CFD Analysis of a Shell and Tube Heat Exchanger Using Different Header Sections

Digvendra Singh[1], Dr. Jitendra Pandey[2] and Dr. Abhishek Tiwari

[1]Post graduate student, Bansal Institute of Research & Technology, Bhopal (M.P.)
[2]Associate Professor and HOD, ME Dept. Bansal Institute of Research & Technology, Bhopal (M.P.)
[3]Assistant Professor, ME Dept. Bansal Institute of Research & Technology, Bhopal (M.P.)

Abstract

In present work CFD prediction is done for shell and tube heat exchanger for different header sections. Heat exchanger has a variety of applications in different industries and in this study one such heat exchanger is taken in to account. The heat exchanger is designed as per the commercial needs of the industry. Kern's technique is used to design the heat exchanger .The designing procedure results in a shell and tube heat exchanger having 21 tubes, 170mm shell diameter and 610 mm long. As the designing procedure doesn't include the type of the header to be used, so we have analyzed three types of header which can provide a uniform velocity in the inlet of each tube. Different geometries are included in different positions of the inlet nozzle for the header. CFD simulations are used for the optimum positioning of the inlet nozzle which could be proposed from the uniform distribution of the liquid methanol and the uniform velocity distribution though each and every tube. The main objective of this paper is to verify the heat exchanger designed with the use of the Kern's technique, by the use of Commercial Computational Fluid Dynamics (CFD) software. For the simulation, purpose a symmetric view of the simplified geometry of the heat exchanger is made using solid works software.

***Keywords**: CFD, shell and tube heat exchanger, Headers, solid works software.*

1. Introduction

Heat exchangers are devices used to transfer heat energy from one fluid to another. Typical heat exchangers experienced by us in our daily lives include condensers and evaporators used in air conditioning units and refrigerators. Boilers and condensers in thermal power plants are examples of large industrial heat exchangers. There are heat exchangers in our automobiles in the form of radiators and oil coolers. Heat exchangers are also abundant in chemical and process industries. Different heat exchangers are named according to their applications. For example, heat exchangers being used to condense are known as condensers; similarly heat exchangers for boiling purposes are called boilers. Performance and efficiency of heat exchangers are measured through the amount of heat transferred using least area of heat transfer and pressure drop. A better presentation of its efficiency is done by calculating over all heat transfer coefficient. Pressure drop and area required for a certain amount of heat transfer, provides an insight about the capital cost and power requirements (Running cost) of a heat exchanger. Usually, there is lots of literature and theories to design a heat exchanger according to the requirements. A good design is referred to a heat exchanger with least possible area and pressure drop to fulfil the heat transfer requirements. Parallel-plate heat exchangers may be finned or corrugated and may be used in single-pass or multipass modes of operation. Flow passages associated with compact heat exchangers are typically small, and the flow is usually laminar.

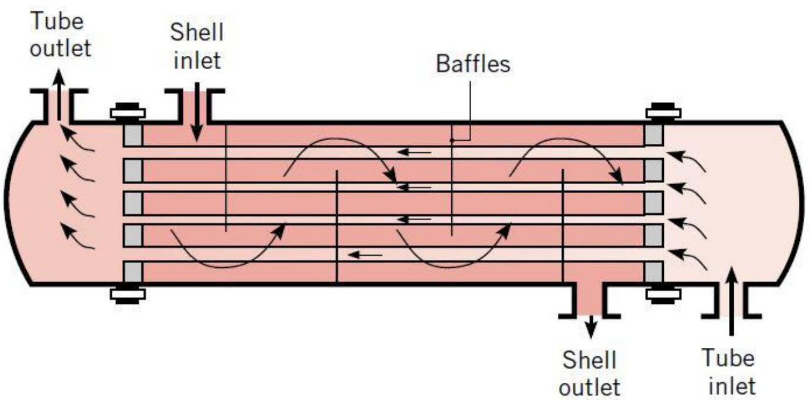

Figure-1. Shell-and-tube heat exchanger with one shell pass and one tube pass (cross-counter flow mode of operation)

2. Model details

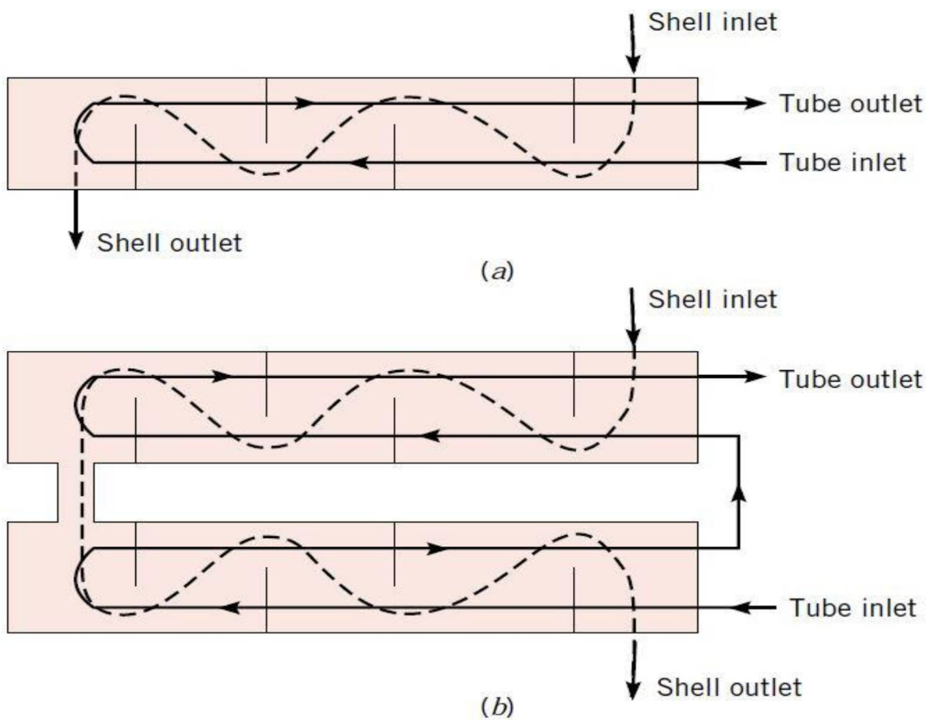

Figure-2 Shell-and-tube heat exchangers. (*a*) One shell pass and two tube passes. (*b*) Two shell passes and four tube passes.

Tube Layout

- ✓ Triangular pitch (30° layout) is better for heat transfer and surface area per unit length (greatest tube density.)
- ✓ Square pitch (45° & 90° layouts) is needed for cleaning.
- ✓ The 30°,45° and 60° are staggered, and 90° is in line.
- ✓ For the identical tube pitch and flow rates, the tube layouts in decreasing order of shell-side heat transfer coefficient and pressure drop are: 30°,45°,60°, 90°.
- ✓ The 90° layout will have the low heat transfer coefficient and the low pressure drop.

3. CALCULATIONS

On the basis of the given problem we will be calculating the values of the basic dimensions of the shell and tube heat exchanger .Only the thermal design will be considered. The Kern's method will be employed for the calculation. As this process is an iterative process, I'm only representing the final iteration of the calculation. As coolant is corrosive, so it is assigned to the tube-side.

3.1 Counter Flow Heat Exchanger

Heat capacity water, Cp_h= 4.182 Kj/kg k

Heat load , $Q = m_h \cdot c_{ph} (t_{h1}-t_{h2})$

$= 0.055036 \text{ X } 4.182 \text{ X } (100-81.31)$

$Q = 4.301\text{kw}$

Heat capacity methanol, C_{pc} = 2.534 kJ/Kg k

The cold and hot stream heat load are equal so, cooling methanol flow rate is calculated as follow

Cooling methanol flow,

$Q = m_c.C_{pc} (t_{c2}-t_{c1})$

$$m_c = \frac{Q}{C_{p_c}\left(t_{c_1} - t_{c_2}\right)}$$

$$= \frac{4.301}{2.534(58.22-25)}$$

$m_c \quad = \quad 0.0511$ kg/s

4. SELECTION OF HEADER

Several different designs for a header type were studied with the help of CFD simulation using ANSYS to achieve uniform distribution of liquid water in the header of a shell-and-tube heat exchanger. The different geometries included the position and shape of the inlet nozzle. In CFD simulation, the k-epsilon realizable turbulent model was employed. Flow patterns in the header could be visualized.

Figure4. Different header configurations of the shell-and-tube type heat exchanger.

4.1 CFD ANALYSIS

Computational Fluid Dynamic (CFD) study of the system starts with building desired geometry and mesh for modeling the domain. Generally, geometry is simplified for the CFD studies. Meshing is the discretization of the domain into small volumes where the equations are solved by the help of iterative methods. Modeling starts with defining the boundary and initial conditions for the domain and leads to modeling the entire system domain. Finally, it is followed by the analysis of the results.

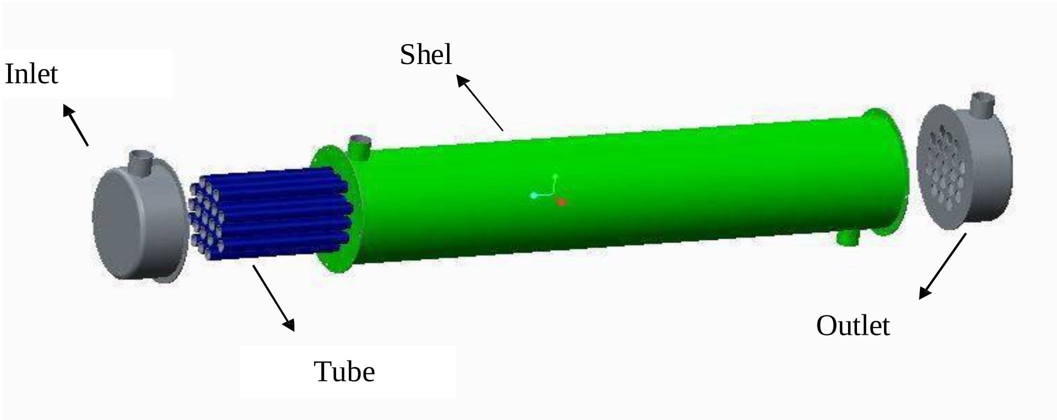

Figure 4.1 Shell and tube heat exchanger original geometry

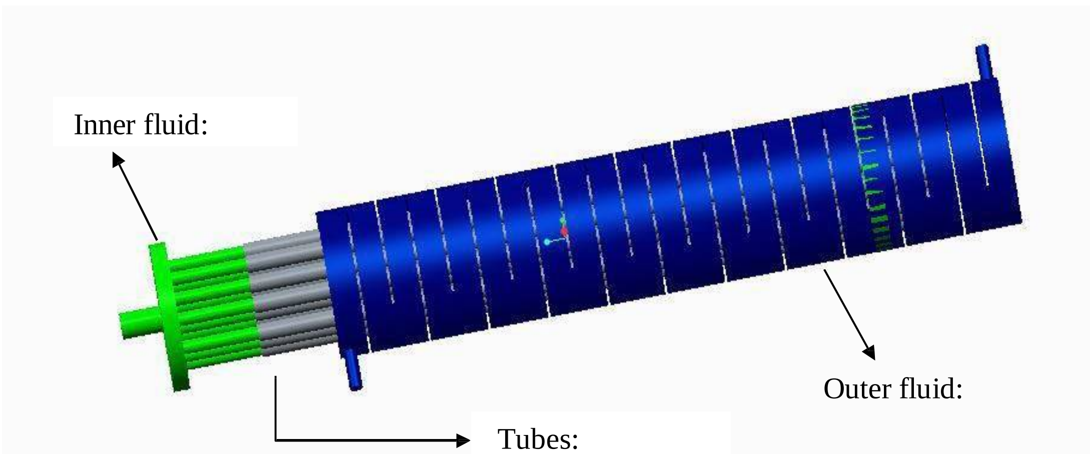

Figure 4.2 Fluid assembly model of the shell and tube heat exchanger

5. RESULTS

In our staring of the project we have calculated the design values for our heat exchanger. Those values calculated were used in the CFD simulation for the analysis of our heat exchanger designed. Now to study our design our through Ansys we studied the temperature pressure and velocity profiles for our heat exchanger.

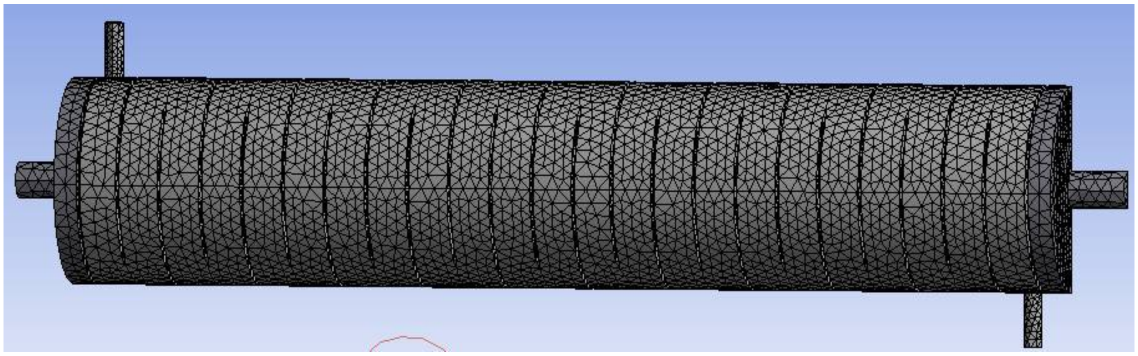

Figure 5.1 the mesh generated for the Geometry of the shell and tube Heat Exchanger

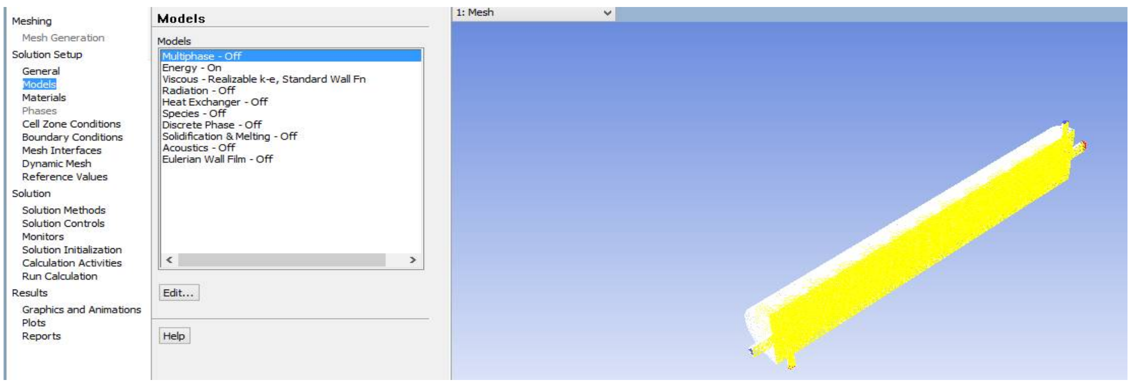

Figure 5.2 Fluent setup

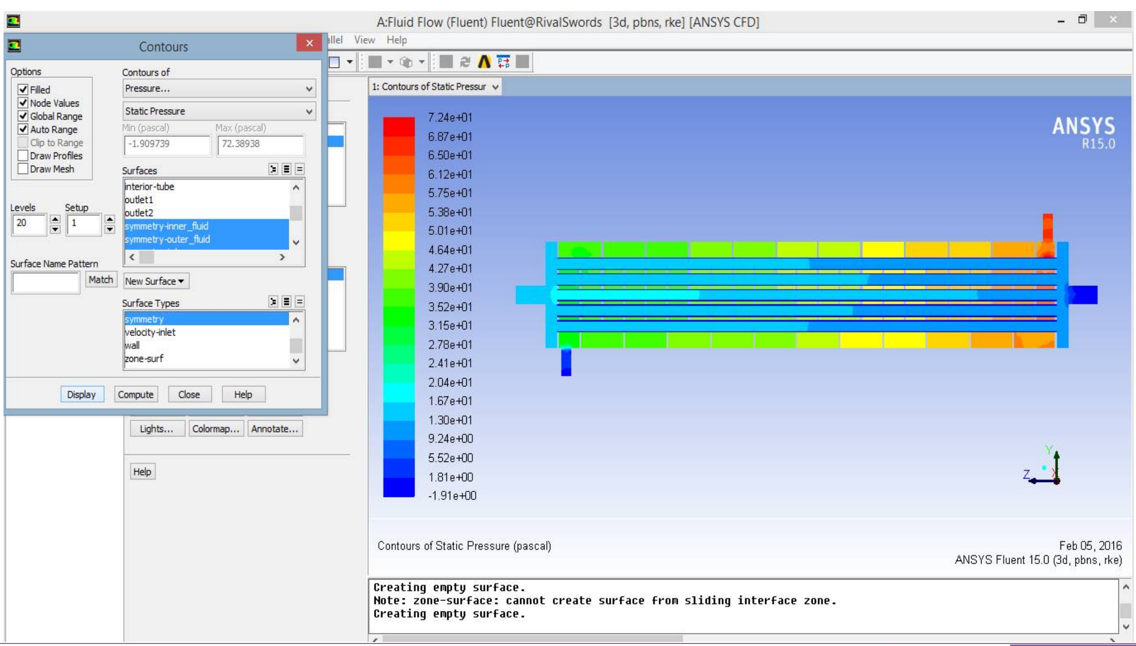

Figure 5.3: Pressure Contour Plot at Symmetrical Plane

These are the contours of pressure for case 2. Pressure on any location can be obtained in Pascal by matching the colour on that location with colour of colour scale.

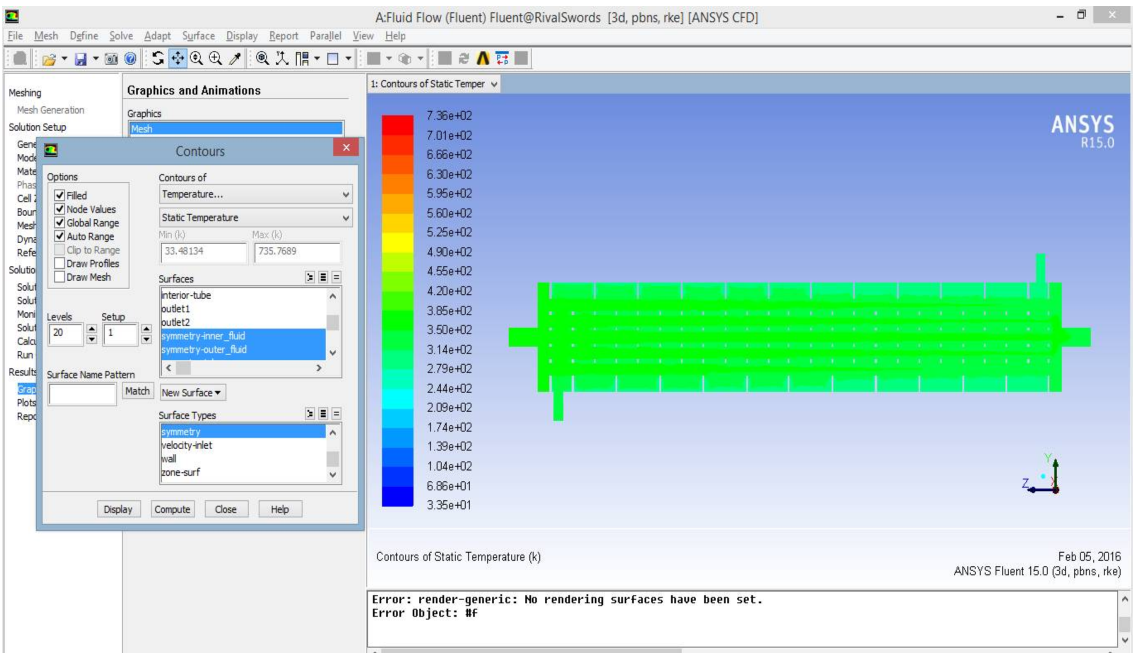

Figure 5.4: Temperature Contour Plot at Symmetrical Plane

These are the contours of temperature for case 2. Temperature of fluid/tubes on any location can be obtained in kelvins by matching the colour on that location with colour of colour scale.

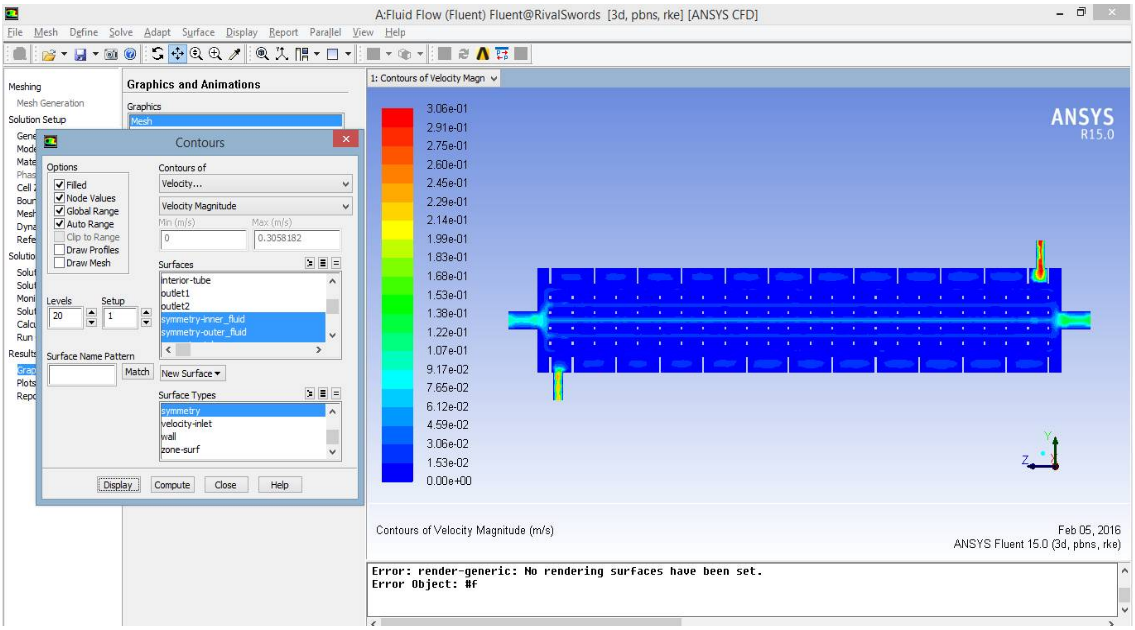

Figure 5.5: Velocity Contour Plot at Symmetrical Plane

These are the contours of velocity for case 2. Velocity of any fluid on any location can be obtained in m/s by matching the colour on that location with colour of colour scale.

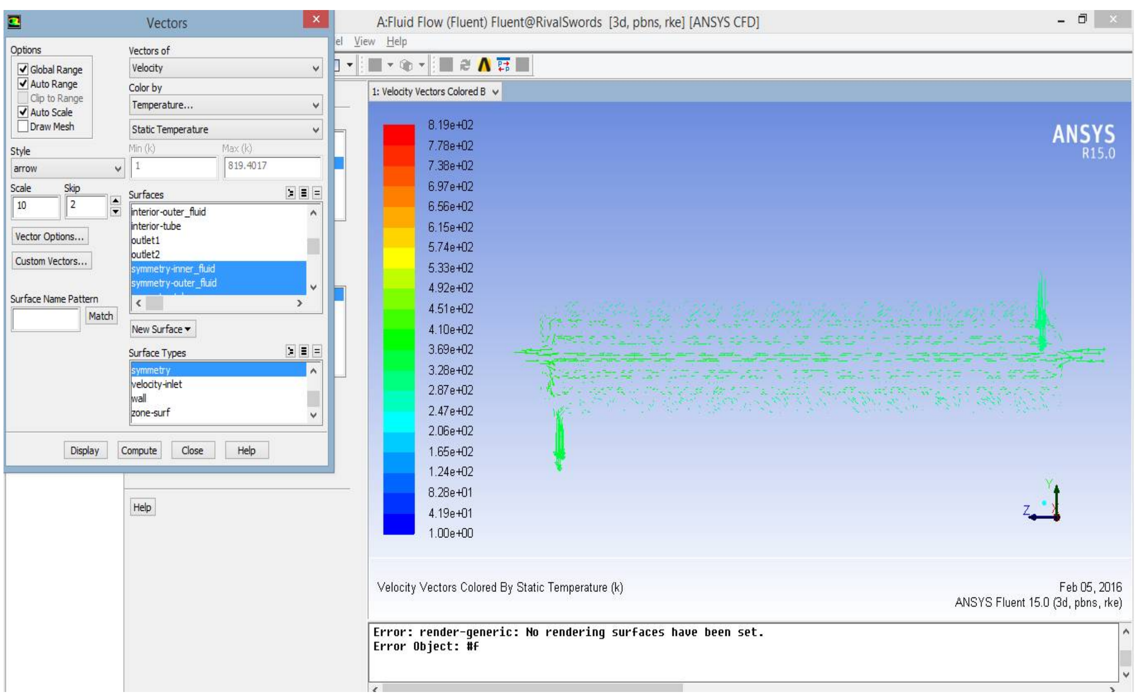

Figure 5.6: Velocity Vectors Contour Plot at Symmetrical Plane

These are the velocity vectors for case 2. These vectors are coloured by temperature. Temperature of fluid molecules on different locations is obtained by matching the colour on that location with colour of colour scale.

6. CONCLUSION

The shell and tube heat exchanger is designed is simulated using Computational Fluid Dynamic (CFD).The header selection for the heat exchanger has also been based on the Computational Fluid Dynamic (CFD) simulation. We can see that the uniform flow in tubes can be achieved using a suitable header. The nozzle placement normal to the plane of tubes and also eccentric to the head side of the headers has been the most effective. The simplified geometry of the shell and tube heat exchanger is used. The assumption of plane symmetry works well for most of the length of heat exchanger except the outlet and inlet regions where the rapid mixing and change in flow direction takes place. Thus improvement is expected if complete geometry is modeled. Furthermore, the enhanced wall functions are not used in this project due to convergence issues, but they can be very useful with k–epsilon models. The heat transfer is found to be poor because the most of the shell side fluid by-passes the tube bundle without interaction. Thus the design can be modified in order to achieve the better heat transfer in two ways. Either, the shell diameter is reduced to keep the outer fluid mass flux lower or tube spacing can be increased to enhance the inner fluid mass flux.

7. Future Scope

- The setup may be tested with totally different layout as an example transmuting the baffle layout or form of Shell and tube device in CFD also. The device style attributes may be transmuted like tube arrangement, tube length etc.
- Use the CFD software taken a additional precise lead to minimum time.

References

[1] Learning the Computer-Aided Engineering (CAE) i.e. Ansys / Computational Fluid Dynamic (CFD) Software from CAD centre Borivali MUMBAI (M.H) INDIA at **August'2015**. **STUDENT ID NO. - M150618456**

[2] *Žarko Stevanović, Gradimir Ilić, Nenad Radojković, Mića Vukić, Velimir Stefanović, Goran Vučković,* ***"Design of Shell-And-Tube Heat Exchangers By victimisation Cfd Technique – half One:*** *Thermo-Hydraulic Calculation", Facta Universitatis Series: Mechanical EngineeringVol.1, No 8, pp. 1091 – 1105,2001.* 01.

[4] S. Sadik Kakac and H. Liu, "Heat exchangers: selection, rating, and thermal design", 2ndEdition, CRC Press, Boca Raton, FL 2002.

[5] M. S. Peters, K. D. Timmerhaus and R. E. West, "Plant design and economics for chemical engineers", 5th Ed., McGraw-Hill, Boston 2003.

[6] R. K. Sinnott, "Coulson & Richardson's Chemical Engineering Design", Vol. 6, 4th Ed., Elsevier Butterworth-Heinemann, Jordan Hill, 2005.

[7] Frank P. Incropera, David P. Dewitt, Theodore L. Bergman, Adrienne S. Lavine "Fundamentals Of Heat And Mass Transfer", John Wiley & Sons SIXTH EDITION, 111 River Street, Hoboken, NJ 07030-5774 2007.

[8] H K Versteeg And W Malalasekera,"An Introduction To Computational Fluid Dynamics", 4th Ed., Pearson Education Limited, England, 2007.

[9] Ender Ozden, Ilker Tari, "Shell Side Cfd Analysis of a Small Shell-And-Tube HeatExchanger", Energy Conversion and Management, Vol. 51, No. 5, pp. 1004-1014 2010.

[10] YANG Xia, ZHANG Jie, YU Jiuyang, WU Yanyang, LUO Yan, GUO Jia., **"Numerical Simulation on Flow and Heat Transfer at Shell-Side of Flat-Tube Heat Exchangers"**, Wuhan University and Springer-Verlag Berlin Heidelberg,Vol.15 No.5, 427-432, 2010.

[11] *Yang-gu Kim, Byoung Ik Choi and Kuisoon Kim*, **"Performance Analysis and Optimal Design of Heat Exchangers Used in High Temperature and High Pressure System"**, Int'l J. of Aeronautical & Space Sciences, Vol. 11, No. 1 March 2010.

[12] Ender Ozden, Ilker Tari, **"Shell side CFD analysis of a small shell-and-tube heat exchanger"**, Energy Conversion and Management Vol. 51, No. 5, pp. 1004-1014 (2010).

[13] http://www.wcr-regasketing.com, **"Heat exchanger applications"** http://www.wcr-regasketing.com/heat-exchanger-applications.htm, 2010.

[14] *Muhammad Mahmood Aslam Bhutta, Nasir Hayat, Muhammad Hassan Bashir, Ahmer Rais Khan, Kanwar Naveed Ahmad, Sarfaraz Khan*,"**CFD applications in various heat exchangers design: A review"**, Elsevier, Vol. 32,1-12,2012.

[15] *Gh.S. Jahanmir, F. Farhadi*, "**Twisted bundle heat exchangers performance evaluation by CFD"**, International Communications in Heat and Mass Transfer Vol. 39, pp. 1654–1660 2012.

[16] *Ansys*, **"ANSYS CFD-Post Tutorials"**, Ansys,Inc. Southpointe 275 Technology Drive Canonsburg, PA15317, 2012.

[17] *Hetal Kotwal, 2D.S PATEL*, **"CFD Analysis of Shell and Tube Heat Exchanger- A Review",** International Journal of Engineering Science and Innovative Technology (IJESIT) Volume 2, Issue 2, March 2013.

[18] "*Cfd-online wiki*." **http://http://www.cfd-online.com/Wiki/Main_Page**, 2011.

[19] Majid Hayati **"Kern technique of Shell-And-Tube Heat Exchanger"**, https://www.researchgate.net Lecture 1, University of Kashan, Kashan, I.R. IRAN 2014.

[20] P M V Subbarao **"Figure : Arrangement of tubes, Tube Layout and Flow Scales" mel709 tutorial** ,Page no. 5 and 15, Professor m*echanical Engineering Department* ***I I T Delhi***

A Brief Author Biography

1st Digvendra Singh **–**Post graduate student of Thermal Engineering in Mechanical Engineering dept. at Bansal Institute of research & technology, Under RGPV University, Bhopal (M.P.)