International Journal of Scientific Engineering and Research (IJSER)
ISSN (Online): 2347-3878
Index Copernicus Value (2015): 62.86 | Impact Factor (2015): 3.791

Fluid Flow Analysis of Parallel Flow Heat Exchanger by Varying Inlet Flow Velocities

T Chandra Sekhar (M.E)[1], Maddi Venu Gopal Reddy (M. Tech)[2]

[1]Assistant Professor (Vasavi College of Engg), Mechanical Engineering, Department Hyderabad -31, Telangana, India

[2]Assistant Professor (Vasavi College of Engg), Mechanical Engineering, Department Hyderabad -31, Telangana, India

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. 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, distill, concentrate, crystallize, or control a process fluid. In a few heat exchangers, the fluids exchanging heat are in direct contact. In most heat exchangers, heat transfer between fluids takes place through a separating wall or into and out of a wall in a transient manner. In many heat exchangers, the fluids are separated by a heat transfer surface, and ideally they do not mix or leak. Such exchangers are referred to as direct transfer type, or simply recuperate. In contrast, exchangers in which there is intermittent heat exchange between the hot and cold fluids—via thermal energy storage and release through the exchanger surface or matrix— are referred to as indirect transfer type, or simply regenerators. Such exchangers usually have fluid leakage from one fluid stream to the other, due to pressure divergences and matrix rotation/valve switching. Common examples of heat exchangers are shell-and-tube exchangers, automobile radiators, condensers, evaporators, air preheaters, and cooling towers.***

Keywords:

1. Applications of Heat Exchangers

- Heat exchangers are used in a wide variety of applications such as home heating, refrigeration, air conditioning, petrochemical plants, refineries as well as in natural gas processing.
- In many industrial processes a heat exchanger helps in using the wasted heat from one process to be utilized in another process which saves a lot of money while being efficient at the same time.
- Cooling of hydraulic fluid and oil in engines, transmissions and hydraulic power packs.
- Heat exchangers are used in many industries, including:
  -Waste water treatment
  -Refrigeration
  -Wine and beer making
  -Petroleum refining
- In commercial aircraft heat exchangers are used to take heat from the engine's oil system to heat cold fuel.

2. Modeling of Heat Exchangers

Figure 1: Single tube Parallel flow heat exchanger

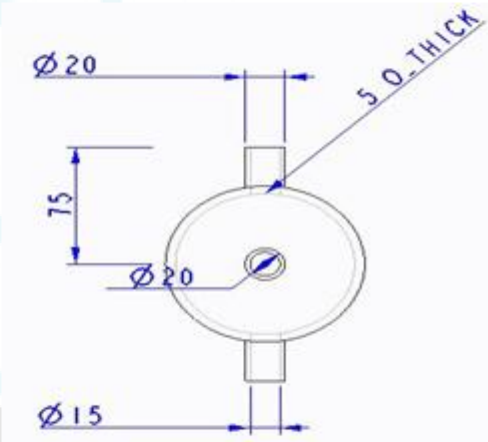

Figure 2: Front view

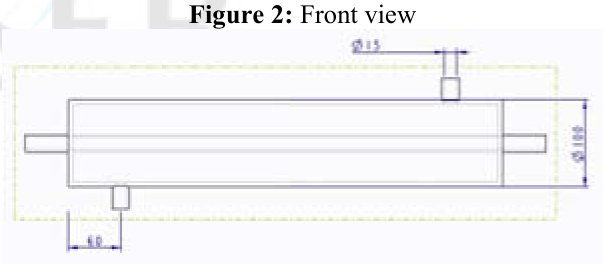

Figure 3: Side view

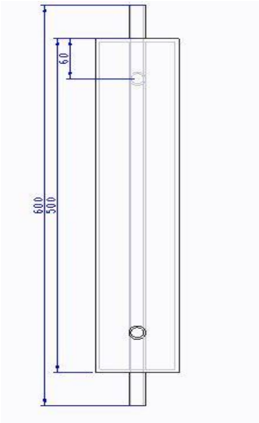

Figure 4: Top view

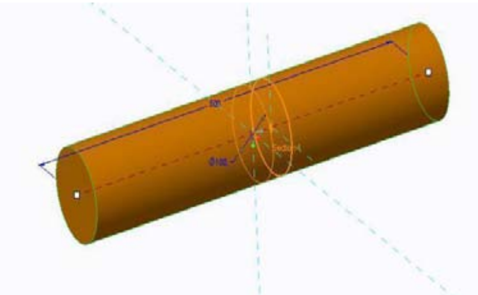

Figure 5: Extruded part

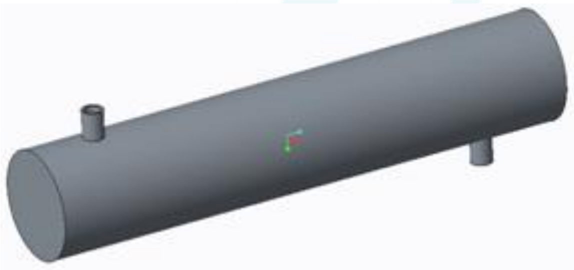

Figure 6: Tube with cold inlets

3. Case-Analyses Tool

ANSYS (Analytical System)
Finite element analysis (FEA) is a computer simulation technique used in engineering analysis. It uses a numerical technique called the finite element method (FEM). There are many finite element software packages, both free and proprietary.

ANALYSIS
Key topics covered: Introduction to the finite element method Getting started with ANSYS software stress analysis dynamics of machines fluid dynamics problems thermo mechanics contact and surface mechanics exercises, tutorials, worked examples With its detailed step-by-step explanations, extensive worked examples and sample problems, this book will develop the reader's understanding of FEA and their ability to use ANSYS's software tools to solve their own particular analysis problems, not just the ones set in the book.

At ANSYS, we bring clarity and insight to customers' most complex design challenges through fast, accurate and reliable simulation. Our technology enables organizations to predict with confidence that their products will thrive in the real world. They trust our software to help ensure product integrity and drive business success through innovation.

Every product is a promise to live up to and surpass expectations. By simulating early and often with ANSYS software, our customers become faster, more cost-effective and more innovative, realizing their own product promises.

Individual Component Analysis:
Importing the component from cad (cero) tool to cae tool (ansys):

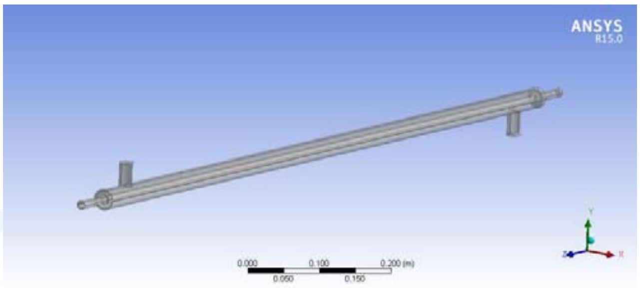

Figure 7: Imported file into ansys

Table 1: Mesh details

Use Advanced Size Function	On: Curvature
Relevance Center	Coarse
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Slow
Span Angle Center	Fine
Curvature Normal Angle	Default (18.0 °)
Min Size	Default (0.552760 mm)
Max Face Size	Default (55.2760 mm)
Max Size	Default (110.550 mm)
Growth Rate	Default (1.20)
Minimum Edge Length	15.7080 mm

Mesh Statistics

Nodes	56504
Elements	193357

CASE 1

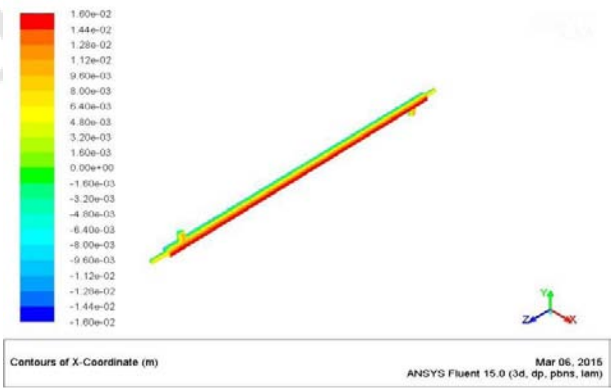

Figure 8: Contours of x-coordinates

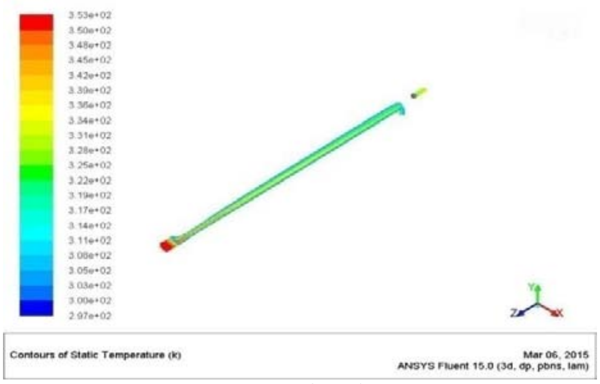

Figure 9: Contours of static Temperature

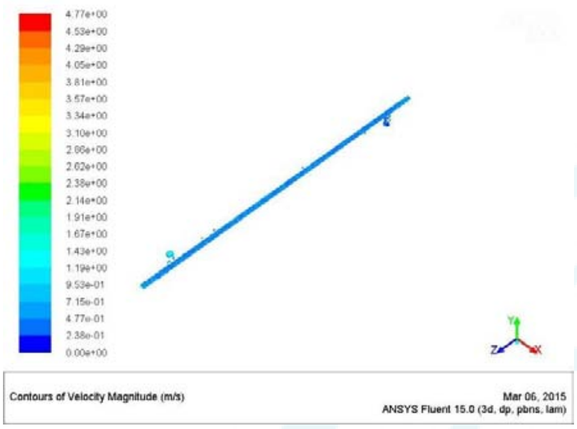

Figure 10: Contours of velocity Magnitude

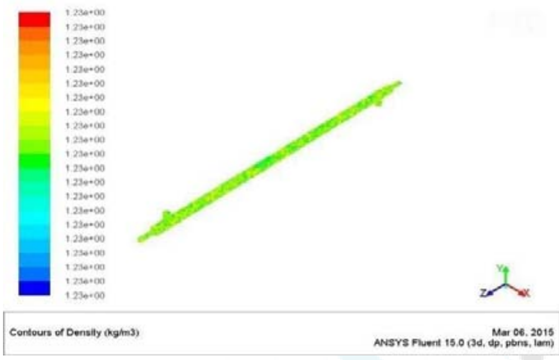

Figure 11: Contours of Density

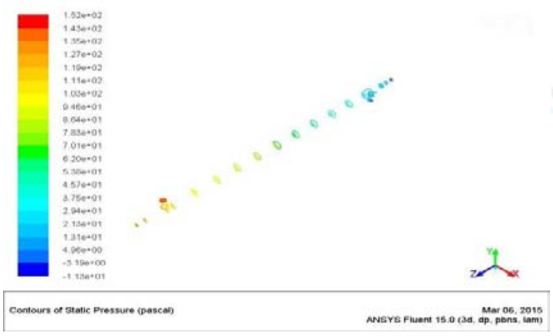

Figure 12: Contours of static pressure

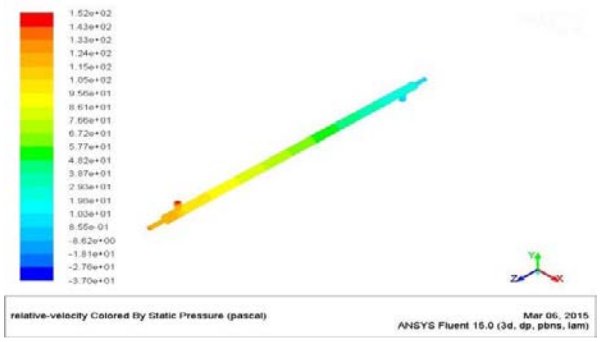

Figure 13: Relative velocity colored by static pressure

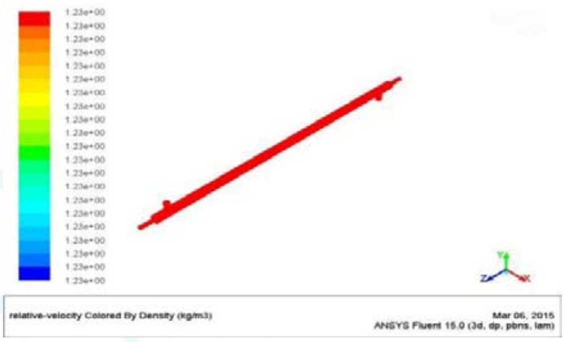

Figure 14: Relative velocity colored by Density

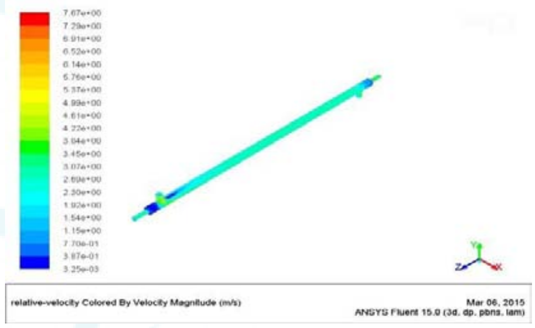

Figure 15: Relative velocity colored by velocity magnitude

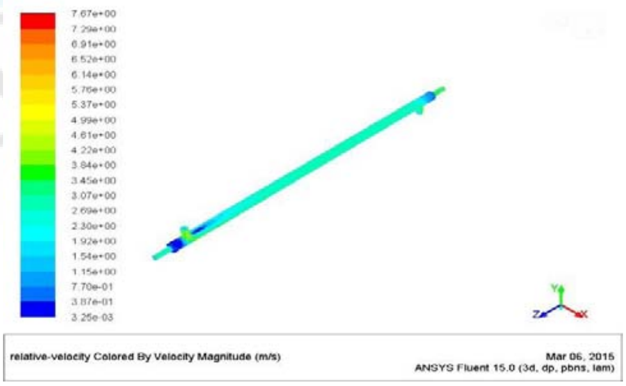

Figure 16: Relative velocity colored by velocity magnitude

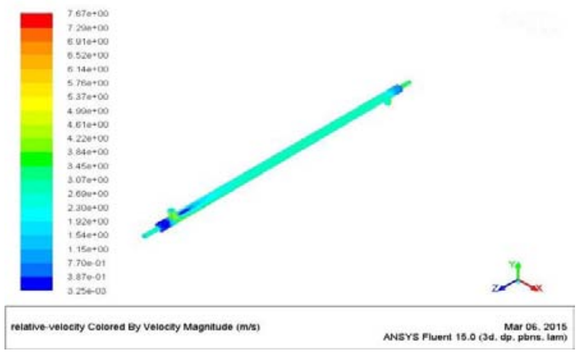

Figure 17: Relative velocity colored by velocity magnitude

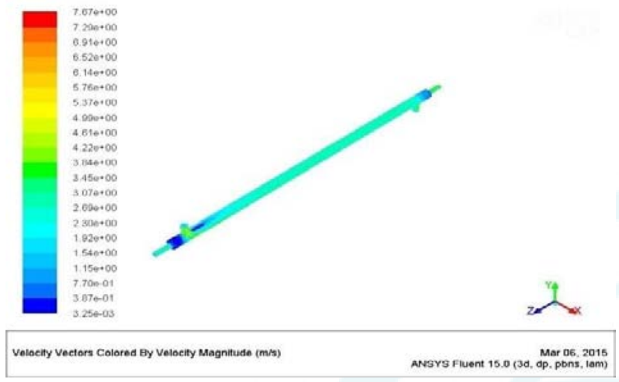

Figure 18: Velocity vectors colored by velocity magnitude

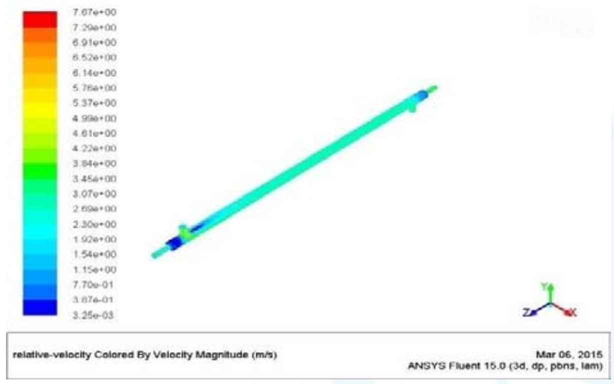

Figure 19: Relative velocity colored by velocity magnitude

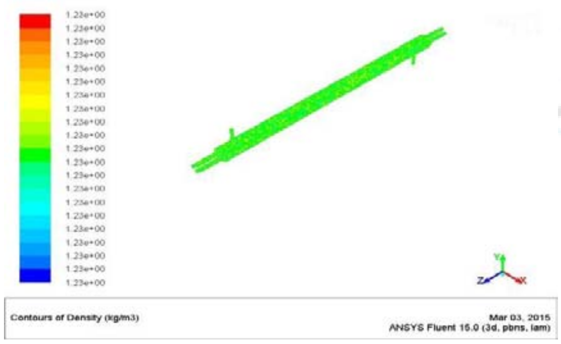

Figure 20: Contours of Density

CASE 2

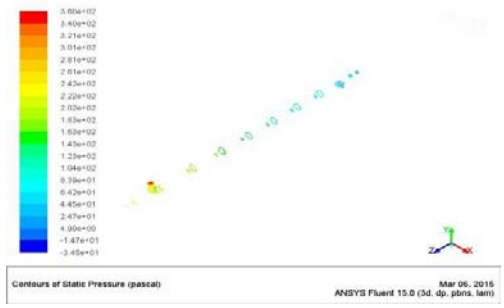

Figure 21: Contours of static pressure

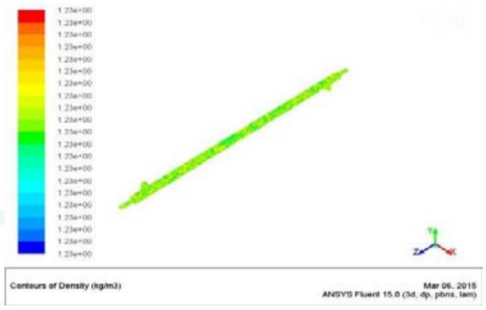

Figure 22: Contours of Density

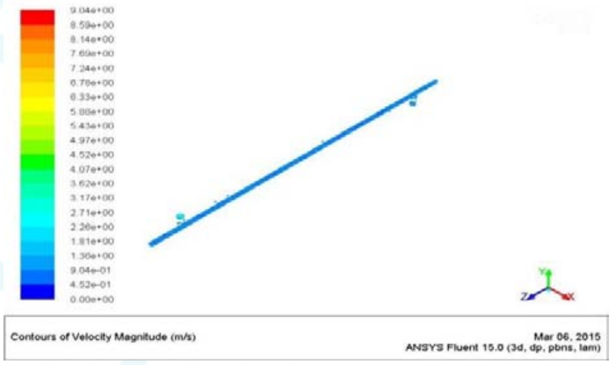

Figure 23: Contours of velocity magnitude

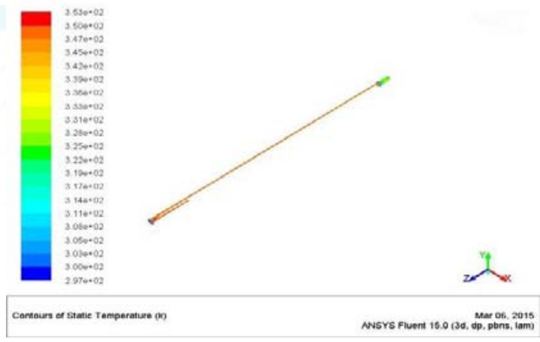

Figure 24: Contours of static temperature

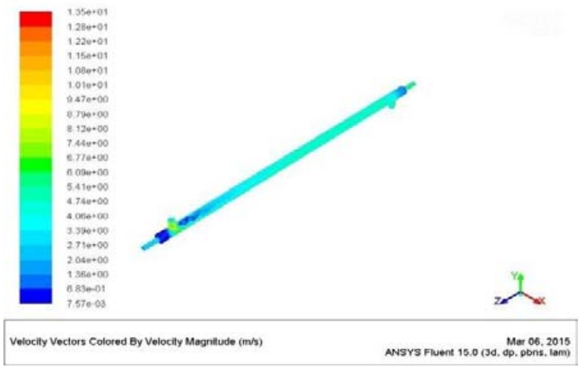

Figure 25: Velocity vectors colored by Velocity magnitude

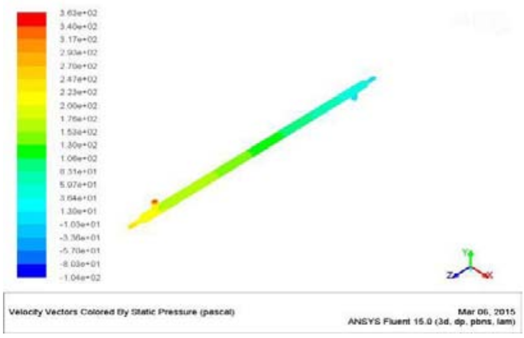

Figure 26: Velocity vectors colored by static Pressure

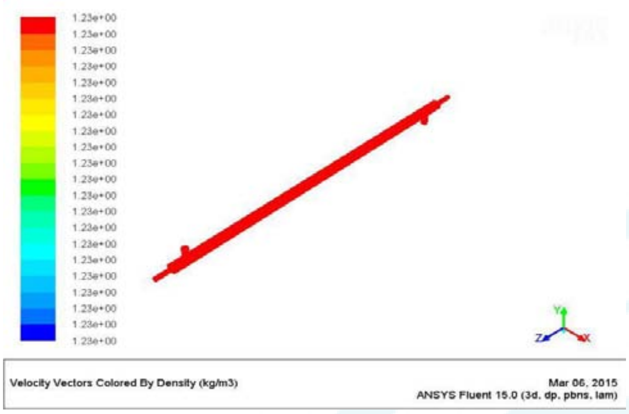

Figure 27: Velocity vectors colored by Density

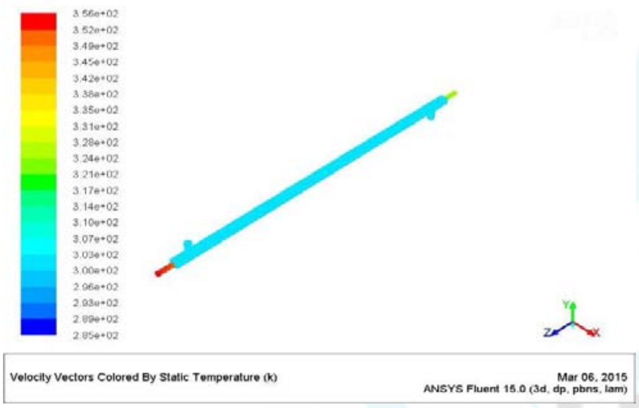

Figure 28: Velocity vectors colored by static temperature

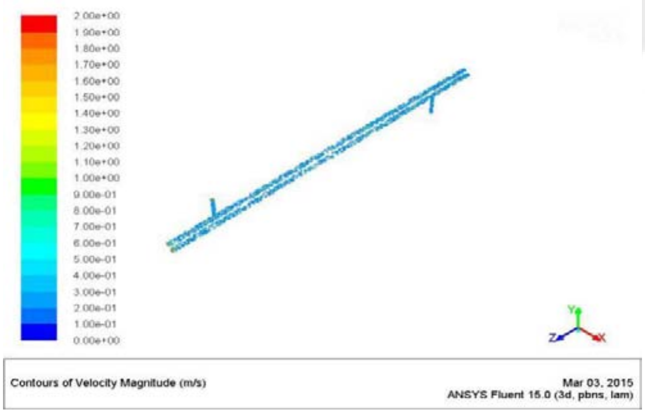

Figure 29: Contours of velocity magnitude

CASE 3

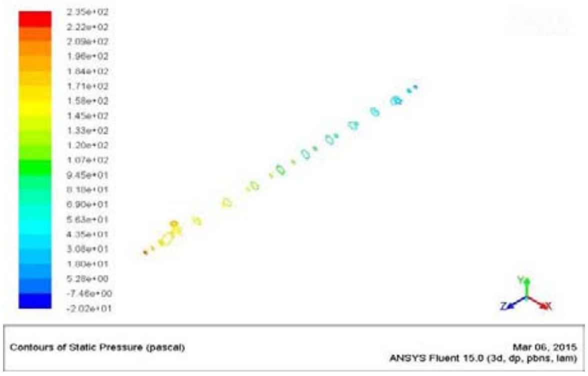

Figure 30: Contours of static pressure

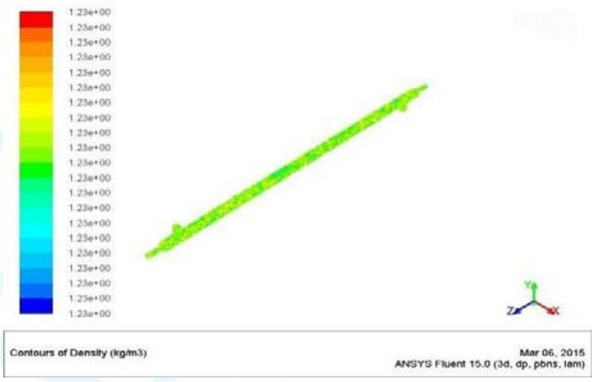

Figure 31: Contours of Density

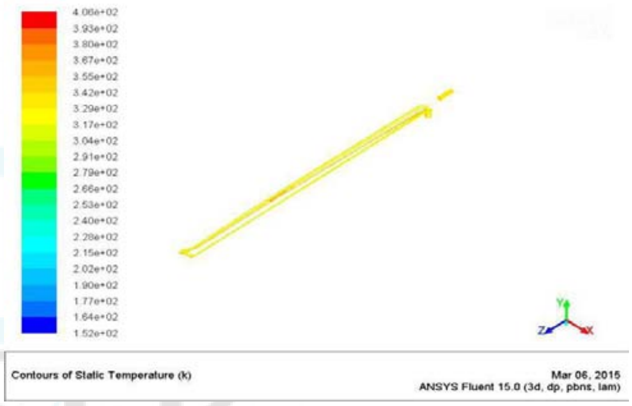

Figure 32: Contours of static temperature

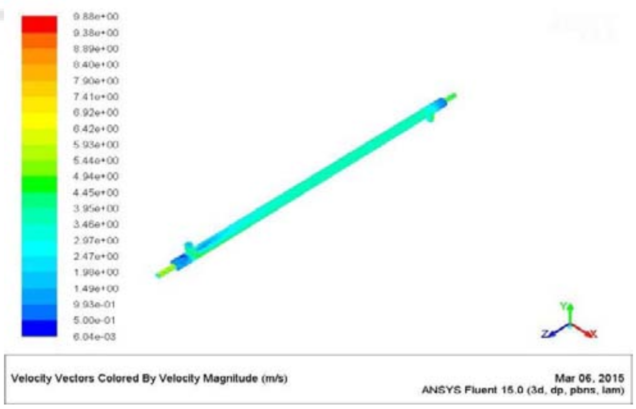

Figure 33: Velocity vectors colored by velocity magnitude

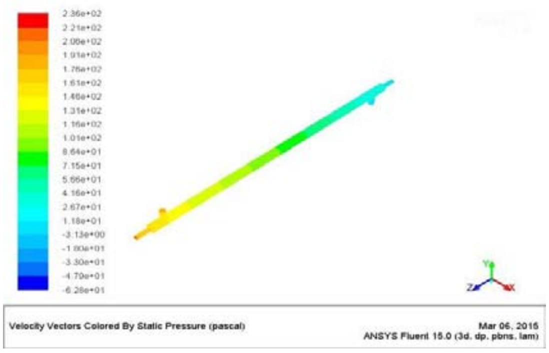

Figure 34: Velocity vectors colored by static pressure

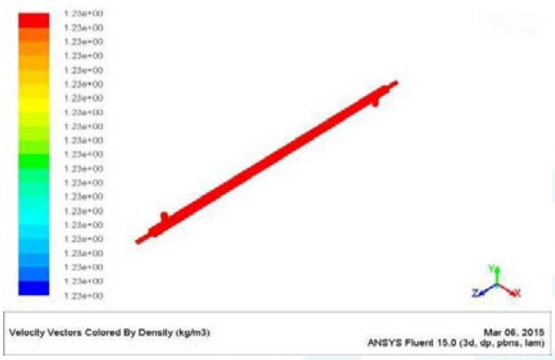

Figure 35: Velocity vectors colored by density

4. Results and Discussion

By following the above steps the modeling of Shell and Tube heat exchanger has been done using solid works software and also using ansys fluent simulation the flow analysis is done to calculate the effectiveness of the shell and tube heat exchanger.

The mass flow rate of water in the shell is 0.8kg/s with an inlet temperature of 300k and the mass flow rate of water in tube is0.2 kg/s at an inlet temperature of 353k. Then after setting up these boundary conditions we can observe the temperature difference and changing in internal energy and enthalpy values.

Case	Static Pressure (kpa)	Temp	Enthalpy (KJ/Kg)	Entropy (J/Kg)	Internal Energy(KJ/Kg)
Case I	13.3	356k	58.1	8.28	-25
	-19.0	297k	-1.59	$-1.7*10^{2}$	-91
Case 2	11.3	356k	57	5	-26.5
	-22.5	298k	-0.014	$-1.7*10^{2}$	-173
Case 3	30.3	357k	58.2	1.04	-25
	-1.6	131k	-268	$-3.5*10^{3}$	-355

From above table we can conclude that case 3 we are getting best temperature drop effects and as the cold fluid flow is slowly through the tubes it will carry max amount of heat energy from hot fluid body.

5. Future Scope Discussion

The design and CFD analysis on shell and tube heat exchanger has been done and the results were compared with the effectiveness NTU method. We had observed a considerable amount of deviation from the actual value to the value that is obtained in the simulation.

This project has further developments like considering different types of flows like cross, parallel and counter flow. And also here we have considered that the radiation and convection losses as zero where as in practical situations they will exist so this project can be further extended in that path.

Since the significance of various design soft wares is increasing day by day this project can be executed in any kind of design soft wares like PRO-E and CATIA. But Since the SOLID WORKS platform is user friendly we opted for this software.

References

[1] Cengel, Y. A., Heat Transfer: A Practical Approach, 2"d Edition, McGraw- Hill, 2003.
[2] Solid Works Flow Simulation 2010
[3] SadikKakaç and Hongtan Liu (2002). Heat Exchangers: Selection, Ratingand Thermal Design (2nd ed.). CRC Press. ISBN 0-8493-0902-6.
[4] Shell and Tube Heat Exchanger Design Software for Educational Applications
[5] Heat exchanger shell side and tube side pressure drop calculator Calculate shell side / tableside pressure drop for an exchanger.