A RESEARCH ON VISUALIZATION OF VELOCITY, PRESSURE AND TEMPERATURE DISTRIBUTION IN A CROSS-FLOW TYPE CASE AND CHAMBER HEAT EXCHANGER USING CFD TOOLS.

Tushar Das
Student, Master of Engineering, Department of Mechanical Engineering, Chandigarh University Gharuan punjab 140413
Amman Jakhar
Assistant Professor, Department of Mechanical Engineering, Chandigarh University Gharuan Punjab 140413

ABSTRACT
This proposition presents study and assessment of execution and optimal arrangement of power exchangers and force exchangers association. The survey consolidates a simulational assessment of fluid stream in case and chamber heat exchanger, premium arrangement of case and chamber heat exchanger and single pass case and chamber heat exchangers, and progression of force exchanger associations. In the simulational assessment of fluid stream in case and chamber heat exchanger a couple of extraordinary limits would be participated to get various plots of entertainment. Considering simulational data, connections for separating and finding out heat move coefficients and strain diminishes , temperature change and speed points have been gained with great accuracy. This exploration paper centers around the representation of different boundaries, for example, speed, strain and temperature varieties along in a cross stream type case and chamber heat exchanger.

KEYWORDS: Shell and tube heat exchanger, scientist, engineer, case and chamber heat exchanger, parallel flow heat exchanger, counter flow heat exchanger, cross flow heat exchanger, computational fluid dynamics, ansys fluent, ansys workbench, cost effective, geometry, mesh, setup, solutions, results.

1.INTRODUCTION

This research paper represents a collective effort to make many ways to work with case and chamber heat exchangers. The report represents a documentation of the master work executed by Tushar das at the department of mechanical engineering at Chandigarh university. The purpose of this research work is to model shell and tube heat exchangers. Various models have been developed by scientists and engineers, Thus the model with which the work has been prepared, is unique in its design and topology. Also care has been taken such that the robustness of the geometry used to model the case and chamber heat exchanger running to its full capacity. Different models of heat exchanger are available in literature, but in this work the case and chamber heat exchanger has been taken into account. A comparison has been made with the different types of materials with which the case and chamber has been constructed. The materials used for solid building blocks are copper and aluminum. The material used for liquid flow is taken as water-liquid. Solid materials are purposely taken from aluminum and copper since these are anti resistant to corrosion from fluid used as liquid water. Further the mode is chosen as coupled and second order upwind for both the kinetic and static analysis purpose. The objective of the work was generic modeling of case and chamber heat exchanger,and the flexibility is left with the users who are using the models to perform analysis in real time scenarios. With the help of simulation software various types of case and chamber heat exchanger can be modeled with ease and accuracy.

There are several types of case and chamber heat exchanger that can be analyzed but for the purpose of this work only two types of heat exchangers have been analyzed. These are counter flow heat exchanger and parallel flow heat exchanger. In parallel flow heat exchanger both the hot fluid and cold fluid enters from the same side of the heat exchanger and leaves from the same side of the heat exchanger, and thus the inlet and outlet of the parallel flow heat exchanger are present on the same side respectively also the fluids flow parallel to each other in the same direction, whereas in the case of counter flow heat exchanger the entry and the exits of the hot and cold fluid are present at the opposite ends of the geometry. Thus the fluids flow opposite to

each other in an opposite direction. There is another type of arrangement present in the case and chamber heat exchanger, that is cross flow heat exchanger.
. Thus this type of heat exchanger is not discussed in the coming literature. During the simulation the inlet velocities are varied so as to obtain a diverse set of results while the simulation is performed. Similarly in the case of temperature of both the inlet and the outlet is varied so that the simulations give us a diverse range of results depending on the range of temperature and velocity taken individually and in combination with each other.

1.1 Types of case and chamber heat exchanger.

There are basically three types of case and tube heat exchanger. These are cross flow , counter flow, and parallel flow case and chamber heat exchanger. In cross flow heat exchanger the two working fluid flow past each other in a right angled fashion, i.e. they flow past each other at 90 degrees to each other, as a result heat exchange takes place such that the warm fluid releases heat energy which is absorbed by the fluid which is at a lower temperature and hence heat transfer takes place. In a countercurrent heat modifier the two working fluid flow past each other in a opposite fashion i.e. the side at which the warm fluid enters is the side at which the fluid at a lower temperature exits the device, similarly the side at which the fluid at a higher temperature exits the device is the side at which the fluid at a lower temperature enters the device. The heat transfer takes place by the exchange of heat from the fluid at a higher temperature to the fluid at a lower temperature arranged in a counter flow arrangement. In case of parallel flow heat modifier, the side at which the fluid at a higher temperature enters the device is the side at which the fluid at a lower temperature enters the device, similarly the fluid which is ata higher temperature exits the device is the same side at which the fluid at lower temperature exits the device. The heat transfer takes place from the fluid which is at a higher temperature to the fluid which is at a lower temperature. And thus in this way heat exchanger devices are classified to the arrangement of the cold and the warm fluid.

2. METHODS

2.1 Ansys Fluent

The software and its hardware dependencies play an important role in running the modeling and simulation works with better ease and accuracy. Initially when analysis needed to be done and results needed to be plotted the large scale models used in industries were used to produce real lifelike results. This method of analyzing the model and interpretation of the results would be very difficult since these real life objects showed a variation in behavior when the experiments were run multiple times to obtain results. The results obtained in each case showed a variation from the results in the previous case.
This variation in results would lead to lower accuracies and difficulty in making and finding the final conclusions. The final conclusions would not be easily made since the large changes in output would require the analyst to again perform the experiments. The variation needed to be taken into account while operating these real lifelike objects in this case: case and chamber heat exchanger. The case and chamber heat exchanger alternatively known as shell and tube heat exchanger is the basic focus of this work, hence we limit the discussion upto only these types of heat exchangers. This case and chamber heat exchanger performs well when its length is large as compared to its diameter,in this case the diameter of the shell. This is because a high aspect ratio results in an efficient running of the heat exchanger, which results in better results and accuracy though it would incur certain costs in running and actuating these large exchangers.
With the help of modeling and simulation software and its required hardware dependencies one can easily perform the simulation of such large machines at a fraction of a cost with much needed accuracy. With the help of modeling and simulation software and its required hardware dependencies one can easily perform the simulation of such large machines at a fraction of a cost with much needed accuracy. The modeling and simulation software used in this work is Ansys

software. Ansys is a major software company which provides simulation solutions with greater ease and accuracy as compared to its counterparts. Further the package that is used while making the work is the "FLUENT" package. The fluent package is basically used for stream flow modeling and simulation. The fluent package is a CFD tool to analyze and visualize various flow field properties.

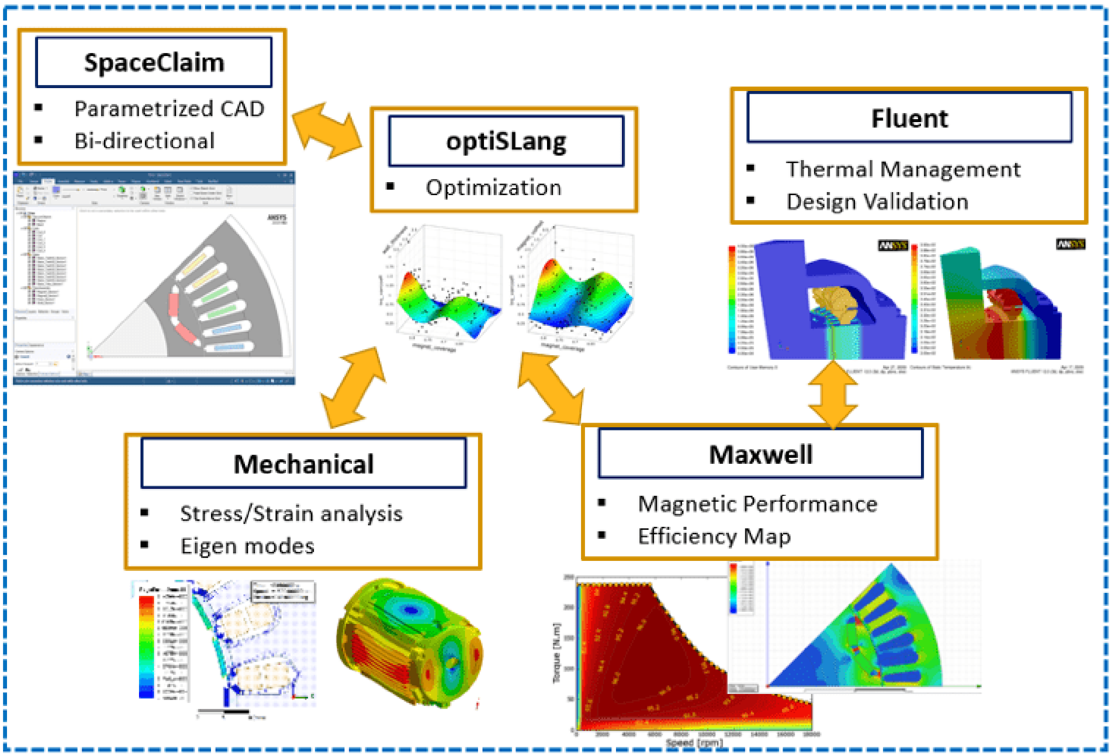

Figure 1 shows the stages of ansys fluent software architecture

A standalone project is created. While the standalone project is created there are multiple options shown in the standalone project. These involve Geometry, mesh, setup, solutions, results. Each option has its own role to play. Now by further manipulating the standalone project a modeling and simulation project can be simulated, i.e. by operating those conditions under the prescribed format. With the help of modeling and simulation software and its required hardware dependencies one can easily perform the simulation of such large machines at a fraction of a cost with much needed accuracy. The modeling and simulation software used in this work is Ansys software. Ansys is a major software company which provides simulation solutions with greater ease and accuracy as compared to its counterparts. Further the package that is used while making the work is the "FLUENT" package. The fluent package is basically used for stream flow modeling and simulation. The fluent package is a CFD tool to analyze and visualize various flow field properties.

3. RESULTS

3.1 Pressure distribution visualization

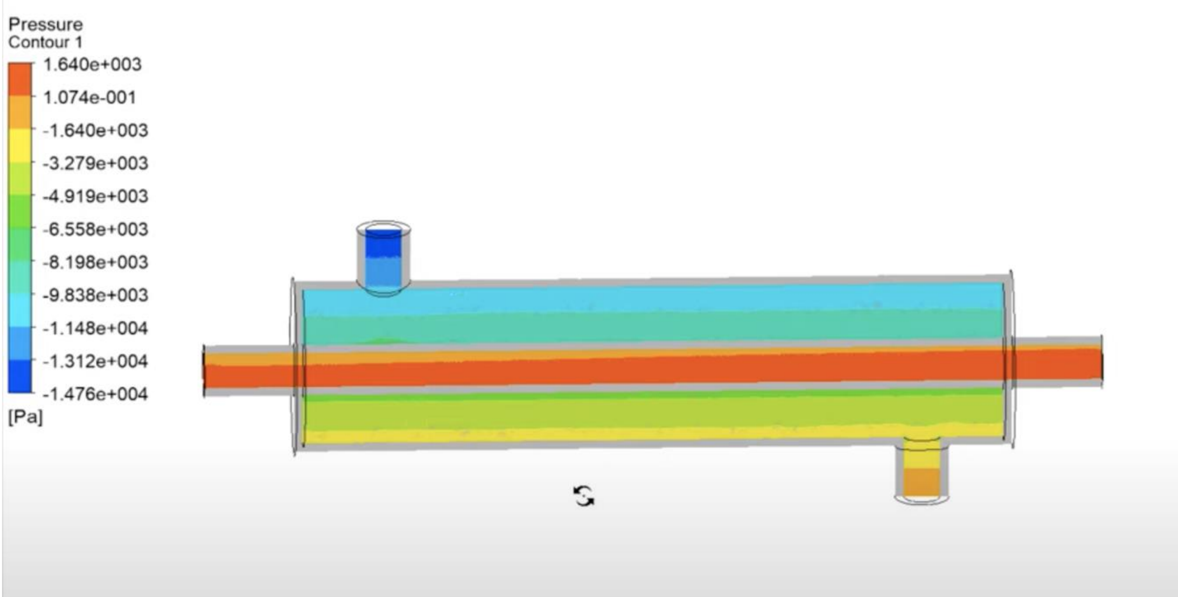

figure 2 a pressure contour distribution is shown above It can be inferred that as the hot fluid passes from the inlet to the outlet there is a positive gradient in pressure across the hot fluid, as shown by the contour plottings, similarly it can be inferred that for the cold fluid, the pressure distribution remains almost always constant since a red color is shown in the contour plottings A remarkable gradient of the pressure contour is obtained while plotting the pressure contour, in the ansys fluent simulation. The pressure contour depicts how there represents a gradual increase in pressure in the hot fluid domain while the fluid enters from the inlet and exits from the outlet. The contour plottings are of significant use since it represents the modeling and simulation when the fluid in this case water fluid enters from both the case and the tube inlet and exits from the case and tube outlet

The below mentioned plots and simulation are done on the Ansys software student version. Since the Ansys software plays a major role in performing simulations and running calculations that are tedious and cumbersome in case it is done manually and thus Ansys software helps in performing real life like simulations.

4. DISCUSSIONS

4.1Vector Distribution

4.1.1 Velocity Vector Distribution

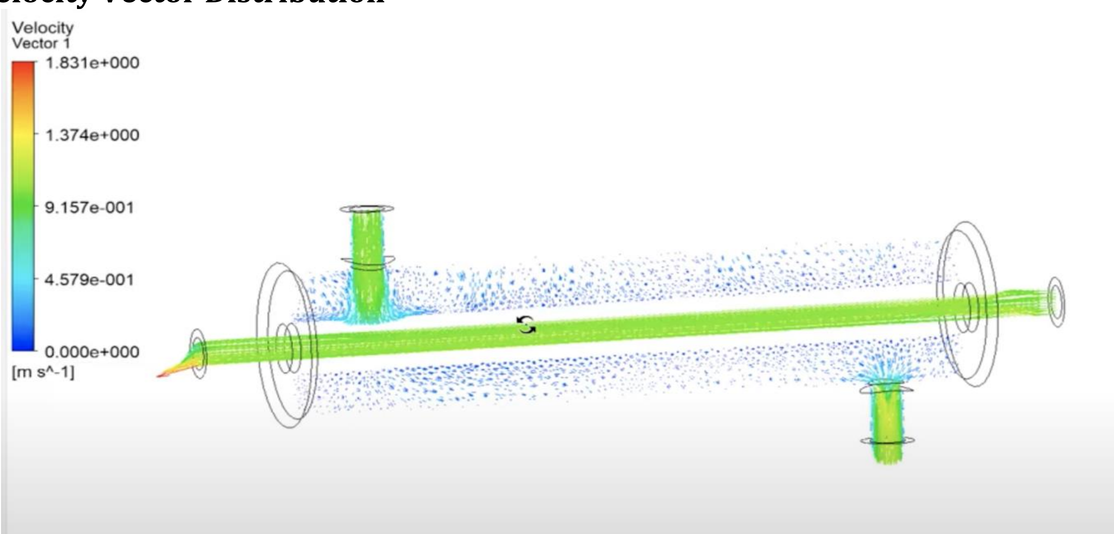

figure 3 the plot shows the velocity vector gradient across the case and tube heat exchanger. The vector plot represents the direction of flow of the fluid in this water-fluid , representing a real life simulation of the flow which can be turbulent as turbulent models have been applied while simulating the flow. The directions are according to the boundary conditions which represent that the simulation is correct in accordance with the imported geometry. The reddish color represents a high velocity which has a gradual gradient to low velocity as shown in the graph. In this case the hot fluid flows from the case inlet to the case outlet with a slight increase in its velocity, whereas the cold fluid enter from the tube inlet and exits from the tube outlet. The cold fluid shows a slight increase in the velocity at the outlet as compared to the inlet.The direction plotted by the above diagram represents the direction of flow of the fluid at higher temperature and fluid at lower temperature through the classifier as shown below.

4.1.2 Temperature Vector Distribution

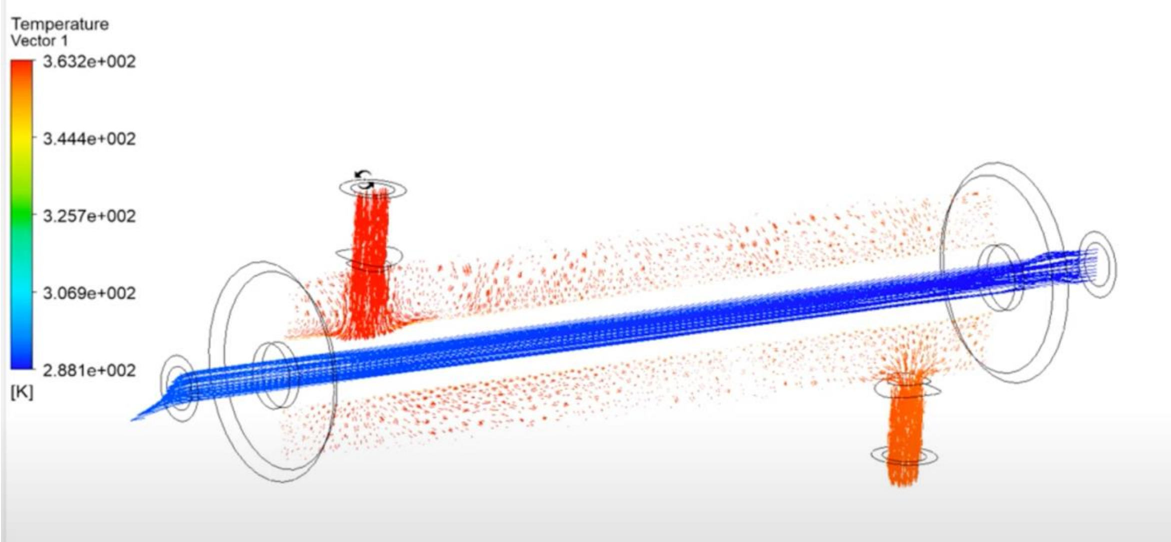

Figure 4 the temperature vector distribution within the intensity heat modifier.

The plot depicts that the hot fluid at the case inlet has been cooled down at the outlet whereas the cold liquid at the tube inlet has shown an increase in temperature while making an exit from the tube outlet The remarkable color gradient represents a clear illustration of the temperature gradient across the field of flow , i.e. the gradual decrease in temperature in the case of fluid at higher temperature flowing in the case side and a gradual increase in temperature for the cold fluid passing through the tube side of the case and tube heat exchanger. The inefficiency of the drawings in the geometry tab have also been taken into account. Thus these errors when taken into account would represent an accurate plot for the temperature vector distribution. Since a vector plot is plotted the direction of the temperature gradient has been plotted as shown in the figure. The direction of decrease of hot fluid temperature is from left to right whereas the direction of increase of temperature of cold fluid is from right to left.

4.1.3 Temperature fluid flow line Distribution

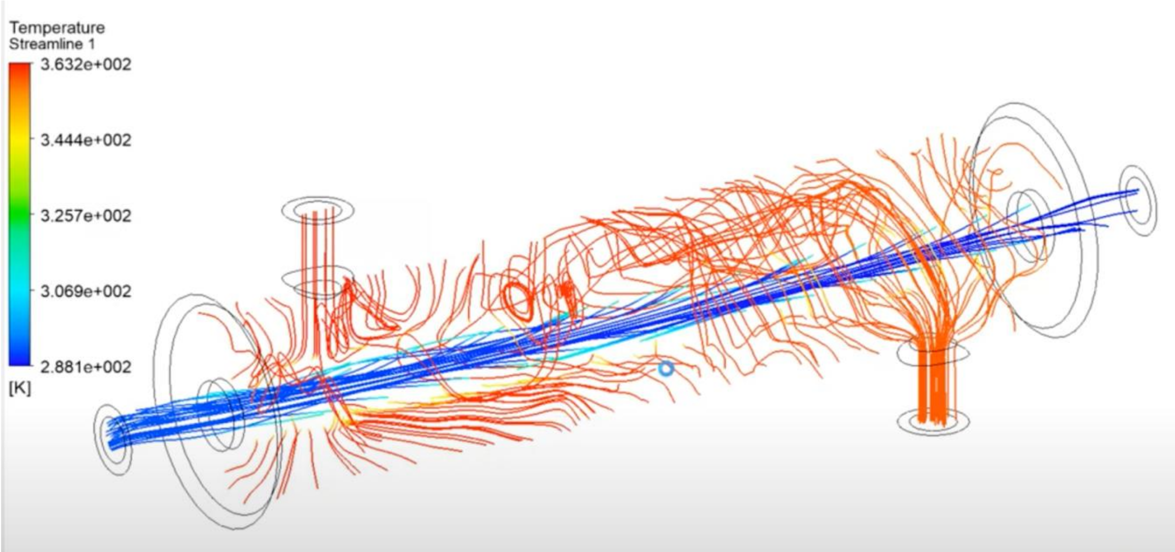

Figure 5 Temperature fluid flowline distribution

The hot fluid as can be seen is flowing around the cooled fluid and hence heat transfer takes place from the fluid at higher temperature to the fluid at lower temperature through the wall of the tube because both the fluids are not in direct contact with each other. There is a wall in between them. Due to this wall there is no physical mixing of the two fluids entering and leaving the case and tube heat exchangers, hence it can be also called as indirect contact heat exchanger. The indirect contact heat exchanger has a low thermal coefficient as compared to direct heat exchanger from one fluid to another i.e. in this case heat from the hot liquid passes to the cold liquid through a wall.
We also know that the effectiveness of a counter flow heat exchanger is greater than a parallel flow heat exchanger since the area available for heat transfer is greater for the former as compared to the later.

References

[1] Emerson, W.H., "case-side pressure drop and heat transfer with turbulent flow in segmentally baffled case-tube heat exchangers", Int. J. Heat Mass Transfer 6 (1963), pp. 649–66.
[2] Haseler, L.E., Wadeker, V.V., Clarke, R.H., (1992), "Flow Distribution Effect in a Plate and Frame Heat Exchanger", IChemE Symposium Series , No. 129, pp. 361-367.
[3] Diaper, A.D. and Hesler, L.E., (1990), "Crossflow Pressure Drop and Flow Distributions within a Tube Bundle Using Computational Fluid Dynamic", Proc. 9th Proc. 9th Heat Transfer Conf., Israel, pp. 235-240.
[4] Jian-Fei Zhang, Ya-Ling He, Wen-Quan Tao , " 3d numerical simulation of case and tube heat exchanger with middle-overlapped helical baffle", a journal ,School of energy and power engineering,china.
[5] Li, H., Kottke, "Effect of baffle spacing on pressure drop and local heat transfer in case and tube heat exchangers for staggered tube arrangement", source book on Int. J. Heat Mass Transfer 41 (1998), 10, pp. 1303–1311.
[6] Thirumarimurugan, M., Kannadasan, T., Ramasamy, E., Performance Analysis of case and Tube Heat Exchanger Using Miscible System, American Journal of Applied Sciences 5 (2008), pp. 548-552.
[7] Usman Ur Ehman , G¨oteborg, Sweden 2011, Master's work 2011:09 on "Heat Transfer Optimization of case-and-Tube & Heat Exchanger through CFD".
[8] Professor Sunilkumar Shinde, Mustansir Hatim Pancha / International Journal of Engineering Research and Applications (IJERA) ,"Comparative Thermal performance of case and tube heat Exchanger with continuous helical baffle using ", Vol. 2, Issue4, July-August 2012.