International Journal of Computer Applications (0975 – 8887)
International Conference on Advances in Emerging Technology (ICAET 2016)

An Investigation into the Performance of Heat Pipe Heat Exchanger using CFD

Gurtej Singh
M.E Student
Mechanical Engg. Department
Thapar University,Patiala
Punjab, India

Lal Kundan
Assistant Professor
Mechanical Engg. Department
Thapar University, Patiala
Punjab, India

Paramjit Singh Bilga, PhD
Professor
Mechanical Engg.
Department, GNDEC,
Ludhiana Punjab, India

ABSTRACT

Heat pipe is a device that transports heat with minimal temperature drop using the latent heat of fluid in a closed container. In vertical type named as thermosyphon (Wickless), capillary action is replaced by the gravitational force, this condenser is placed above evaporator. In this paper, to investigate the effect of inlet air temperature and the air mass flow rate was studied.The temperature was varied from 100 deg. to 200 deg. by taking a step size of 50 deg. Step Inlet air mass flow rate from evaporator was varied from 0.03 kg/sec to 0.09 kg/sec with 0.03 kg/sec step. Heat pipes were assumed as a solid rod of constant conductivity. Two geometries were made to study the effect no. of rows on the performance of heat pipe heat exchanger. Boundary conditions for both the geometries were kept constant and in second geometry pipes taken in staggerd configuration instead of aligned.

Keywords

Heat pipe, wickless,staggerd, thermosyphon, contours, simulation, superconductors, humid, geometries.

1. INTRODUCTION

Heat Pipe is a device which uses the concept of Perkins tube to transfer the heat. Heat pipes which use, the wick structure to be purposed by Gauglar [1] and later Grover [2] given its name of heat pipe due to its exceptional thermal conductivity. Heat pipe is a two phase device which uses the latent heat of fluid to transfer the heat. It consist of three parts evaporator, condenser and adiabatic section. In the horizontal heat pipes wick is used to pump back the liquid from condenser section to evaporator section. In case of vertical thermosyphon (gravity assisted heat pipes) condenser is placed above the evaporator and liquid come back to evaporator with gravity force and cycle continues. The heat pipe technology is being used in numerous application from cryogenic to high temperature. Heat pipes are also termed as superconductors. Since its inception lot of research has been carried out on the heat pipes for heat recovery application, dehumidifying pipes for hot and humid climates. Heat pipes are also used in space applications. Amir Fahgiri [3] developed a heat pipe to control the human body temperature. H. Jouhara carried out its research on for medium temperature application for air-water heat pipe heat exchanger and successfully validated it with numerical simulation using Fluent [4].

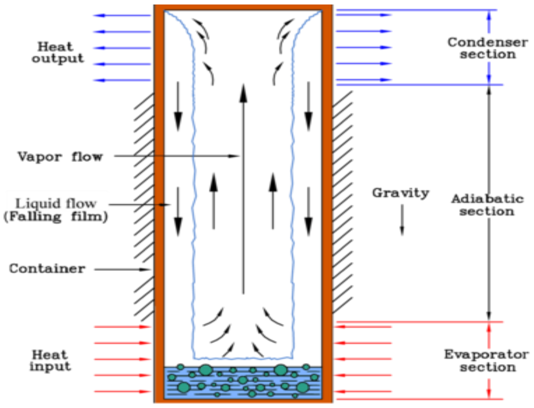

Figure 1. [Source: https://www.thermalfluidscentral.org/]

Yat H. Yau studied the effect of different fluid charges on the performance of heat exchanger [5]. T.S. Jadhav performed the research on the energy analysis on the air conditioning using heat pipe heat exchanger [6]. Dobson performed his research on the high temperature nuclear reactor technology to eliminate the risk of tritium into the coolant stream and also reducing the one heat exchanger by using his novel concept of heat pipe heat exchanger [7]. M. H. Saber performed a simulation to increase the thermal efficiency of heat pipe heat exchnager by using CFD [8]. Babek Rashidain developed a Matalb program to use the heat exchanger for heat recovery systems [9].

In this paper heat pipes were assumed as a solid rod of constant conductivity. Two geometries were made to study the effect no. of rows on the performance of heat pipe heat exchanger. Boundary conditions for both the geometries were kept constant and in second geometry pipes taken in staggerd configuration instead of aligned.

2. GEOMETRIC MODELING

Two geometries of the same length were made in the Ansis design modular. In geometry 1 three heat pipes were used in aligning configuration, while the in geometry 2, two rows were made consisting of 6 heat pipes in a staggered configuration. Pitch of heat pipes in heat exchanger was kept constant in both the geometries. Other important parameters related to the geometries are listed below in the table1.

Table 1. Parameters of geometric model

Geometry parameters	Geometry 1	Geometry 2
Length of Heat Pipe (HP)	80 mm	80
Diameter of HP	14 mm	14

No of HP’s	3	6
Length of Evaporator	40 mm	40
Length of Condenser	40 mm	40

Geometry 1

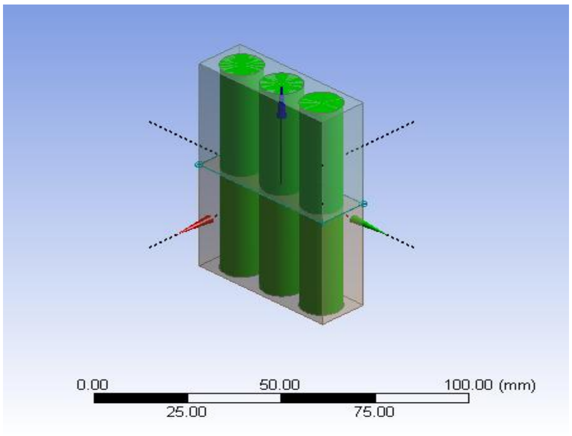

Figure 2. First geometry

Geometry 2

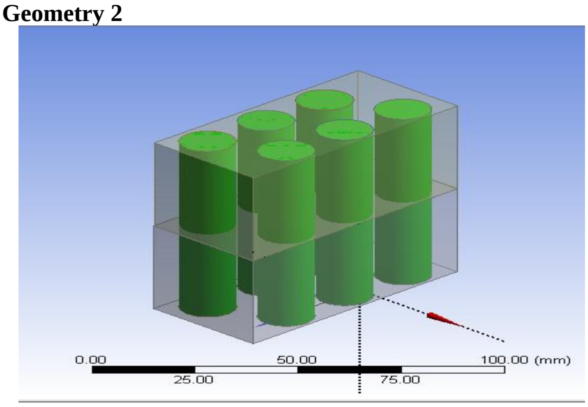

Figure 3. Second geometry

3. MESHING GENERATION

Meshing is very important parameter in order to accurately simulate any system. In this work all the meshing parameters were varied in order obtain the required mesh. The grid dependency test was also performed to validate the mesh. In the grid dependent test all the meshing methods the value of the maximum skewness was checked. . Inflation was added to capture the in turbulent flow near the wall Then the sizing and assembly parameters were also varied to get the required mesh

All the parameters involved in the meshing are listed in the tables below for both the geometries.

Mesh Dependency

Geometry 1:

Table 2. Parameters of meshing for geometry 1

Method	No. of Cells	Types of Cels	Skewness	Time/ Iter.
Auto Mesh	4704	Tetra+Hexa	Max. 0.46	10-12 sec
Cutshell	12892	Sqr+ Hexa	Max. 0.94	15-20 sec
Tetrahedron	62194	Tetra + Hexa	Max. 0.91	18-21 sec

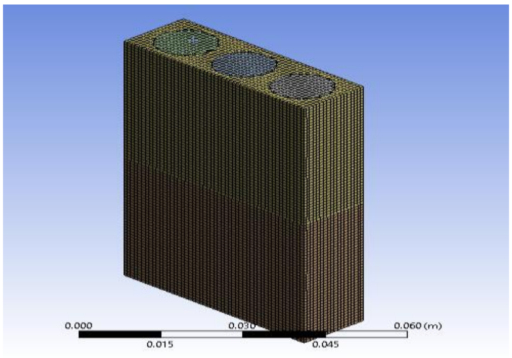

Figure 4. Meshing of first geometry

Geometry 2:

Table 3. Parameters of meshing for geometry 2

Method	No. of Cells	Types of Cells	Skewness	Time/ Iter.
Auto Mesh	12117	Tetra+Hexa	Max. 0.46	15-20 sec
Cutshell	82136	Sqr+ Hexa	Max. 0.936	25-32 sec
Tetrahedron	614227	Tetra + Hexa	Max. 0.923	40-59 sec

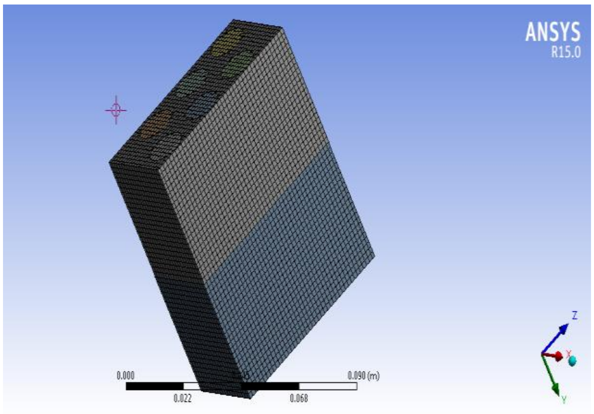

Figure 4. Meshing of second geometry

4. RESULTS AND DISCUSSIONS

In this work, numerical simulation of heat pipe heat exchanger has been performed. Heat pipe was assumed as a solid rod of constant conductivity. This analysis was carried out on the copper water heat pipe(Thermosyphon) to investigate the effect of inlet air temperature and the air mass flow rate was studied.The temperature was varied from 100 deg. to 200 deg. by taking a step size of 50 deg. Step Inlet air mass flow rate from evaporator was varied from 0.03 kg/sec to 0.09 kg/sec with 0.03 kg/sec step.The mesh dependency and grid independence test was performed in order to reduce the error in the work.The temperature contours for condenser and evaporator was plotted. Boundary conditions used in this simulation are :

Parameter Varied:
Hot fluid inlet temperature-
100-200 deg
Velocity of hot fluid inlet-
2.6 m/sec – 4.6 m/sec
Cold Fluid inlet Temperature
14 deg (constant)
Cold fluid inlet Velocity
3.6 m/secc (constant)

Operating conditioning for both the geometries were kept constant and values were varied according to the data given in the above table.

Inlet Temperature of Cold Fluid- 14 deg

Inlet Temperature of Hot Fluid - 100 deg

Velocity of Cold Inlet – 3.6 m/sec

Velocity of Hot Inlet- 2.6 m/sec

Contours for Temperature for geometry 1

Case 1

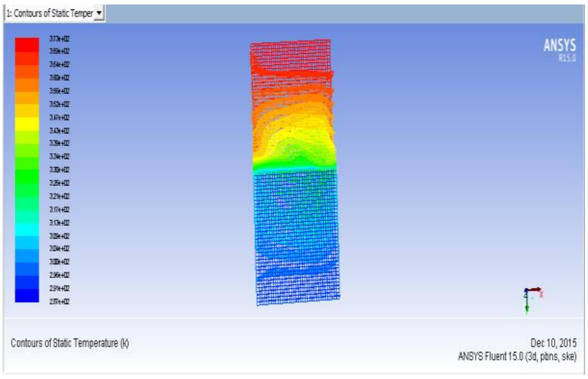

Figure 5. Temperature contour for case 1

[Min : 283 K , Max 423 K]

Case 2

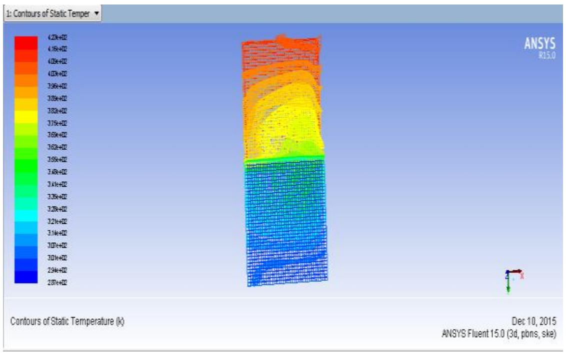

Figure 6. Temperature contour for case 2

[Min : 289 K, Max 438K]

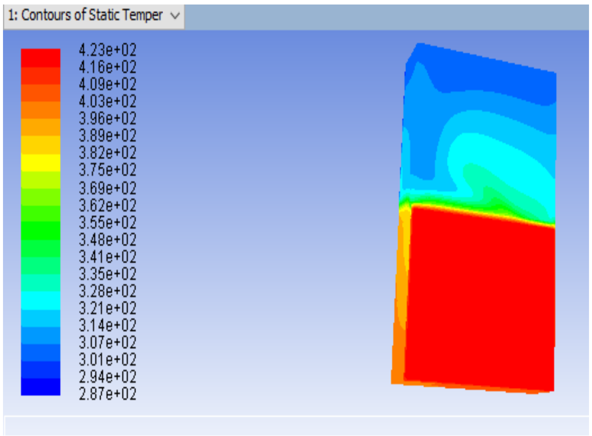

Figure 7. Temperature contour for case 2

[Min : 283 K , Max 423 K]

Contour of Internal Energy

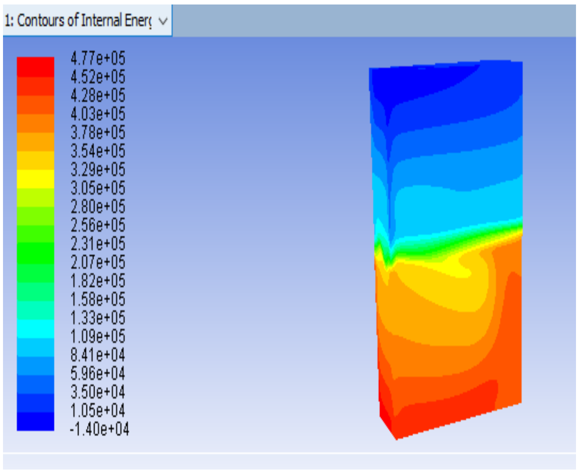

Figure 8. Internal energy contour

Contours for Turbulent Kinetic energy and Velocity Vector

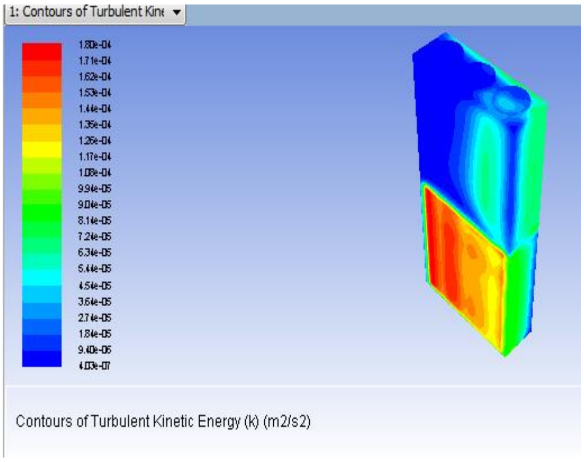

Figure 9. Turbulent Kinetic energy contour

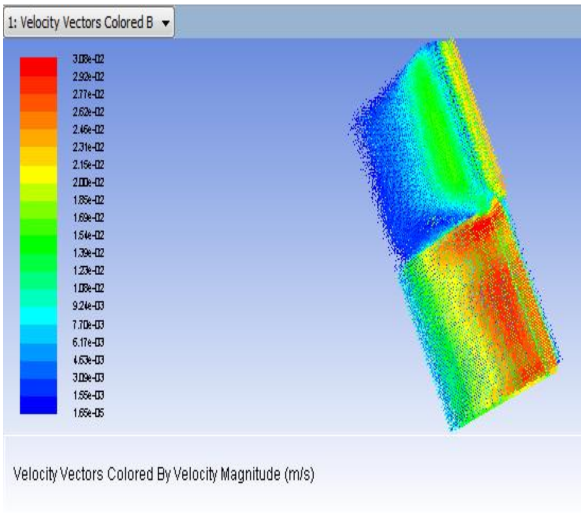

Figure 10. Velocity vector contour

Temperature Contours for Geometry 2

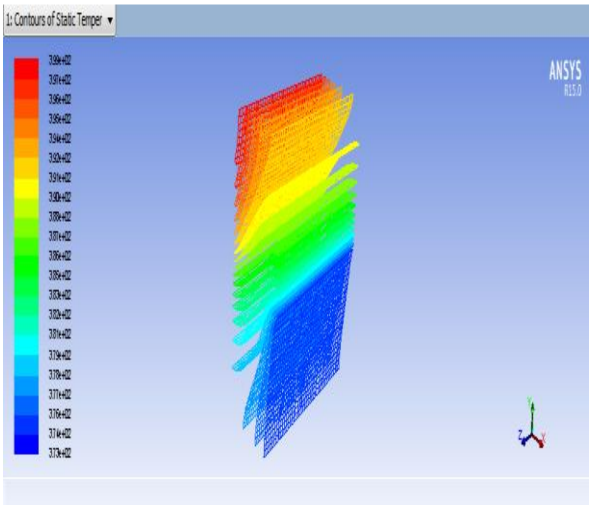

Figure 11. Temperature contour (geometry 2)

[Min 423K, Max 432K]

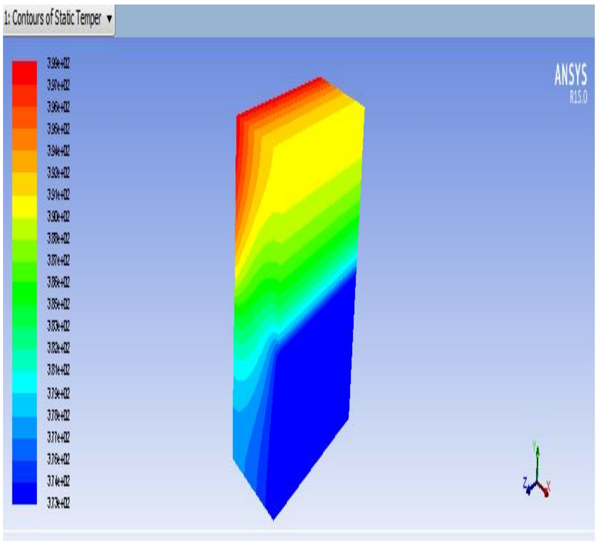

Figure 12. Temperature contour 2

[Min 423K, Max 432K]

Turbulent profile and Wall Temperature

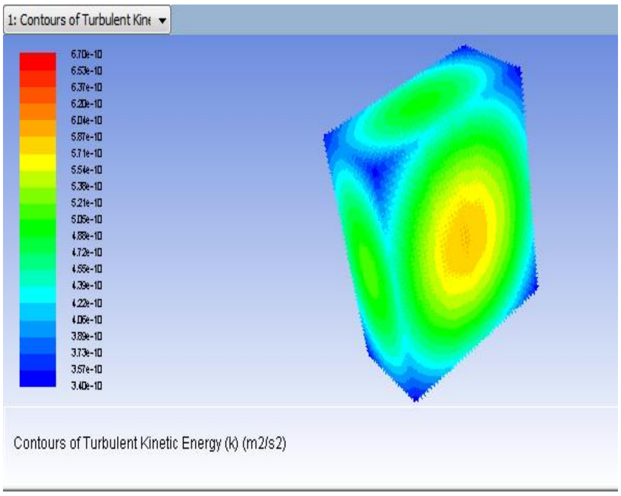

Figure 13. Turbulent profile contour

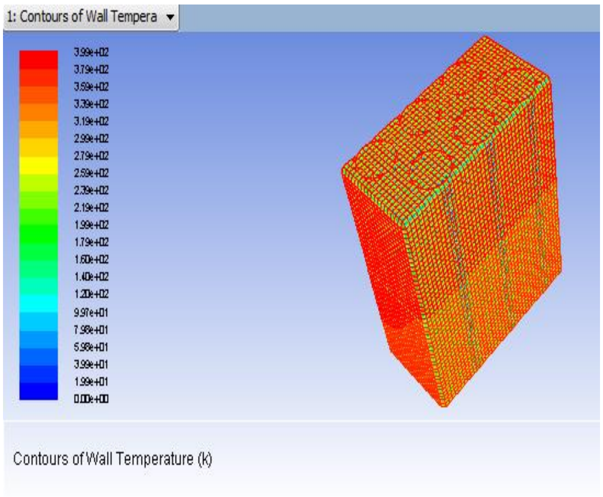

Figure 14.Wall Temperature contour

[Min 423K, Max 432K]

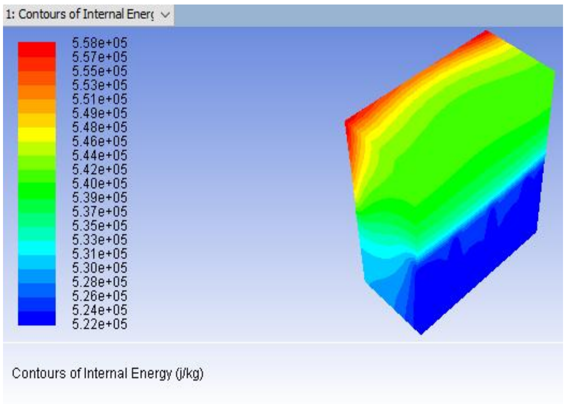

Figure 15. Internal energy contour

[Min 522362.4 j/kg, Max 558439.5 j/kg]

Plots for Geometry 1 and 2 for Temperature, Turbulence versus variation of other parameters:

Geometry 1

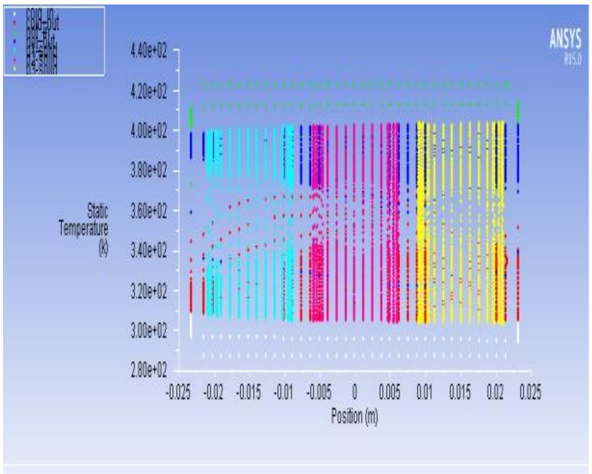

Figure 16. Static temperature contour (geometry1)

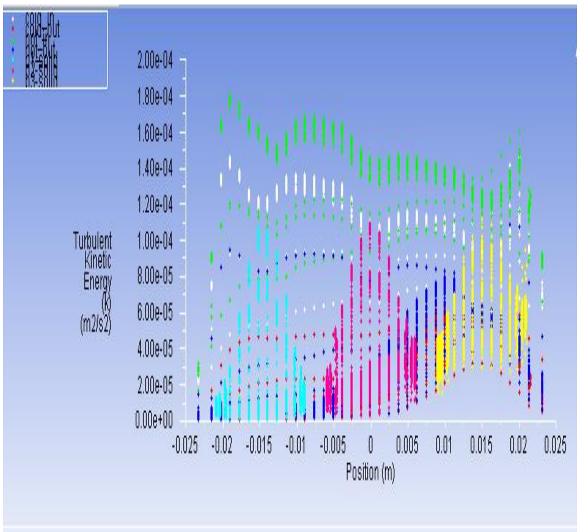

Figure 17. Turbulent Kinetic energy contour (geometry 1)

Geometry 2

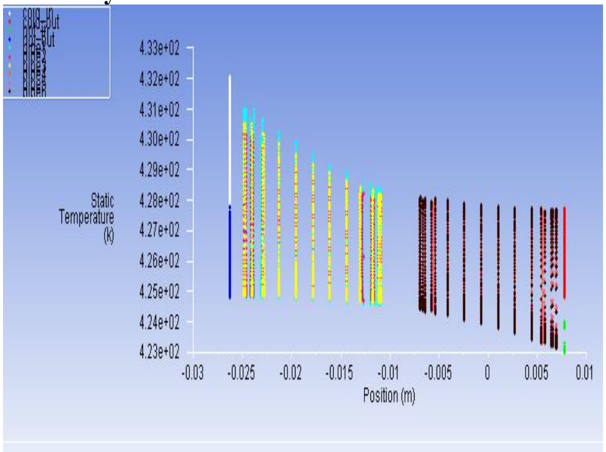

Figure 18. Static temperature contour (geometry2)

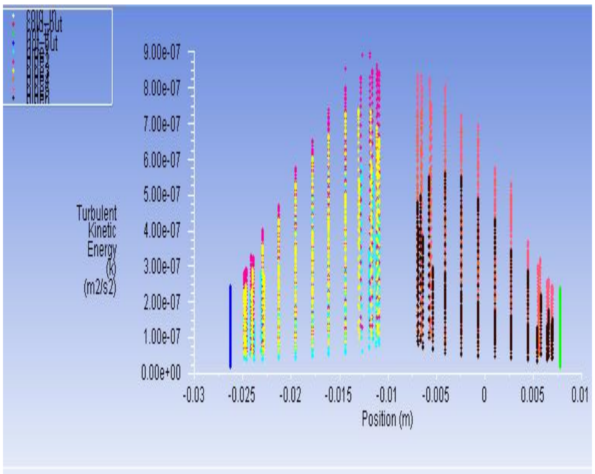

Figure 19. Turbulent Kinetic energy contour (geometry 2)

5. CONCLUSION

Heat pipes can be used for heat transfer applications with great efficiency. In this work heat pipe has been used in heat exchanger and its numerical simulation has been performed. The higher heat transfer rate is achieved at higher inlet temperatures and mass flow rates in the simulations carried out. Out of the two geometries used, for the 2nd geometry, K-epison gives the fast convergence over the k-omega. Meshing time increases with the increase in the number of meshing elements. The change in the maximum stagnation temperature of the geometries were not much varied. The values of turbulent kinetic energy for geometry 2 was very high due the increase in the no of rows of heat exchnager. The position of heat pipes also effected the heat transfer. As the heat pipes were considered as soild rods their conductivity was not varied along the length in the simulations

6. REFERENCES

[1] R.S. Gaugler, U.S. Patent, 2350348 (1944)

[2] G.M. Grover, U.S. Patent, 3229759 (1966)

[3] A. Faghri, US Patent 5269369

[4] H. Mroue , J.B. Ramos , L.C. Wrobel , H. Jouhara , Appl. Therm. Eng., **78**, 339-350 (2015).

[5] Y. H. Yau and M. Ahmadzadehtalatapeh, Heat Transf. Eng., **35**,1539–1548 (2014).

[6] T.S. Jadhav, M.M. Lele, Eng. Sci. Technol. an Int. J., **18**, 669-673 (2015).

[7] Ryno Laubscher, Robert T. Dobson, Appl. Therm. Eng., **61**, 259-267 (2013).

[8] M. H Saber, H. Mazaher Ashtiani, Contin. Mech. Fluids, Heat.

[9] Babek Rashidian, *Proc. 2nd WSEAS Int. Conf. Eng. Mech. Struct. Eng. Geol.*, 114-119

[10] https://www.thermalfluidscentral.org/