International Journal of Innovative Technology and Exploring Engineering (IJITEE)
ISSN: 2278-3075 (Online), Volume-9 Issue-4, February 2020

CFD Analysis of Heat Pipe Heat Exchanger to Predict the Temperature Distribution.

Tajamul Hamid Mir, Amritpal Singh, Ajay Singh Rana.

Abstract: The Computational Fluid Dynamic (CFD) Analysis of Heat Pipe Heat Exchanger (HPHE) is done to predict the temperature distribution using ANSYS-ICEM modular/meshing and FLUENT solver. In this study, HPHE is modeled in four different cases with and without fillet near the inlet and outlet sections including (standard HPHE, with enlarge inlet and outlet sections, with horizontal plate near the entrance zone, using three different cone of angles (36.03 degree, 30 degree and 45 degree)). The mass flow rate 3.75kg/sec of hot air or gas as given at the inlet section. The Standard k-ε-Realizable turbulence model was used for fluid flow in simulations. The magnitude and location of the temperature distribution, velocity, and turbulence kinetic energy are influenced by prescribed conditions. However, pressure drop is reduced up-to certain extent (due to change in turbulence kinetic energy) for all the cases in which round corner/fillet at the inlet and outlet section was made in the model. At the same time jet type flow is also reduced because of reduction in axial velocity and increment of Y & Z directional velocity which tends to expansion of flow toward the y and z direction.

***Keywords*: Fillet, Heat Pipe Heat Exchanger, Turbulence, Temperature Distribution.**

I. INTRODUCTION

Heat pipe heat exchanger is a heat transfer device in which latent heat of vaporization is utilized to transfer heat over long distance corresponding to small temperature difference. There is an individual closed tube bundle inside the HPHE, these tubes are filled up with a working fluid which evaporates at the evaporator section and condenses at the condenser section. The condensed fluid is then returned back to the evaporator section and this process repeats itself. A heat pipe can have high thermal conductivity much more than a solid conductor like silver or copper. If the working fluid is a mixture of solid- liquid, the heat pipe works on both the principles of thermal conductivity and phase change with high thermal conductivity. Hence, the high heat transport capacity makes heat exchanger with heat pipes much smaller than traditional heat exchangers in handling high heat fluxes.

Revised Manuscript Received on February 28, 2020.
***** Correspondence Author
Tajamul Hamid Mir*, Bachelors of technology: Chandigarh engineering college, Landran, Punjab, India.
Amritpal Singh, Associate Professor in RIMT University, Mandi Gobindgarh, Punjab, India.
Ajay Singh Rana, Associate Professor and Head of Department of Mechanical, RIMT University, Punjab, India.

Apart from this, various characteristics like simplicity of design, small temperature drop, and ability to transfer heat at high rate at different temperature levels makes the heat pipe system unique. Since 1966, heat pipes are being used in many industrial and research applications such as cooling of electronic components, spacecraft, energy conservation and much more.

The main regions of the standard heat pipe are shown in Figure 1.1. In the longitudinal direction the heat pipe is made up of an evaporator section and a condenser section. Further, adiabatic section can be included to separate the evaporator and condenser section. The cross-section of the heat pipe consists of the container wall, the wick structure and the vapor space. If the wick is not used in the heat pipe structure then it is known as thermosyphon. In thermosyphon, working fluid returns from condenser to evaporator by gravity action.

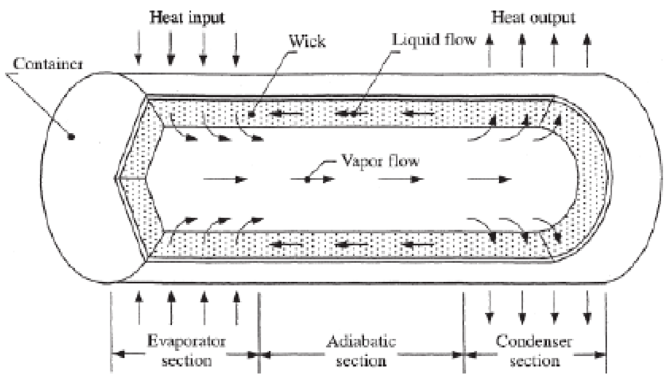


Figure 1.1 Main regions of heat pipe.

Heat transfer capability of a heat pipe is limited by five physical phenomena which are: the boiling limit, the sonic limit, the viscous limit, the capillary limit and the entrainment limit form. As shown in figure 1.2.

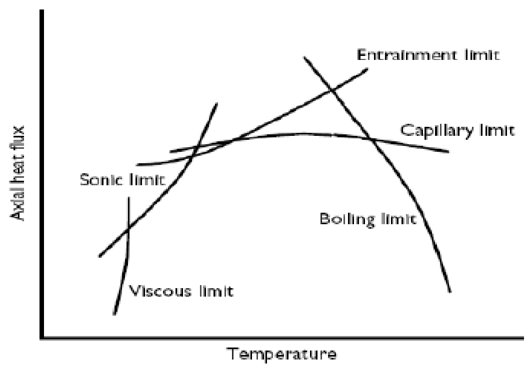


Figure 1.2 Limitations to heat transport in heat pipe.

II. OBJECTIVE AND MOTIVATION

(a) To analyze the fluid flow behavior over tube bundles in HPHE
(b) To analyze the pressure drop in HPHE
(c) To analyze the temperature distribution through HPHE system.

The motivation of the work is the analysis of the fluid behavior inside HPHE using CFD. Because Computational Fluid Dynamic (CFD) is very useful tool to predict the internal fluid flow behavior (single and multiphase flow), distributions of flow field and numerical simulations of any complex geometry. CFD can predict approximate results without carrying any experimentation on protocol or models and also tends to reduce the experiment cost. Due to these advantages CFD is nowadays widely used in industrial purposes and research areas.

III. METHODOLOGY

Analysis of temperature distribution Heat Pipe Heat Exchanger. Using Computer Aided Designing (CAD) tool a model was drawn of the HPHE. After modeling the flow domain, the physical parameters or boundary conditions were imposed over it. Then flow domain was discretized in number of elements or cell in other words tetra meshing was done in ANSYS ICEM. A complete discretized flow domain was imported into the ANSYS code to simulate the problem. Input parameters such as (mass flow rate, fluid properties, temperature, turbulent model, convergence criteria etc.) were given as per boundary conditions.

IV. MODELING AND SIMULATION

Computational fluid dynamics (CFD) is an important tool which is used to solve problems associated with fluid flow, heat flow and much more, by simulating the same in the computer. The CFD solves the problem by using various numerical approaches and by algorithms that finally helps to optimize the results without any experimentation on physical models or prototype. The flow visualization characteristic of the code inside the domain makes it a very power-full tool in the research field and other areas. CFD divides the flow domain into number of cells and solves the governing equations for each cell by converting partial differential equations into algebraic form.

The accuracy and validity of the CFD depends upon the number of factors: quality of mesh, models, type of boundary conditions, convergence criteria level and significance of the obtained results.

CFD simulation is a process to get the solution, information about the flow field inside, around and over any simple or complex geometry. The few steps have been derived to complete the process.

Preprocessing: It is the first stage of the simulation in which modeling and meshing is done over the flow domain.

Modeling: In the modeling the flow domain or geometry is created in the various codes like ANSYS, Pro-E, and Solid-works etc. As per the dimensions of the model. The modeling of the flow domain may be 2D or 3D.

Meshing: In the mesh generation, the flow domain is discretized into small number of elements. A fine mesh generates more accurate solutions than course mesh but it takes long computing time and more power is consumed.

Solver: It is the secondary stage of the CFD analysis process in which all the physical boundary conditions, flowing materials, flow parameters, convergence criteria are assigned or set up to get the solution.

Post Processing: It is final stage of the CFD-analysis process in which iterations are performed and after the completion of the iterations the results in the form of contour and vectors are plotted.

Turbulence is an unwanted vortices or unsteadiness in the flow areas of flowing fluid due to change in the properties like momentum diffusion, variation in pressure and velocity with respect to space and time. Some turbulence models include:

Realizable k-ε: The model is suitable for the complex shear flow pattern and also associated with transient or swirl flow.

Standard k-ε: This model is mostly used where the pressure gradient is very less and minor separation near the walls is occurred.

V. COMPUTATIONAL FLUID DYNAMIC MODELING

A model of HPHE has been modeled in Computer Aided Designing Software (Solid Works) v. 2017 as shown from Fig 5.1 to Fig 5.5.

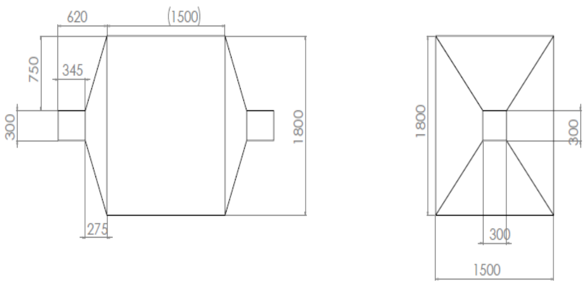
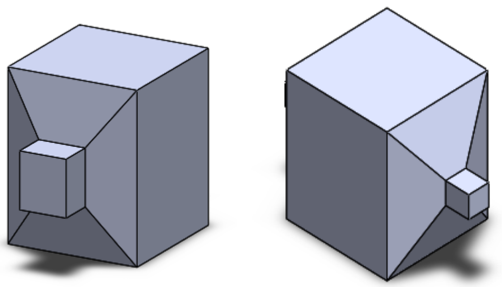


Figure 5.1 Dimensions in (mm)



(a) (b)

Figure 5.2 (a) (b)

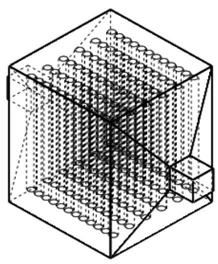


Figure 5.3 HPHE with 72 tubes

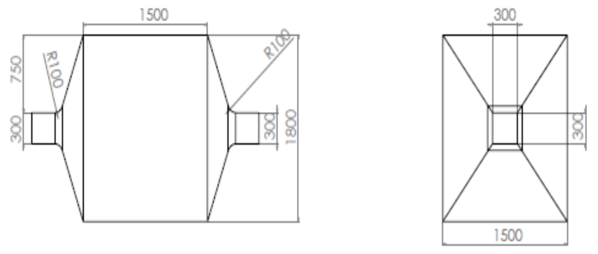


Figure 5.4 HPHE dimensions (in mm) with fillet near ends

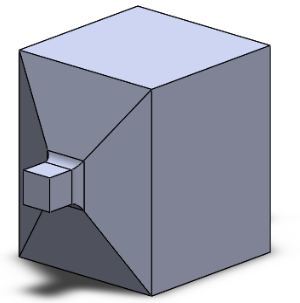
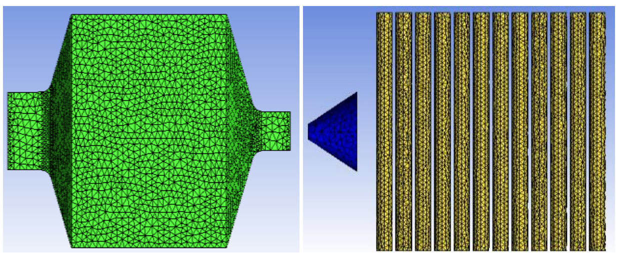


Figure 5.5 HPHE model with fillet near entry and exit.

After modeling the HPHE has been imported in the ANSYS software module ICEM-CFD, where the model has been discretized into small number of elements. The size and shape of the elements are given in the Table 5.1 and shown by figure 5.6.



(a) (b)

Figure 5.6 Meshing (a) on walls, (b) cone and tubes.

Table 5.1 Mesh size at different regions

Element size (mm)	Region/Part/Section	Type of elements
30	Inlet	Triangular shape elements
30	Outlet	
30	Wall-Outer-fillet-zone	
30	Wall-tubes	
100	Wall-Cone	
100	Wall-Outer	

Temperature distribution is evaluated using standard HPHE by enlarging the entrance area and by varying the cone angle as given detail in Table 5.2.

Table 5.2: Total number of elements

Geometry type	Case number/name	Total-Elements
Geometry with 30cm x 30cm inlet and outlet sections.	Case-1.(a)	929750
Geometry with 30cm x 30cm inlet and outlet sections with 10cm fillet near entrance of HPHE.	Case-1.(b)	943062
Geometry with enlarge inlet and outlet sections.	Case-2.(a)	943930
Geometry with enlarge inlet and outlet sections with 10cm fillet near entrance of HPHE.	Case-2.(b)	952690
Geometry with plate near the entrance.	Case-3.(a)	2148130
Geometry with plate near the entrance and with 10cm fillet near entrance of HPHE.	Case-3.(b)	1358846
Geometry with 30cm x 30cm inlet and outlet sections using 36.03^{o} cone near entrance of HPHE.	Case-4.(a)	936711
Geometry with 30cm x 30cm inlet and outlet sections using 36.03^{o} cone and 10cm fillet near entrance of HPHE.	Case-4.(b)	941942
Geometry with 30cm x 30cm inlet and outlet sections using 30^{o} cone near entrance of HPHE.	Case-4.(c)	935082
Geometry with 30cm x 30cm inlet and outlet sections using 30^{o} cone and 10cm fillet near entrance of HPHE.	Case-4.(d)	940656
Geometry with 30cm x 30cm inlet and outlet sections using 45^{o} cone near entrance of HPHE.	Case-4.(e)	933734
Geometry with 30cm x 30cm inlet and outlet sections using 45^{o} cone and 10cm fillet near entrance of HPHE.	Case-4.(f)	942951

Retrieval Number: D1830029420/2020©BEIESP
DOI: 10.35940/ijitee.D1830.029420
Journal Website: www.ijitee.org

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication

The physical boundary conditions which are imposed in this CFD simulation while using the ICEM and Fluent code are given in Table 5.3. Generally, the boundary conditions depend upon the flow field and type of flow domain. Simulation of the thermal analyses is carried out to predict the temperature distribution, to evaluate the velocity and total kinetic energy using the single phase model. Hot-air is used as a continuum inside the flow domain.

Table 5.3: Input models, parameters and boundary conditions for simulation.

Type	Description	Input
Model	Turbulent model	• Energy Equation • Standard k-ε-Realizable • Standard wall function type.
Materials	Properties	• Flowing fluid: hot Air or Gas • Density 0.88kg/m3 • Thermal conductivity: 95 W/m-k • Viscosity: 2e-05 kg/m-s • Solid domain
Operating conditions	Gravitational acceleration	• -9.81 in vertical downward
Boundary Condition	Inlet	• Mass flow inlet: 3.75kg/sec. • Thermal condition: 793k.
	Outlet	• Outflow
	Walls	• No-slip • Pipe Wall flux: -156865 W/m2 or -2178.680/72 W/m2 tubes
Solution controls	Discretization	• Pressure : standard • Momentum: first order upwind • Turbulence kinetic energy: first order upwind • Turbulence dissipation rate: first order upwind • Energy solution: second order.
Solution parameters	Defaults	

VI. RESULTS AND DISCUSSIONS

A. TEMPERATURE DISTRIBUTION.

The temperature detail at the inlet and outlet section of HPHE were evaluated while simulation process as given in Table 6.1.The temperature near the center region is higher as compared to the top and bottom zone of the HPHE and tube bundle in case-1(a){Refer Table 5.2}. The temperature drop found between inlet and outlet section of case-1.(a) and case-1.(b) is 19.40k and 16.70k respectively. The temperature drop is found between inlet and outlet section of case-2.(a) and case-2.(b) is 18.60k and 16.19k respectively. The results of case-3.(a) and case-3.(b) are almost similar to case-1.(a) and case-1.(b) respectively. Means less significance of the plate is observed. But with cone with 30degree temperature drops in upper section rather than lower section which is further improved in case-4(d). Moreover a uniform temperature distribution can be seen in case-4(e) and 4.(f) as shown in fig 6.1 to fig 6.4.

Table 6.1: Temperature drop between inlet and outlet.

Case-number	T_{IN} (k)	T_{OUT} (k)	ΔT (k)
CASE-1.(a)	793	773.60	19.40
CASE-1.(b)	793	776.30	16.70
CASE-2.(a)	793	774.4	18.60
CASE-2.(b)	793	776.81	16.19
CASE-3.(a)	793	774.20	18.80
CASE-3.(b)	793	774.23	18.76
CASE-4.(a)	793	775.62	17.37
CASE-4.(b)	793	774.30	18.69
CASE-4.(c)	793	774.32	18.67
CASE-4.(d)	793	774.80	18.19
CASE-4.(e)	793	774.04	18.95
CASE-4.(f)	793	774.40	18.59

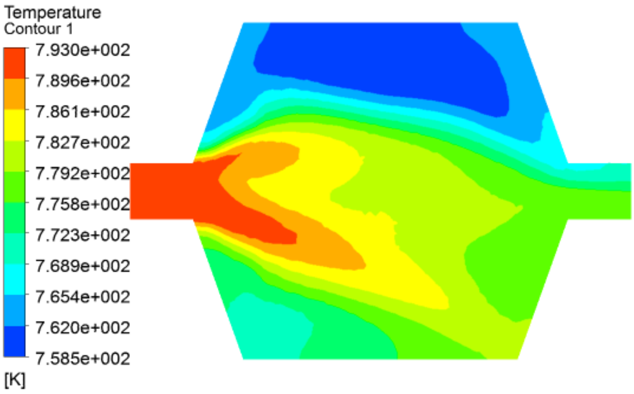


Fig 6.1 Case 4(c)

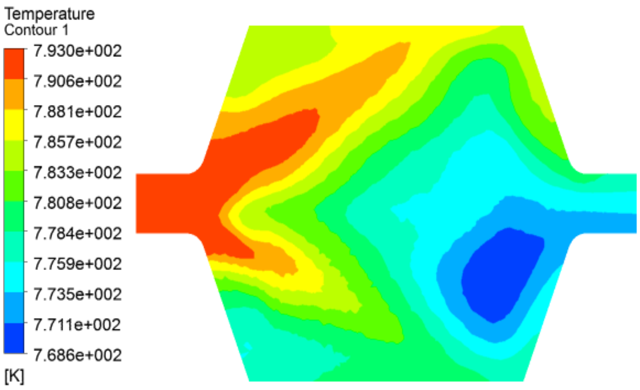


Fig 6.2 Case 4(d)

Retrieval Number: D1830029420/2020©BEIESP
DOI: 10.35940/ijitee.D1830.029420
Journal Website: www.ijitee.org

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication

International Journal of Innovative Technology and Exploring Engineering (IJITEE)
ISSN: 2278-3075 (Online), Volume-9 Issue-4, February 2020

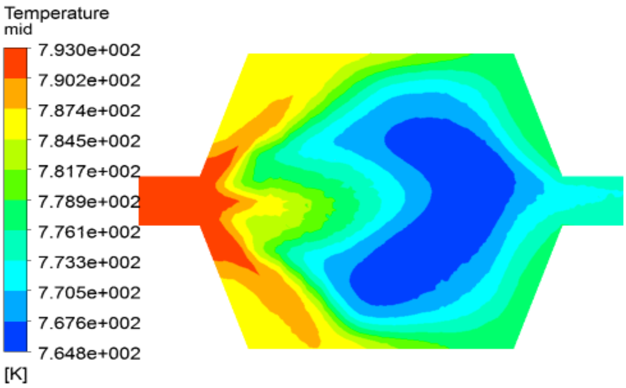


Fig 6.3 Case 4(e)

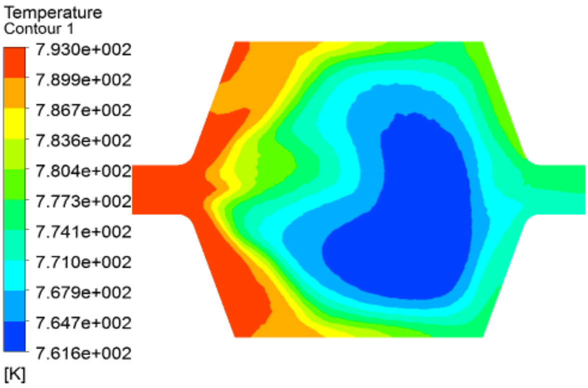


Fig 6.4 Case 4(f)

B. VELOCITY DISTRIBUTION.

The study of velocity distribution reveals that a long jet type flow is observed near the horizontal axis region for case-1.(a) but this jet type flow becomes shorten in case 1.(b) due to rounding the sharp corners near the entrance and exit zone of HPHE. The axial velocity in x-direction is decreased from 37m/s to 10m/s contrary increases (in y and z) directions for case 2(a). Similarly for case 2(b) velocity changes from 27m/s to 9 m/s at the center line of the HPHE. Velocity at the vertical mid line is slightly higher than case-1(a), case-1(b), case-2(a) and case-2(b). By using the cone hot-gases flow is divided into three different sections upper, middle and lower. With cone angle 36.03 degree maximum flow is imparted to central region shown but found as symmetric in following case. With cone angle 30 degree maximum flow is imparted to lower side but found as symmetric in case-4.(d). Although with cone angle 45 degree maximum flow is imparted symmetric but found as more uniform and symmetric over the tubes zone in case-4.(f).

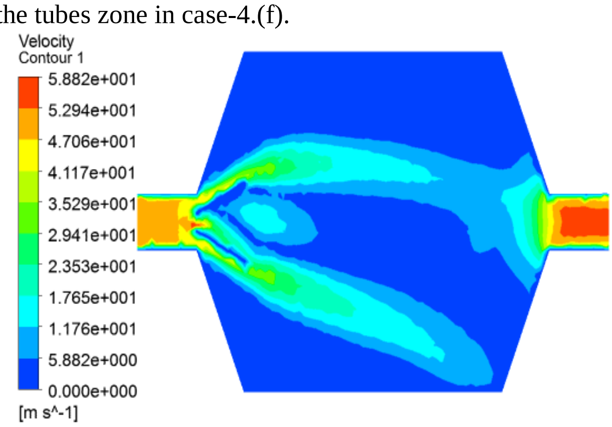


Fig 6.5. Case 4(c)

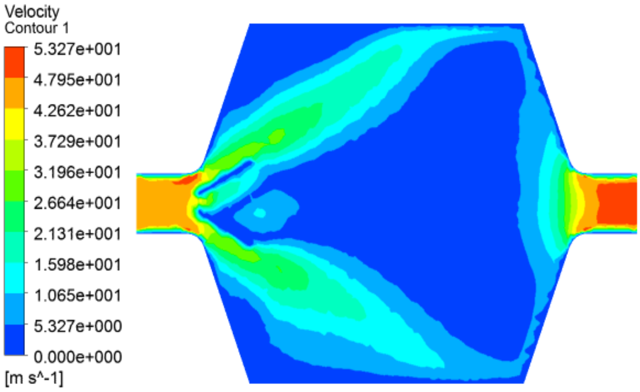


Fig 6.6. Case 4(d)

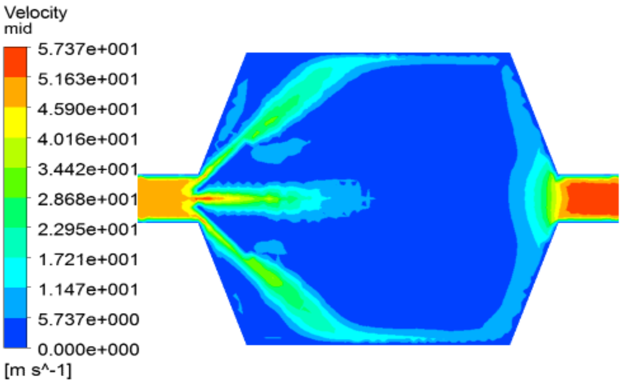


Fig 6.7 Case 4(e)

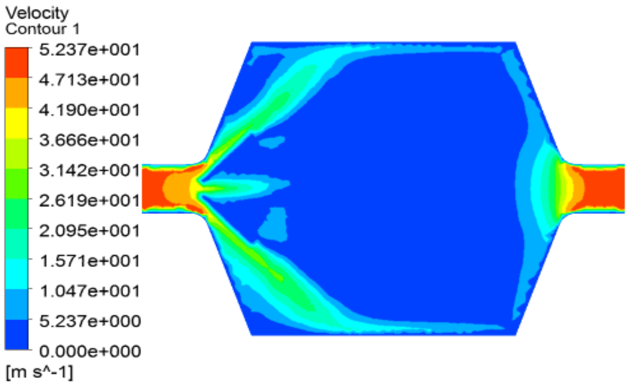


Fig 6.8 Case 4(f)

C. TOTAL KINECTIC ENERGY.

As the corners are rounded in case 1(b) the axial velocity (y and z directions) increases (rather than x-axis) which tends to create expanded or more turbulence kinetic energy exists. Hence TKE helps to make uniform temperature distribution over the tube as compare to case 1(a). The T.K.E decreases up to certain extent which tends to improve the temperature distribution over the tube bundle for case 2(a) and case 2(b). Very less T.K.E exists in case 2(b) due to enlarging and rounding near the entrance and exit zones. The maximum T.K.E is found as 46.26 j/kg in case-3(a) and 25.66 j/kg for case 3(b). Maximum T.K.E for case-4(a) is observed as 127j/kg which reduced to 54 j/kg in case 4(b). However with decreasing the con angle max. T.K.E is observed as 159.2 j/kg and 39.78 j/kg for case-4(c) and case 4(d) respectively. Similarly for 45 degree cone angle T.K.E is completely separated in three section upper middle and lower.

Hence max T.K.E for case 4(e) and case 4(f) is evaluated as 117.4 j/kg and 34.31 j/kg respectively.

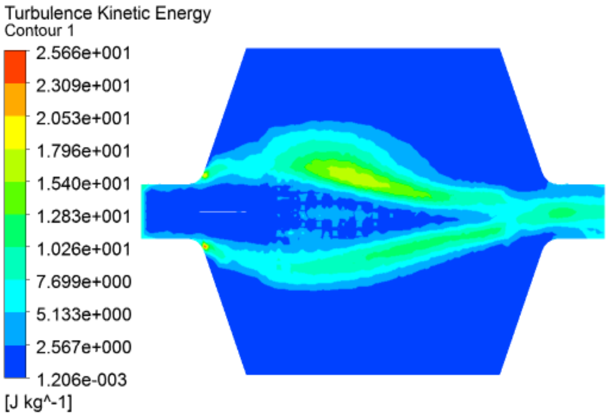


Fig 6.9 Case 3(b)

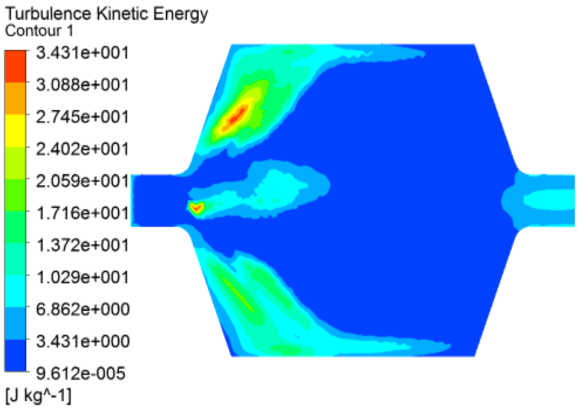


Fig 6.10 Case 4(f)

VII. CONCLUSIONS

As per this CFD analysis, high temperature distribution over the pipe-bundle is found at middle zone only which becomes wider with enlarging the inlet cross-section. By using plate, almost same temperature distribution is found as in case 1. But better temperature distribution was found with cone. However temperature distribution was improved with rounding the sharp comers. Jet type flow was observed in case-1.(a) which becomes smoother with enlarging inlet area and was imparted two and three sections by using plate and cone respectively. Although round corner have made uniform flow for all the cases just because of changing the turbulence kinetic energy near the round zones. Maximum Pressure drop is found as 2142.741 Pascal whereas minimum Pressure drop is observed as 482.22. It is observed that pressure drop is reduced up-to certain extent with the help of round corners at entry and exit rather than sharp corners.

REFERENCES

1. Grover G.M., (1966), "Evaporation – condensation heat transfer device", US Patent No. 3229759.
2. Gaugler R.S., (1994), "The heat pipe is potrayed as applied to a refrigeration framework", US Patent No. 2350438.
3. Peterson G.P., (1994), "An introduction of heat pipes, Modeling, Testing and Applications", John Wiley and Sons, New York: 245.
4. Faghri A., (1995), "Heat Pipe Science and Technology", Taylor and Francis, USA: 134.
5. Martínez, Francisco Javier Rey, Mario Antonio Álvarez-Guerra Plasencia, Eloy Velasco Gómez, Fernando Varela Díez, and Ruth HerreroMartín. "Design and experimental study of a mixed energy recovery system, heat pipes and indirect evaporative equipment for air conditioning." Energy and Buildings 35, no. 10 (2003): 1021-1030.
6. Yau, Y. H., and A. S. Tucker."The execution study of a wet six-row heat-pipe heat exchanger working in tropical buildings." International Journal of Energy Research 27, no. 3 (2003): 187-202.
7. Loh, C. K., Enisa Harris, and D. J. Chou. "Relative study of heat pipes performances in different orientations." In Semiconductor Thermal Measurement and Management IEEE Twenty First Annual IEEE Symposium, 2005., pp. 191-195. IEEE, 2005.
8. Martín, R. Herrero, FJ Rey Martínez, and E. Velasco Gómez. "Thermal comfort analysis of a low temperature squander vitality recuperation framework: SIECHP." Energy and Buildings 40, no. 4 (2008): 561-572.
9. Yau Y.H. and Ahmadzadehtalatapeh M., (2010), "A audit on the utilization of horizontal heat pipe heat exchangers in air conditioning systems in the tropics", Applied Thermal Engineering, 30: 77–84.
10. Firouzfar E., Soltanieh M. , Noie S. H. and Saidi M. H., (2011), "Use of Heat Pipe Heat Exchangers in Heating, Ventilation and Air Conditioning (HVAC) Systems" , Scientific Research and Essays, 6(9):1900-1908.
11. Hashemian, Mehran, SamadJafarmadar, and HamedSadighiDizaji. "A comprehensive numerical study on multi-criteria design analyses in a novel form (conical) of double pipe heat exchanger." Applied Thermal Engineering 102 (2016): 1228-1237.
12. Kim, Kyung Mo, Yeong Shin Jeong, In Guk Kim, and In Cheol Bang. "Development of Passive In-Core Cooling System for Nuclear Safety Using Hybrid Heat Pipe." Nuclear Technology 196, no. 3 (2016): 598-613.
13. Kusuma, M. Hadi, Nandy Putra, SuripWidodo, and AnharRizaAntariksawan. "Simulation of heat flux effect in straight heat pipe as passive residual heat removal system in light water reactor using RELAP5 Mod 3.2." In Applied Mechanics and Materials, vol. 819, pp. 122-126. Trans Tech Publications, 2016.
14. Ramos, Joao, Alex Chong, and HussamJouhara. "Experimental and numerical investigation of a cross flow air-to-water heat pipe-based heat exchanger used in waste heat recovery." International Journal of Heat and Mass Transfer 102 (2016): 1267-1281.
15. Saber M. H. and Ashtiani H. Mazaher., (2016), "Simulation and CFD Analysis of Heat Pipe Heat Exchanger Using Fluent to Increase of the Thermal Efficiency", Continuum Mechanics, Fluids, Heat, 183-189.
16. Saeedan, Mahdi, Ali Reza SolaimanyNazar, YaserAbbasi, and Reza Karimi. "CFD Investigation and neutral network modeling of heat transfer and pressure drop of nanofluidsin double pipe helically baffled heat exchanger with a 3-D fined tube." Applied Thermal Engineering 100 (2016): 721-729

AUTHORS PROFILE

Tajamul Hamid Mir, Masters of technology: RIMT University. Bachelors of technology: Chandigarh engineering college, Landran, Punjab.

Amritpal Singh, Masters of technology: Guru Nanak Dev Engineering College, Ludhiana, Punjab. Bachelors of technology: Giani Zail Singh College of engineering, Punjab. Presently he is working as an Associate Professor in RIMT University, Mandi Gobindgarh, Punjab.

Ajay Singh Rana, Masters of technology: Thapar University, Patiala, Punjab. Bachelors of technology: Nagpur University in 1999. Presently he is working as an Associate Professor and Head of Mechanical Department in RIMT University.