IJESRT
INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY

ANALYSIS OF MULTIPLE SHELL AND TUBE HEAT EXCHANGER BY VARYING THE INPUT PARAMETERS COMPUTATIONAL FLUID DYNAMICS

Pavan Vishwakarma*, Jitendra Jayant
* M-Tech Scholar, Department of Mechanical Engineering, Lakshmi Narayan College of Technology, Indore, MP, India
Associate Professor, Department of Mechanical Engineering, Lakshmi Narayan College of Technology, Indore, MP, India

DOI: 10.5281/zenodo.802830

ABSTRACT

A heat exchanger is a device that is used to transfer thermal energy (enthalpy) between two or more fluids, between a solid surface and a fluid, or between solid particulates and a fluid, at different temperatures and in thermal contact. In heat exchangers, there are usually no external heat and work interactions. The aim of the present dissertation work focus on the design and thermal analysis of multiple shell and tube heat exchanger. Heat exchanger designed, justifying the phenomenon of convection within the shell and tubes. The study presents the linear static-steady thermal analysis of a multiple shell and tube heat exchanger using computational fluid dynamics approach. Altair HYPERMESH software is used to perform the analysis. The number of shell tubes, shell tubes arrangements at various angle, and temperature were considered as input parameters.

KEYWORDS: Hypermesh, Multiple Shell and Tube Heat Exchanger, Tube Heat Exchanger, CFD, Parallel Flow.

INTRODUCTION

In the recent years, there is increasing need of highly efficient heat exchangers. This project primarily focuses on designing and simulation of tubular heat exchanger using Computational fluid Dynamics through ANSYS FLUENT (CFD) as software tool. The major relevant literature has been studied in order to summarize the major approaches and design available at the present stage. The studied review of literature directed this dissertation work to design the tubular heat exchanger. This main emphasis to design the simple tubular heat exchanger is to simulate it for the particular parameters and the heat transfer rate and heat transfer coefficient can be analysed. Thus this study provides a numerical analysis ofheat transfer coefficient for the simple tubular heatexchanger.

Heat exchange between flowing fluids is one of the most important physical process of concern, and a variety of heat exchangers are used in different type of installations, as in process industries, compact heat exchangers nuclear power plant, HVACs, food processing, refrigeration, etc. The purpose of constructing a heat exchanger is to get an efficient method of heat transfer from one fluid/gas to another, by direct contact or by indirect contact. The heat transfer occurs by three principles: conduction, convection and radiation. In a heat exchanger the heat transfer through radiation is not taken into account as it is negligible in comparison to conduction and convection. Conduction takes place when the heat from the high temperature fluid flows through the surrounding solid wall. The conductive heat transfer can be maximized by selecting a minimum thickness of wall of a highly conductive material. But convection is plays the major role in the performance of a heat exchanger. Forced convection in a heat exchanger transfers the heat from one moving stream to another stream through the wall of the pipe. The cooler fluid removes heat from the hotter fluid as it flows along or across it

Tubular heat exchangers are built of mainly of circular tubes although some other geometry has also been used in different applications. This type of construction offers a large amount of flexibility in design as the designing parameters like the diameter, length and the arrangement canbe easily modified. This type is used for liquid-to-liquid (phase changing like condensing or evaporation) heat transfer. Again this type is classified into shell and tube, double pipe and spiral tube heat exchangers.

OBJECTIVES OF WORK

The design of tubular heat exchanger has been facing problems because of the lack of experimental data available regarding the behaviour of the fluid in helical coils & tubes and also incase of heat transfer data, which is not the case in Shell & Tube Heat Exchanger. So to thebest of our effort, thermal analysis was carried out to determine the heat transfer characteristics for a tubular heat exchanger by varying the different profiles like different temperatures and diameters of pipe and coil. The objective of the project is to obtain a better and more quantitative insight into the heat transfer process that occurs when air flows in thetube.

METHODOLOGY

ProblemDescription:

Design and analysis of simple tubular heat exchanger with number of varying pipes inside the heat exchanger using hyper mesh software. To study the temperature variation inside the heat exchanger tube with different mass flow rate**.**

ComputationModel:

The computation model of an analysis test tubular heat exchanger with the varying number of pipe is shown in figure and the geometry parameters are listed in table below [21]. As shown in figure there are five and nine number of tubes inside the tubular heat exchanger. The wholecomputation domain is bounded inside of the tube and everything inside the tube contained in the domain. This IGES file of the model is imported from CAD software.

Some of the basic characteristics of the process to simplify the simulation assumption are made as follows.

- The tube side fluid has constant thermalproperties.
- The heat transfer process and fluid flow are in steady statecondition.
- The leak between tube and shell areneglected.

The heat exchanger is insulated so that heat loss to the environment is completely neglected.

Ansys fluent:

Ansys Fluent is a commercially finite element pre-Processor and post-processor that quick creates finite element and finite difference models foe engineering analysis. Hyper mesh using the followingprocess:

- Import and translate surface geometry from externalfile.
- Create collectors andcomponent,
- Build the model by buildingelement.
- Mesh the modeland
- Check the mesh qualitygeometry.

It also provides CAD surface manipulations, of particular interest is the ability of hyper mesh to do automatic mesh generation and surface stitching. The organisation of mesh generation algorithms, algorithms parameter and smoothing operation are available to improve grid quality. The available mesh generation algorithms are divided into two types:

- Those that require the presence of asurface.
- Those that working purely from node or linedata.

The smoothing algorithms are categorized as size pr shape corrected algorithms. The size correcting smoothing uses a modified Laplacian over- relaxation. The shape correction smoothing algorithms employs iso-parametric-centroidal over-relaxation that corrects the elements shape allowing variation in element size.

Gridformation

The three dimension model is then discretized in hyper mesh. In order to capture the thermal boundary layer the entire model is discretized using Quadra-hedral mesh element which is precise and involves less computation effort. Fine control of the Quadra-hedral mesh near the wall surface allows capturing the boundary layer gradient accurately. The entire geometry is divided into three domains Air inlet, air tube shell, and five and nine number of pipes.

The heat exchanger is discretized into solid in order to have better to control over the number of nodes. The mesh is made is made finer for simulating conjugate heat exchanger phenomenon. The fluid domain from the tube surface is maintained at 100 microns to capture the thermal boundary layer. The discretized model is checked for quality and ifit once found free from error and minimum required quality it is exported to HYPRMESH Pre-processor.

MeshGeneration

Meshing generation is an integral part of the computer-aided engineering analysis process. The mesh influences the accuracy, convergence and speed of the solution. Furthermore, the time taken to create a mesh model often consumes the significant portion of the project duration of simulation for the complex components. Therefore, the more the automated meshing tools, faster the convergence of equation residuals and better the solution. The mesh generation process in this system has started with creation of 2D quad elements in the diffuser and partingplanes. Initially a relatively coarser mesh is generated with 1.8 Million cells. This mesh contains cells of Quadra-hedral faces at the boundaries. Care is taken to use structured cells as much as possible, for this reason the geometry is divided into several parts for using automatic methods available in the Hyper mesh meshing client. It is meant to reduce numerical diffusion as much as possible by structuring the mesh in a well manner, particularly near the wall region. Later on, for the mesh independent model, a fine mesh is generated with 5.65 Million cells. For this fine mesh, the edges and regions of high temperature and pressuregradients are finely meshed. A high quality mesh is very important for successful numerical simulations. The smaller the size of the element near the wall of the tube and the slot, the more detailed and accurate flow structure will be captured. However, for the 3D simulation, a small change in the size of element will lead a substantial increase in the number of elements. That will results in a significant increase of computational time. In order to balance the accuracy of the simulations and CPU time, an optimum size of mesh need to be chosen. “Symmetry” boundary condition is imposed on the velocity field at $y = 0$ mm and $z = 0$ mm.The computational domain is divided into two zones: the inlet and tube bank zone and the exit zone. Data are transferred between the two zones by the connective interface. Non uniform meshing is used to discretize the equations in the inlet and tube bank zone. An inflation method is applied at the wall of tubes. The boundary layers on walls of tubes, upper and lower bounds are refined by putting several inflation layers. A coarser non uniform mesh is employed in the exit region. At the interface between the two zones, contact sizing is applied for coordinating the finer mesh in the inlet and tube bankzone by adjusting the global size of mesh element, contact size at the interface and inflation layers near the wall of the tubes, three meshes with different magnitudes of elements in the tube bank zone are generated [23]. The mesh near the wall of the tubes and near the slots is dense. With the use of zone meshing, the mesh in the tube zone can be refined with a minimal increase of CPU time. That enables us to obtain very accurate 3-D flow structure in the tube zone.

RESULT AND DISCUSSION

The simulation is carried out with setting all the boundary conditions using Ansys Fluent (as a simulation tool – CFX) 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 ranges from 300 to 500 K. Also the study in this dissertation is focused on varying the no. of pipes inside the shell. Three different plots of temperature profile are taken into account for the comparison to analyze the heat transfer rate of simpler tube heat exchanger. Here a comparative study is generated where number of pipes varies as $N = 4$ and $N = 6$. Air is blown at five temperatureranges.

The main objective of the present work is to identify the effects of inlet hot fluid temperature at four levels 300-400 K with and equal interval of 25 K, the value of inlet cold temperature is kept constant (i.e. 75, 100 K) during the each analysis (iterations). Effects of inlet hot fluid temperature was analyzed on various output responses such as; (i) pressure distribution; (ii) temperature distribution; (iii) density distribution; (iv) turbulence; (v) enthalpy and (vi) entropy of the shell and tube type heat exchanger. This Analysis will help in predicting the life and in reducing the chances of failure of Shell and tube type heat exchanger, which is one of the most ignorant areas of research in shell and tube type heat exchanger (STHE).

To conduct this study computational fluid dynamics tool CFX of Ansys has been used.

Set-I of the Experiments: First set of the experiments have been performed using inlet cold temperature constant at 75 K while varying the inlet hot fluid temperature in the range of 300-375 K. Four iterations have been done and presented in the next subsequent subsections as Iteration I-IV.

Iteration-I

In the first iteration of set-I, inlet temperature (K) of cold fluid was kept constant at 75K, while inlet temperature of the hot fluid has been varied at five levels from 300K respectively. Responses have been measured in terms of (i) pressure distribution; (ii) temperature distribution; (iii) density distribution; (iv) enthalpy and (v) entropy of the shell and tube type heat exchanger. Figure 6.1 presents the variation in the measured responses of the shell and tube type heat exchanger.

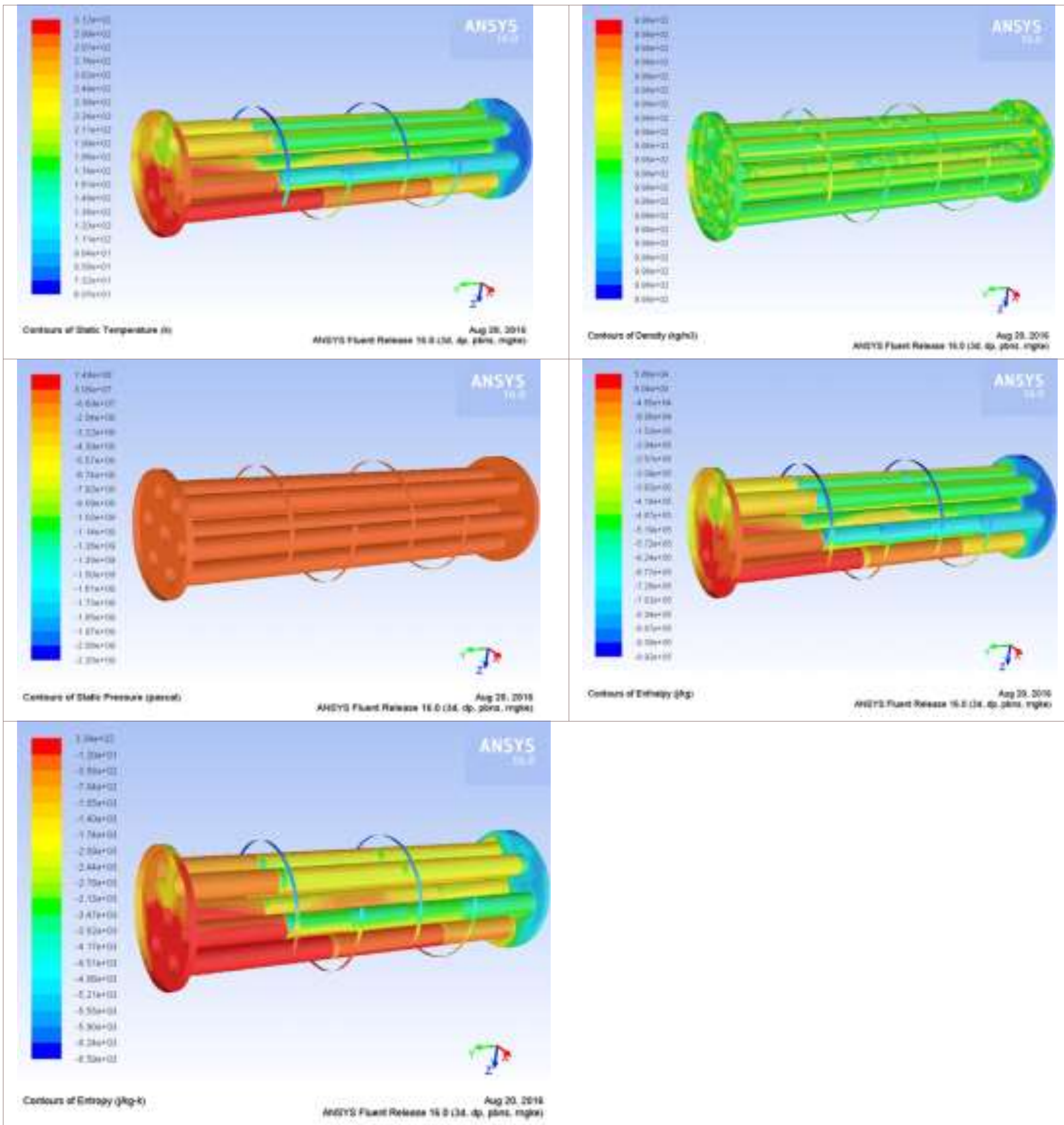

Fig. 6.1 Variations in the considered responses at inlet cold temperature 75 K and inlet hot temperature 300K

Iteration-II

In the second iteration of set-I, inlet temperature (K) of cold fluid was kept constant at 75K, while inlet temperature of the hot fluid has been varied at five levels from 325K respectively. Responses have been measured in terms of (i) pressure distribution; (ii) temperature distribution; (iii) density distribution; (iv) enthalpy and (v) entropy of the shell and tube type heat exchanger. Figure 6.2 presents the variation in the measured responses of the shell and tube type heat exchanger.

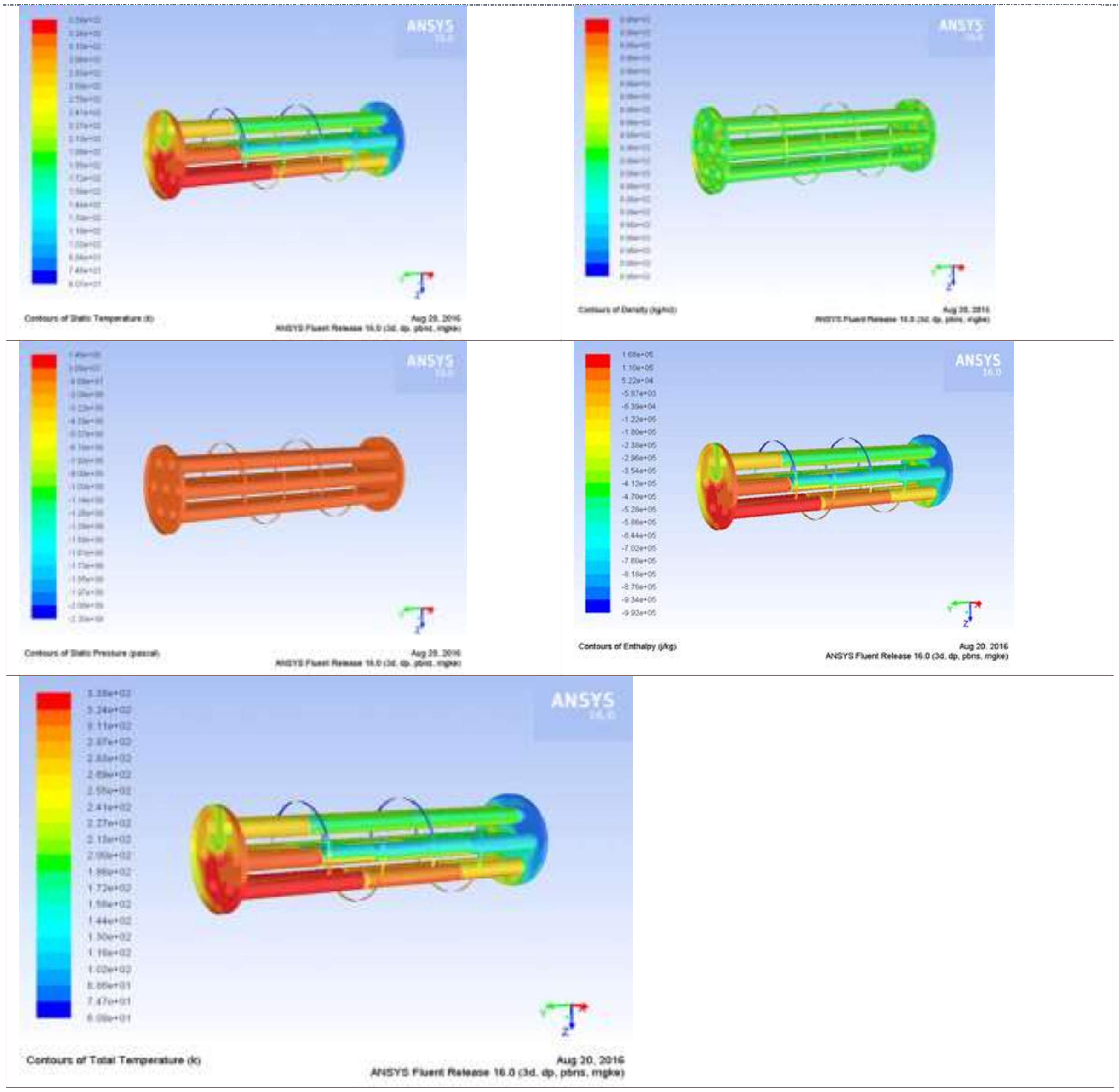

Fig. 6.2 Variations in the considered responses at inlet cold temperature 75 K and inlet hot temperature 325K

Iteration-III

In the third iteration of set-I, inlet temperature (K) of cold fluid was kept constant at 75K, while inlet temperature of the hot fluid has been varied at five levels from 350K respectively. Responses have been measured in terms of (i) pressure distribution; (ii) temperature distribution; (iii) density distribution; (iv) enthalpy and (v) entropy of the shell and tube type heat exchanger. Figure 6.3 presents the variation in the measured responses of the shell and tube type heat exchanger

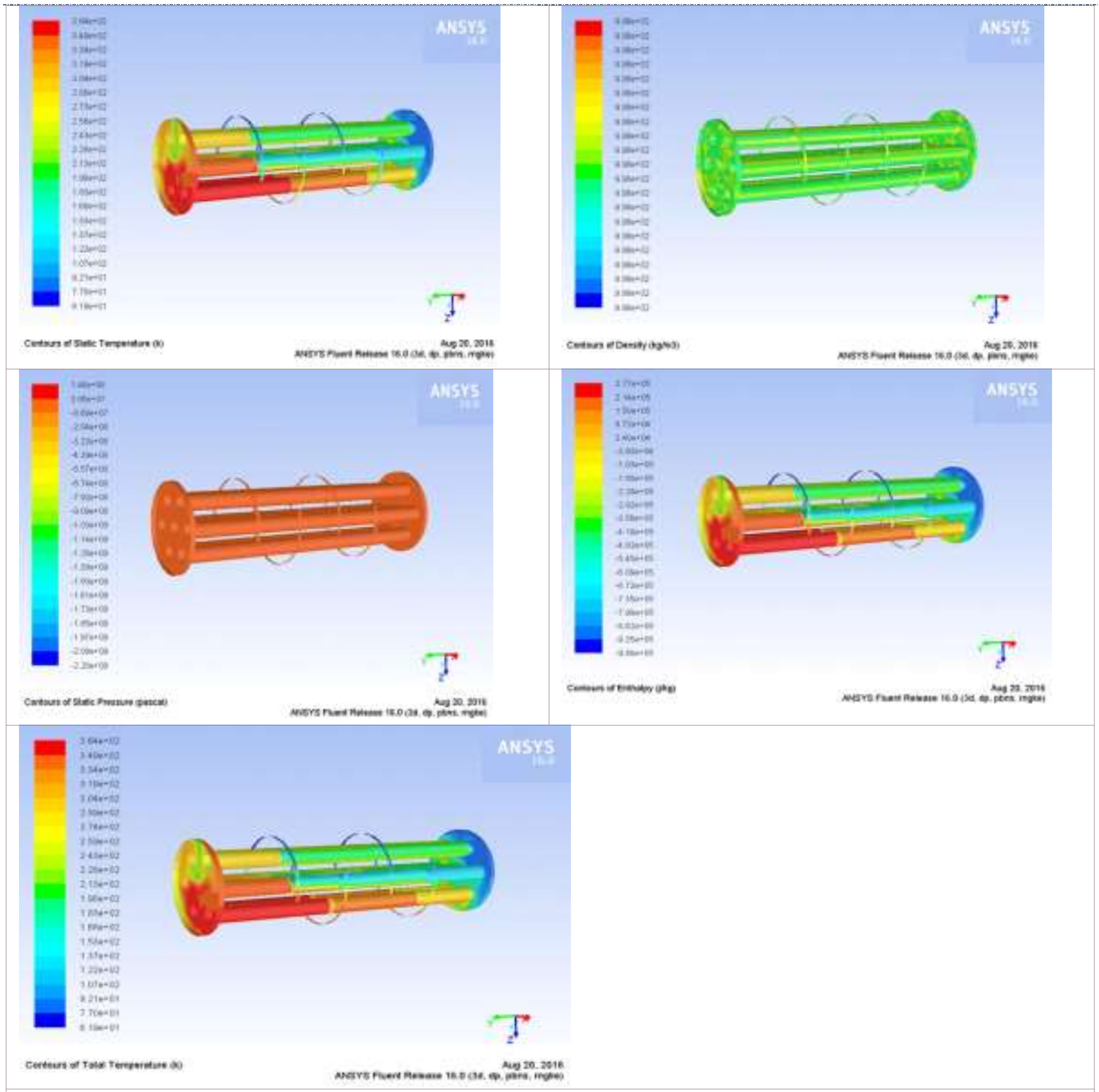

Fig. 6.3 Variations in the considered responses at inlet cold temperature 75 K and inlet hot temperature 350K

Iteration-IV

In the fourth iteration of set-I, inlet temperature (K) of cold fluid was kept constant at 75K, while inlet temperature of the hot fluid has been varied at five levels from 375K respectively. Responses have been measured in terms of (i) pressure distribution; (ii) temperature distribution; (iii) density distribution; (iv) enthalpy and (v) entropy of the shell and tube type heat exchanger. Figure 6.4 presents the variation in the measured responses of the shell and tube type heat exchanger.

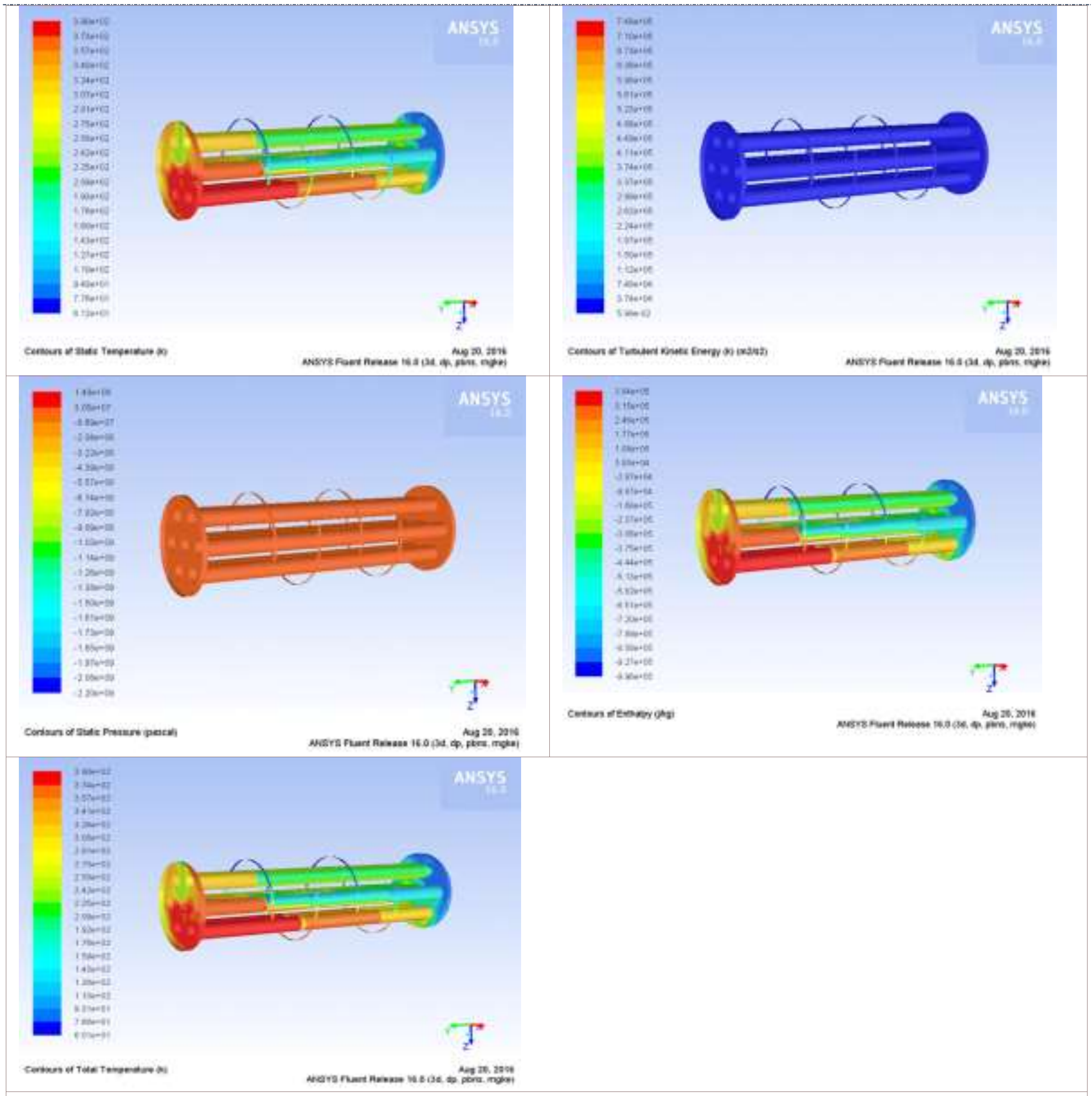

Fig. 6.4 Variations in the considered responses at inlet cold temperature 75 K and inlet hot temperature 375K

Set-II of the Experiments: Second set of the experiments have been performed using inlet cold temperature constant at 100 K while varying the inlet hot fluid temperature in the range of 300-375 K. Four iterations have been done and presented in the next subsequent subsections as Iteration I-IV.

Iteration-I

In the first iteration of set-II, inlet temperature (K) of cold fluid was kept constant at 100K, while inlet temperature of the hot fluid has been varied at five levels from 300K respectively. Responses have been measured in terms of (i) pressure distribution; (ii) temperature distribution; (iii) density distribution; (iv) enthalpy and (v) turbulence of the shell and tube type heat exchanger. Figure 6.5 presents the variation in the measured responses of the shell and tube type heat exchanger.

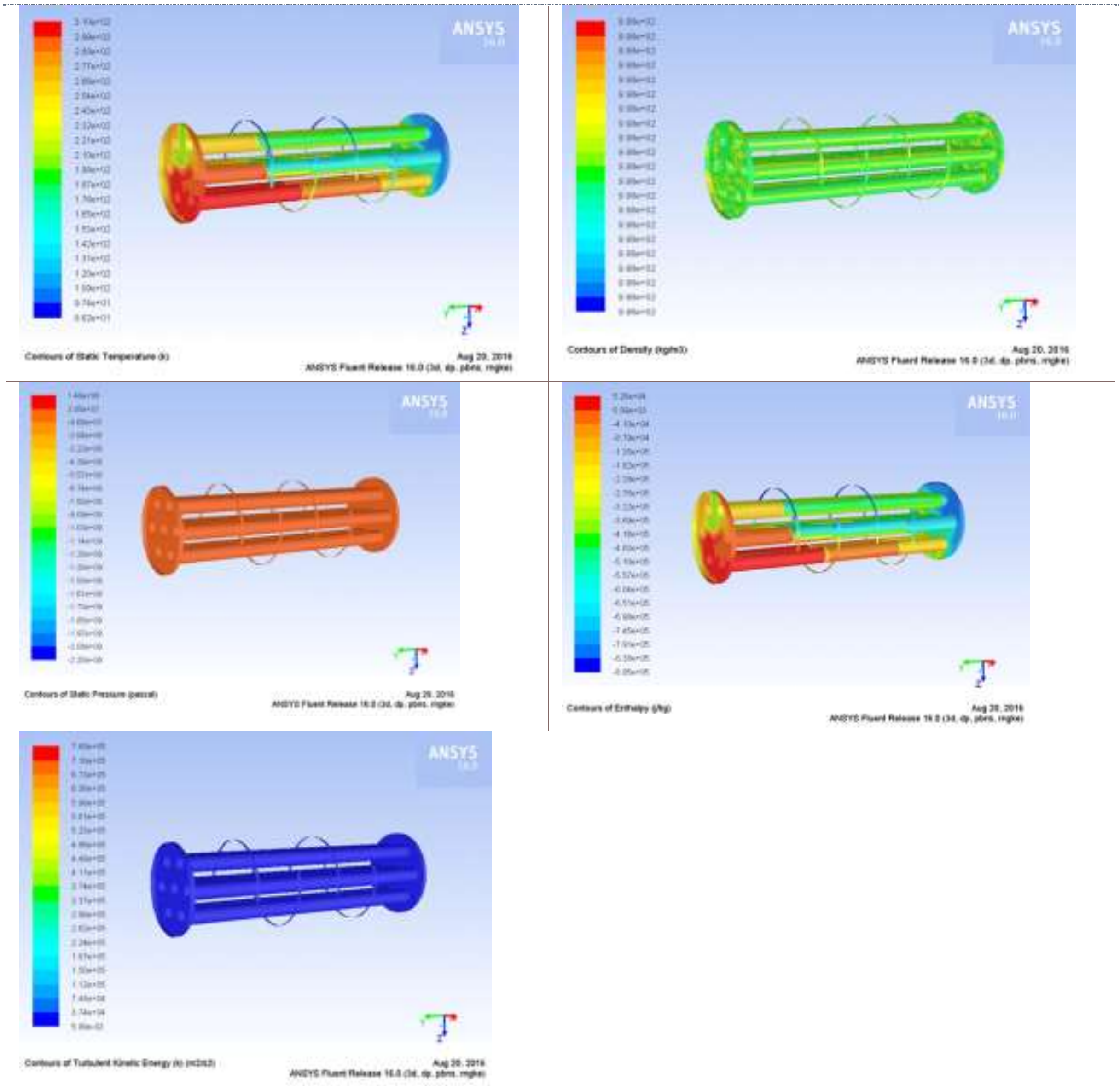

Fig. 6.5 Variations in the considered responses at inlet cold temperature 100 K and inlet hot temperature 300K

Iteration-II

In the second iteration of set-II, inlet temperature (K) of cold fluid was kept constant at 100K, while inlet temperature of the hot fluid has been varied at five levels from 325K respectively. Responses have been measured in terms of (i) pressure distribution; (ii) temperature distribution; (iii) density distribution; (iv) enthalpy and (v) turbulence of the shell and tube type heat exchanger. Figure 6.6 presents the variation in the measured responses of the shell and tube type heat exchanger.

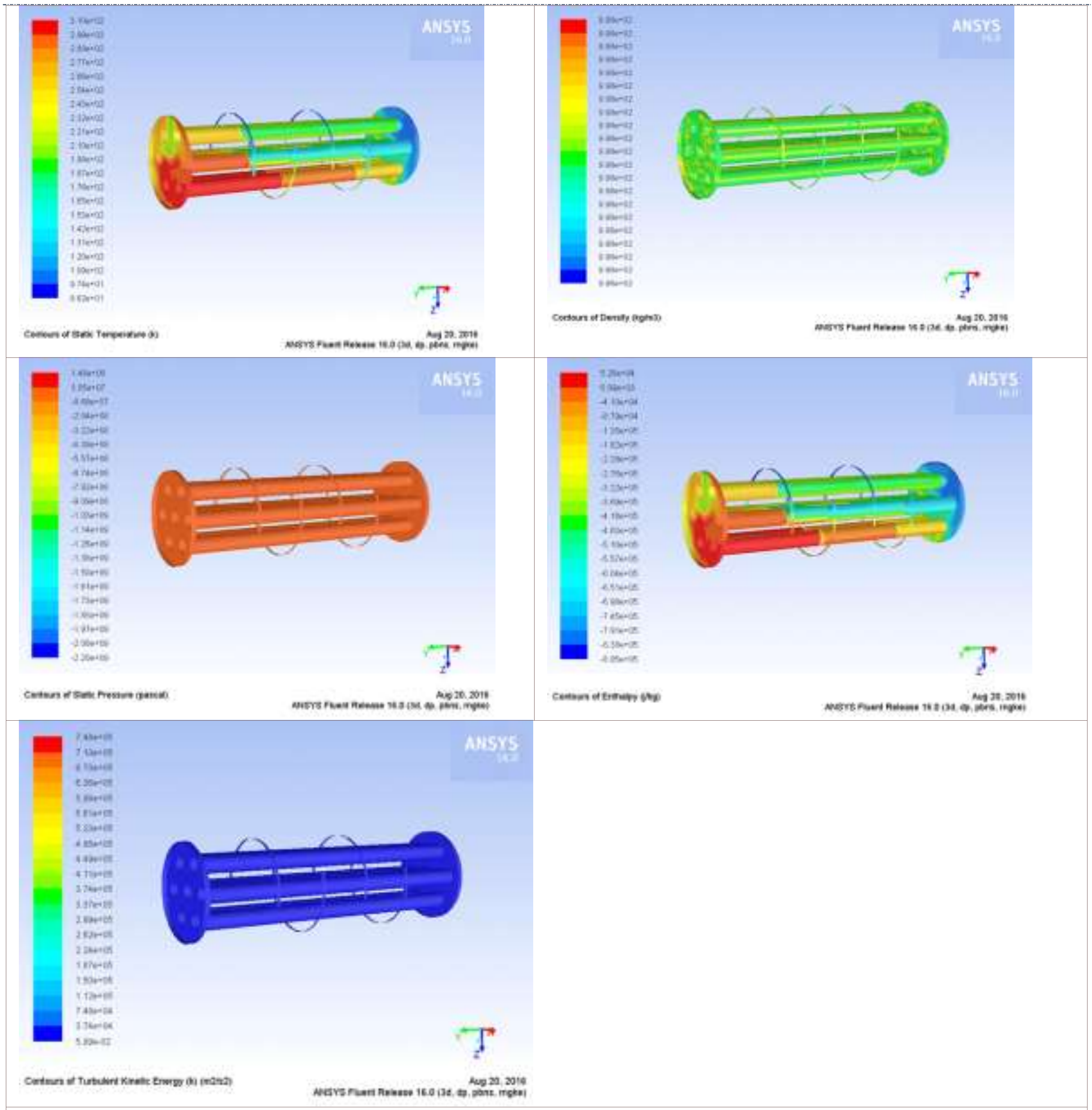

Fig. 6.6 Variations in the considered responses at inlet cold temperature 100 K and inlet hot temperature 325K

Iteration-III

In the third iteration of set-II, inlet temperature (K) of cold fluid was kept constant at 100K, while inlet temperature of the hot fluid has been varied at five levels from 350K respectively. Responses have been measured in terms of (i) pressure distribution; (ii) temperature distribution; (iii) density distribution; (iv) enthalpy and (v) turbulence of the shell and tube type heat exchanger. Figure 6.6 presents the variation in the measured responses of the shell and tube type heat exchanger.

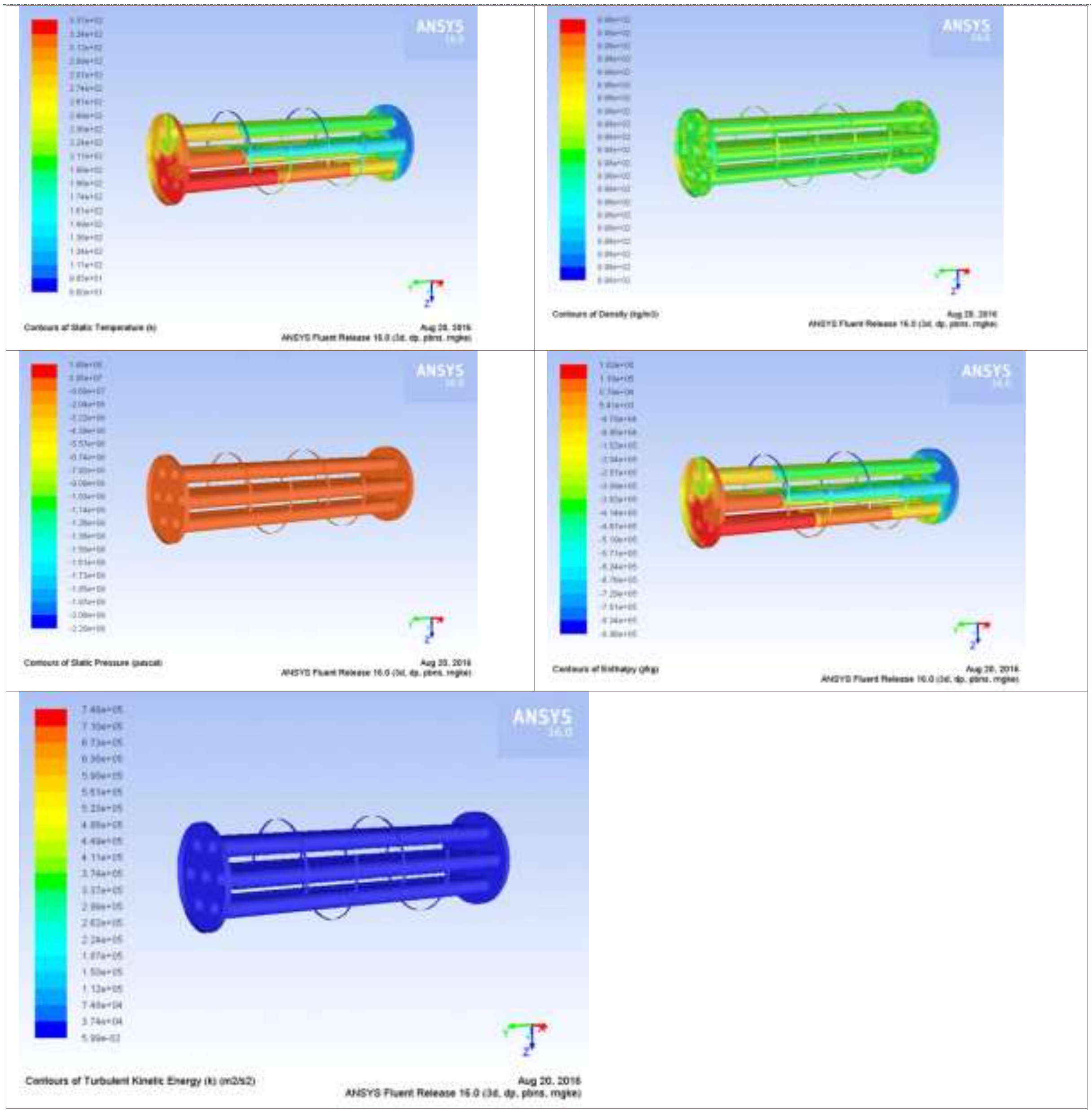

Fig. 6.6 Variations in the considered responses at inlet cold temperature 100 K and inlet hot temperature 350K

Iteration-IV

In the fourth iteration of set-II, inlet temperature (K) of cold fluid was kept constant at 100K, while inlet temperature of the hot fluid has been varied at five levels from 375K respectively. Responses have been measured in terms of (i) pressure distribution; (ii) temperature distribution; (iii) density distribution; (iv) enthalpy and (v) turbulence of the shell and tube type heat exchanger. Figure 6.6 presents the variation in the measured responses of the shell and tube type heat exchanger.

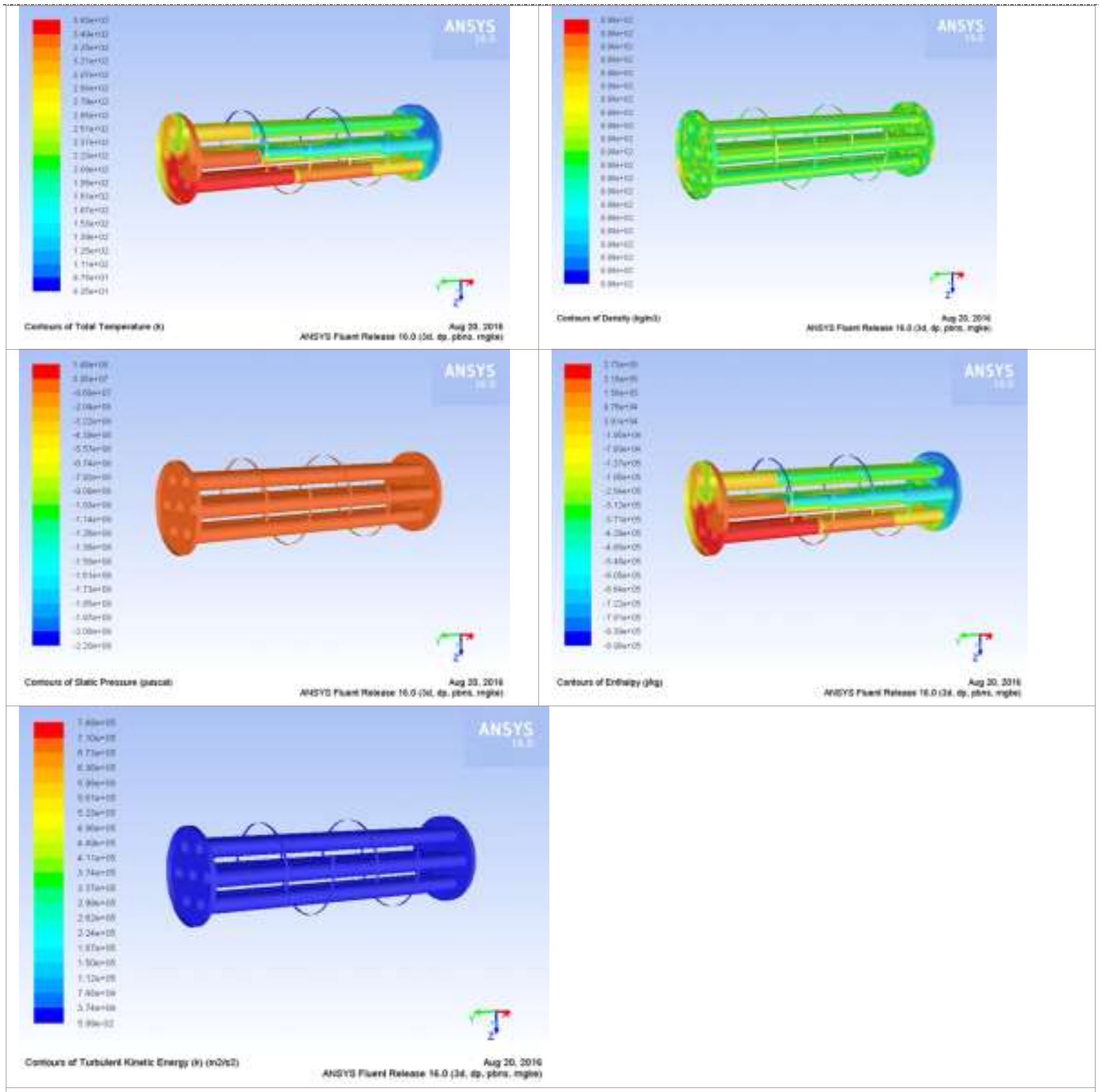

Fig. 6.9 Variations in the considered responses at inlet cold temperature 100 K and inlet hot temperature 375K

CONCLUSION

The study presents the linear static-steady thermal analysis of a multiple shell and tube heat exchanger using computational fluid dynamics approach. Altair HYPERMESH software is used to perform the analysis. The number of shell tubes, shell tubes arrangements at various angle, and temperature were considered as input parameters.The design of tubular heat exchanger has been facing problems because of the lack of experimental data available regarding the behaviour of the fluid in helical coils & tubes and also incase of heat transfer data, which is not the case in Shell & Tube Heat Exchanger. So to thebest of our effort, thermal analysis was carried out to determine the heat transfer characteristics for a tubular heat exchanger by varying the different profiles like different temperatures and diameters of pipe and coil.

REFERENCES

[1] Ender Ozden, Ilker Tari, "*Shell Side CFD Analysis of A Small Shell and Tube Heat Exchanger",* Middle East Technical University, 2010.

[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.

[3] Daniel Flórez-Orrego et.al "Experimental and CFD study of a single phase cone-shaped helical coiled heat exchanger: an empirical correlation" ECOS June 26-29, 2012.

[4] J.S.Jayakumar "Helically Coiled Heat Exchangers and J.S. Jayakumar et.al "Experimental and CFD estimation of heat transfer in helically coiled heat exchangers" Chemical Engineering Research and Design, 2008 Pg. 221-232.

[5] Timothy John Rennie "*Numerical and Experimental Studies of a Double pipe Helical Heat Exchanger*" Dept. of Bio-resource Engineering. McGill University, Montreal August 2004.

[6] 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

[7] Dr. K. E. Reby Roy et al. "*Numerical Analysis of Forced Convective Heat Transfer through Helical Channels*" International Journal of Engineering Science and Technology (IJEST) ISSN: 0975-5462 Vol. 4 No.07 July 2012 Pg.3352-3359

[8] Ashok B. Korane, P.S. Purandare and K.V. Mali "*Heat Transfer Analysis of Helical Coil Heat Exchanger with Circular and Square Coiled Pattern*" IJESR, June 2012, vol-2, issue-6.

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

[10] Siddhartha Shankar Behera "*CFD analysis of heat transfer in a helical coil heat exchanger using fluent*" a dissertation work presented at NIT Rourkela in MECHANICAL Department, May 2013.

[11] Anil kumar samal "*Shell and tube heat exchanger design using CFD"* a dissertation work presented at NIT Rourkela in chemical Department,2013.

[12] Haolin Ma, "*Computational Fluid Dynamics and Heat Transfer Analysis for a Novel Heat Exchanger*" dissertation work presented in mechanical engineering at Lehigh University, October 2012

[13] Khairun Hasmadi Othman, " CFD simulation of heat transfer in shell and tube heat exchanger", A dissertation in chemical Engineering (Gas Technology),April 2009.

CITE AN ARTICLE

Vishwakarma, P., & Jayant, J. (2017). ANALYSIS OF MULTIPLE SHELL AND TUBE HEAT EXCHANGER BY VARYING THE INPUT PARAMETERS COMPUTATIONAL FLUID DYNAMICS. *INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY*, *6*(6), 72-83. doi:10.5281/zenodo.802830