being the air at various temperatures ranges [2].

In this paper five pipes are placed inside the heat exchanger in nearby vicinity to each other in order to create a uniform flow of air across the heat exchanger [8]. The geometric model is being optimized by varying the temperature ranges from 100 to 500 degree Celsius. The computation modeling involves pre-processing solving and post processing. The geometry modeling of tubular heat exchanger is explained below;

3.1 Geometry modeling

The model is being designed accordingly TEMA codes (Tubular Exchanger Manufacturers Association), using Altair HYPERMESH [11]. Design parameter are fixed geometric parameter have be taken similar to simple heat exchanger.

Table 1: Geometric dimension of tubular heat exchanger

Length of heat Exchanger	500mm
Outer diameter of tubular heat-exchanger	100mm
Inner diameter of tubular heat exchanger	80mm
Outer diameter of tubes of heat exchanger	10mm
Inner diameter of tubes of heat exchanger	8mm
Length of the tubes of heat exchanger	500
Number of tube	5

ISSN: 2277-9655
(I2OR), Publication Impact Factor: 3.785

3.2 Boundary conditions

Boundary conditions are needed and set according to the analysis and its modeling simulation [13]. The inlet and outlet condition for the velocity or pressure and temperature of the operating air is set before the simulation of the design considered. The following considerations are taken into account:

- The air is working fluid in this simulating design inside the tubes & shell.
- The air with thermal energy ranging from 100, 200, 300, 400 & 500 kW/mm^2 is inlet into the shell in iterative form.
- The constant wall temperature is assigned to the shell & tube walls.
- The outlet wall is assigned to zero gauge pressure to obtain relative pressure drop between inlet and outlet nozzle.
- The inlet velocity of air is uniform through out simulation.
- All surfaces are assigned with no slip.
- The zero heat flux boundary condition is assigned to the shell outer wall, assuming the shell is perfectly insulated outside [3].

3.3 Mesh selection

The process of meshing and mesh selection is obtained through simulation tool Altair hyper mesh. The shell volume is meshed using quadra-hedral element. The three dimension models are then discretized in HYPERMESH. In order to capture the thermal boundary layer the entire model is discretized using quadra-hedral mesh element along with grid generation of 2mm, which are precise and involve less computation effort. Fine control of the quadra-hedral mesh near the wall surface allows capturing the boundary layer gradient accurately. The heat exchanger is discretized into solid in order to have better to control over the number of nodes. The mesh is made finer for simulating conjugate heat exchanger phenomenon, so as to get the better results [6], [7].

RESULTS

The simulation is carried out with setting all the boundary conditions using Altair Hyper-mesh (as a simulation tool). The simulation results are obtained for different temperature ranges. The different iterations are placed for inlet air into the shell containing group of arranged pipes which from range 100 to 500 degree Celsius. Three different plots of temperature profile are taken into account for the comparison to analyze the heat transfer rate of simpler tube heat exchanger. For five numbers of pipes, air is blown at five temperature ranges.

4.1 Variation in Grid Temperature:

The contour plot for the Grid Temperature is obtained while taking the shell under consideration. The maximum & minimum temperatures are observed for the iterative temperature of air blown. The variation of temperature is summarized in the table below:

S.No.	Temperature variations (in ° C)	Grid temperature (in kW/mm^2)	
		Minimum	Maximum
1	100	-0.5157×10^{-2}	3.008×10^{15}
2	200	-1.031×10^{-2}	6.016×10^{15}
3	300	-1.547×10^{-2}	9.024×10^{15}
4	400	-2.063×10^{-2}	12.03×10^{15}
5	500	-2.579×10^{-2}	15.04×10^{15}

Table 2: Variation in Grid temperature

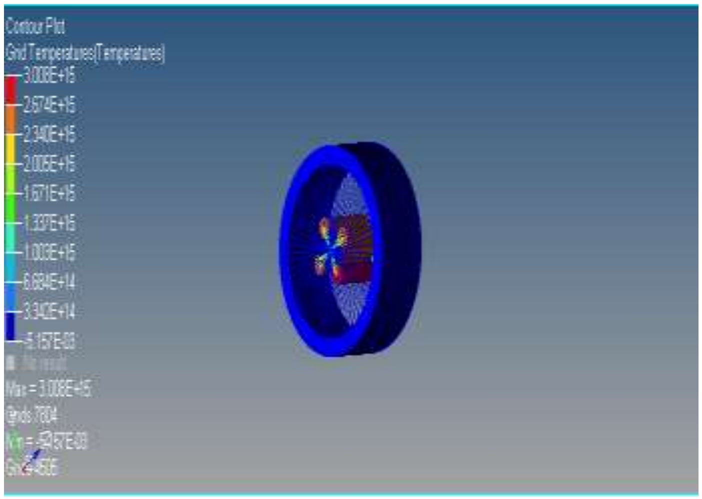

Figure 1: Contour plot for Grid temperatures at 100˚ C

4.2 Variation of Element temperature gradient:

The temperature Contours plots (for element temperature gradient) across the length of the tubular heat exchanger at different y axis distance will give an idea of the heat transfer flow in detail. Temperature variation can be observed, while having thermal analysis in the HYPERMESH. The maximum temperature can be viewed in the centre of the pipe, i.e. at a distance of 0.517 mm from left corner. The maximum & minimum temperature as observed are noted as in table 3:

S.No.	Temperature variations (in ˚ C)	Grid temperature (in kW/mm^2)	
		Minimum	Maximum
1	100	0.00	2.931
2	200	0.00	5.862
3	300	1.431 x 10^{-7}	8.419
4	400	1.908 x 10^{-7}	11.72
5	500	2.384x 10^{-7}	14.23

Table 3: Variation of Gradient Temperature

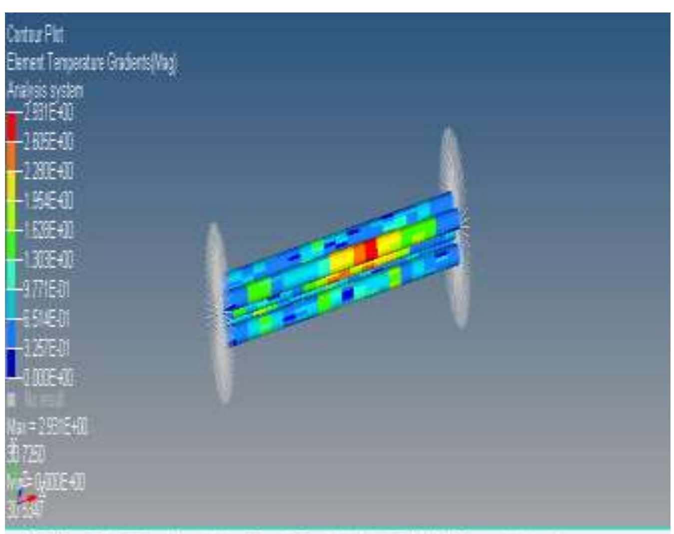

Figure 2: Contour plot for element Temperature Gradient at 100˚ C

The pictorial representation of results obtained for element gradient temperature is viewed to be in linear state. Similarly minimum gradient temperature is plotted against varying temperature of air.

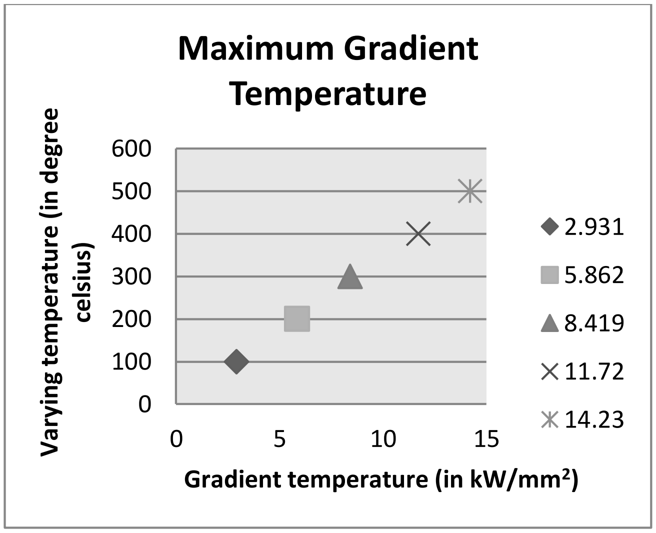

Figure 3: Maximum gradient temperature against temperature variations

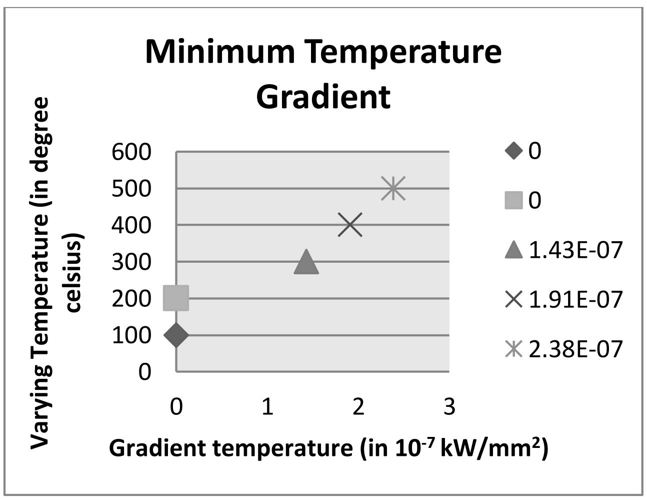

Figure 4: Minimum gradient temperature against temperature variations

4.3 Variation in Element Fluxes:

The simulation results for fluxes can be obtained from the contour plot for pipes along with shell geometry. The contour plot for the element flux reads out 33.72E+02 as maximum flux and zero minimum flux in the surrounding of pipes. These minimum and maximum values are generated for different temperatures of air as in table 4:

S.No.	Temperature variations (in °C)	Grid temperature (in kW/mm^2)	
		Minimum	Maximum
1	100	0.00	6.947×10^2
2	200	0.00	13.89×10^2
3	300	3.390×10^{-5}	19.95×10^2

4	400	4.679×10^{-5}	27.70×10^{2}
5	500	5.651×10^{-5}	33.72×10^{2}

Table 4: Variation of Element Flux

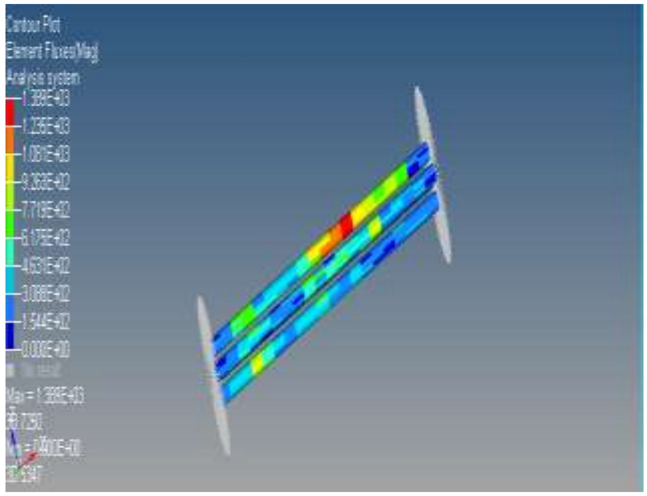

Figure 5: Contour plot for Element Flux at 200° C

The element Flux is also visualized on the X-Y axis by plotting the different values of maximum & minimum flux obtained through simulation. The graphical representation is viewed as below:

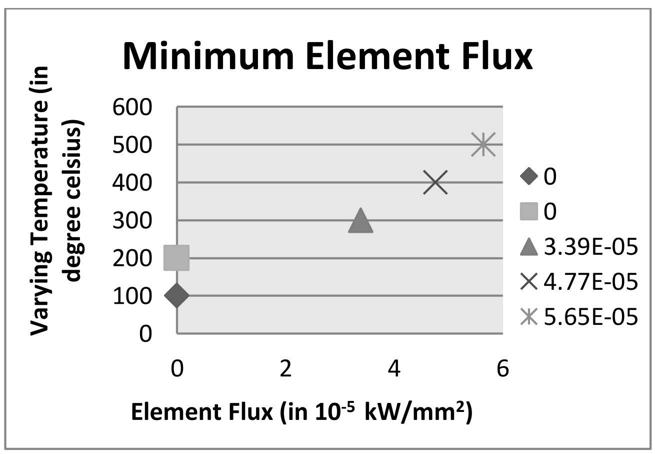

Figure 6: Minimum Element Flux against temperature variations

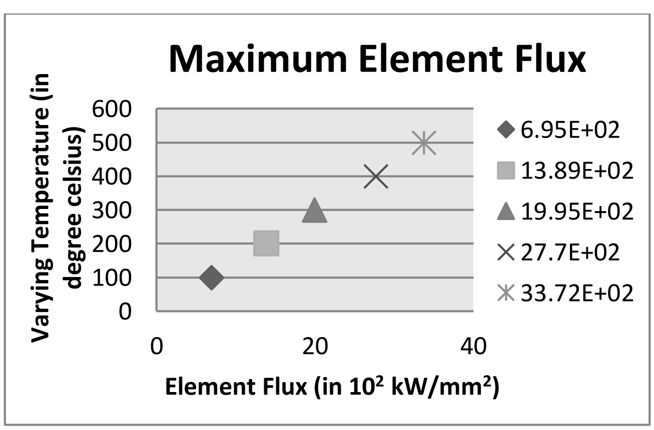

Figure 6: Maximum Element Flux against temperature variations

CONCLUSION

From this study an attempt is made to obtain the variation of temperature, element temperature gradient and element flux within the shell containing pipes. These parameter varies in accordance with the variation in the air blown in the shell. The results gives a clear idea about the variations which can be summarized & concluded to give a linear steady state thermal analysis for simple tubular heat exchanger.

The heat transfer rate is poor because of less no of pipes used in side shell. Thus the design can be modified for better heat transfer in two ways either the decreasing the shell diameter, so that it will be in a proper contact with the inner tubes or by increasing the no of pipes, thus tubes will be proper contact with the shell. It is because the heat transfer area is not utilized efficiently in less no of tubes.

REFERENCES

[1] Hetal Kotwal & D.S. patel "CFD Analysis of Shell and Tube Heat Exchanger-A Review" International Journal of Engineering Science and Innovative Technology (IJESIT) ISSN: 2319-5967 ISO 9001:2008 Certified Volume 2, Issue 2, March 2013

[2] Sunil-kumar Shinde, Mustansir Hatim Pancha "Comparative Thermal Performance Analysis Of Segmental Baffle Heat Exchanger with Continuous Helical Baffle Heat Exchanger using Kern method" Journal of Engineering Research and Applications (IJERA)" ISSN: 2248-9622 www.ijera.com Vol. 2, Issue4, July-August 2012, pp.2264-2271 2264

[3] M. Thirumarimurugan, T.Kannadasan and E.Ramasamy, Performance Analysis Of Shell And Tube Heat Exchanger Using Miscible System, American Journal Of Applied Sciences 5 (5): 548-552, 2008.

[4] K. Sudhakara Rao, Analysis Of Flow Mal distribution In Tubular Heat Exchangers By Fluent, National Institute Of Technology Rourkela ,2007.

[5] Khairun Hasmadi Othman, CFD Simulation of Heat Transfer In Shell And Tube Heat Exchanger, University Malaysia Pahang, April 2009.

[6] Prithiviraj, M., and Andrews, M. J, Three Dimensional Numerical Simulation of Shell-and-Tube Heat Exchangers, Foundation and Fluid Mechanics, Numerical Heat Transfer Part A - Applications, 33(8), 799–816, (1998).

[7] Serth Robert W., Process heat transfer principles and application, ISBN: 0123735882, 246-270, (2007).

[8] Master B.I, Chunagad K.S. Boxma A.J. Kral D, Stehlik P, Most frequently used Heat exchangers from pioneering Research to Worldwide Applications, vol. No.6, (2006), 1-8.

[9] Kern DQ. Process heat transfer. New York (NY): McGraw-Hill; 1950.

[10] Avinash D Jadhav, Tushar A Koli "CFD Analysis of Shell and Tube Heat Exchanger to Study the Effect of Baffle Cut on the Pressure Drop" International Journal Of Research In Aeronautical And Mechanical Engineering ISSN(Online): 2321-3051Proceedings of the National Conference on Trends and Advances in Mechanical Engineering, YMCA University of Science & Technology, Faridabad, Haryana, Oct 19-20, 2012 pp. 167

[11] Abdur Rahim, S.M.Saad Jameel "Shell Side Cfd Analysis Of A Small Shell-And-Tube Heat Exchanger Considering The Effects Of Baffle Inclination On Fluid Flow"

[12] Mukherjee, R., Practical Thermal Design of Shell-and-Tube Heat Exchangers, Begell House.Inc, New York, 2004.

[13] Usman Ur Ehman , Goteborg, Sweden 2011, Master's Thesis 2011:09 on "Heat Transfer Optimization of Shell-and-Tube & Heat Exchanger through CFD".

[14] Khairun Hasmadi Othman, " CFD simulation of heat transfer in shell and tube heat exchanger", A thesis submitted in fulfillment for the award of the Degree of Bachelor in chemical Engineering (Gas Technology),April 2009.