CFD Analysis of Dimple Tube and Corrugated Tube and Validated Results with Experiments

Jaydip Khade, Yash Varkute[1], Jay Varkute[1], Sanket Gare[1]

[1]*Department of Mechanical Engineering, Alamuri Ratnamala Institute of Engineering and Technology, Shahapur*

---***---

Abstract- CFD analysis of dimple, corrugated and plain tube is done in this study. Here, voltage is varying from 60-100 v. There is wide increase in heat transfer from corrugated tube after 80 Volts. Hence, we have high heat transfer in the corrugated tube at higher voltages. Also, We use dimple tube in this study and we try to find out the heat transfer coefficient of dimple tube and corrugated tube and plane tube. we use natural convection set up for this study and we find out Corrugated tube shows good results as compared to other tubes. Also, CFD of dimple is done by using ANSYS fluent.

Keywords: CFD Analysis, Corrugated Tube Heat Exchanger, Dimple Tube

1.INTRODUCTION

To lower the heat flow due to convective heat transfer you can reduce the area in contact with the fluid, or decrease the convective heat transfer coefficient. Heat transfer is the area that deals with the mechanism responsible for transferring energy from one place to another when a temperature difference exists. Natural convection is one of the most economical and practical methods of cooling and heating. Natural convection is caused by temperature or concentration induced density gradient within the fluid. Natural convection flow occurs because of influence of gravity forces on fluids in which density gradients have been thermally established.

In the study of heat transfer, both equilibrium and non-equilibrium processes are encountered. The science of heat transfer allows us to determine the time rate of energy transfer caused by the more practical non-equilibrium processes. With the growing sophistication in technology and with the increasing concern with energy and the environment, the study of heat transfer has, over the past several years, been related to a very wide variety of problems, each with its own demands of precision and elaboration in the understanding of the processes of interest. Areas of study range from Atmospheric, geophysical, and environmental problems to those in heat rejection, space research and manufacturing systems.

Heat transfer by convection has wide range of engineering applications of practical and functional significance. The mechanism is found very commonly in everyday life and includes central heating, air conditioning, electronic cooling, cooling towers in power plants and in industries, steam turbines, heat exchangers, pipe flow etc. It is mostly required to predict the significant energy change that takes place because of temperature difference. Convective heat transfer is largely categorized as: Free/Natural and forced. Free convection refers to fluid motion by buoyant forces arising due to density gradients which are a result of temperature gradients. Whereas in, forced convection, the flow of the fluid is enhanced by external sources. The present work focuses on a free convection configuration investigating an aspect yet to be discovered. By proper experimentations, the postulations of dependable variables viz., tube surface orientation, heat source power input and enclosure effects with the aid of heat transfer coefficient. The interest in this class of problems is specifically driven by the need to have better understanding of convective heat transfer occurring over materials. The contributions have been reported in several reviews like Cess (1961), Szewczyk (1964), Whitaker (1972), Cooper et al., (1986), Copeland (1998), Kim et al., (1999). The works provide an excellent review on the developments up to the end of the century. Cheng et al., (2002) investigated natural convection on a flat plate with inertia effect and thermal dispersion. They noted that the rate of unsteady heat transfer can be accelerated by the thermal dispersion. Sartori (2006) studied equations of the natural convection heat transfer coefficient over flat surfaces. He reasoned that there must be a decay of heat transfer coefficient along the plate dimension in the wind direction. Abreu et al., (2006) worked on similarity solutions of boundary layer flows in free and forced convection for evaluation of the coupled effects of heat and mass transport. They showed that all convection cases depend on different similarity variables. Yao et al., (2008) studied natural convection due to a non-Newtonian fluid past a flat plate using a modified power-law viscosity model. They showed that the most significant effects occur near the leading edge gradually tailing off far downstream. Seyyedi et al., (2012) probed effects of a splitter plate and an inclined square cylinder with 45° inclination on 2-D unsteady laminar flow and heat transfer in a plan channel using the lattice Boltzmann method.

2. LITERATURE REVIEW

Compared with vertical plates, inclined and horizontal plates have reduced fluid Velocities along the plate. One might expect that there is a reduction in convection heat transfer. Whether, in fact, there is such a reduction depends on whether one is looking at the top or bottom surface of the plate. On top of a cold inclined plate, the reduction of the gravity that acts in the direction parallel to the plate reduces the convection heat transfer. However, on the bottom of the same plate gravity moves fluid from the surface and a boundary layer development is interrupted by the discharge of parcels of cool fluid from the surface. The cool fluid close to the lower surface is continuously replaced by hot fluid by a three-dimensional flow, which reduces the effective thermal boundary layer thickness and increases the convection heat transfer although the gravity component along the plate is smaller than for a vertical plate. An equivalent discussion can be made for a heated inclined plate. For the top of the cold plate and the bottom of the heated plate and $0^0 < \theta < 60^0$ the same Nusselt number correlations as for vertical plates can be used if g is replaced by g cos θ. For the opposite surfaces, literature should be consulted.

For horizontal plates the following correlations are recommended for upper surface of heated plate or lower surface of cooled plate

$Nu=0.54\ Ra^{1/4}\ (10^4 < Ra < 10^7)$

and for lower surface of heated plate or upper surface of cooled plate

$Nu=0.27\ Ra^{1/3}\ (10^7 < Ra < 10^{11})$

The pioneering work on the problem of natural convection in vertical parallel plate is traced back by W. Elenbass [2] (1942) who analysed the laminar natural convection heat transfer in a smooth parallel plate vertical channels without internal bodies & a detailed study of the thermal characteristics of cooling by natural convection was reported. Followed by many experimental, theoretical & numerical investigations for both laminar & turbulent flow regimes. Only the configuration of natural convection in vertical channel with internal objects will be reviewed here.

I.H.Toruka [3] performed experimental study free convection from a cylinder array arranged in a vertical line between parallel walls. Empirical formulas were proposed to predict the average heat transfer coefficient. An enhancement of average Nusselt Number for an entire array of cylinders between parallel walls by 10% to 15% in comparison with the case of free space.

Y.Shen, P.Tong[4][1998], explained light scattering experiment of turbulent convection in water is carried out in a convection cell with rough upper and lower surfaces. The vertical heat flux is found to be increased by ,20% when the Rayleigh number becomes larger than a transition value. The experiment reveals that the main effect of the surface roughness is to increase the emission of large thermal plumes, which travel vertically through the central region. These extra thermal plumes enhance the heat transport, and they are responsible for the anisotropic behaviour of velocity fluctuations at the cell centre.

Y.B. Du[5](1998), performed novel convection experiment is in a cell with rough upper and lower surfaces. The heat transport across the rough cell is found to be increased by more than 76%. Flow visualization and near wall temperature measurements reveal new dynamics for the emission of thermal plumes. The discovery of the enhanced heat transport has important applications in engineering and atmospheric convection.

S.M.Guo,C.C.Lai[9](2000) studied influence of surface roughness on heat transfer coefficient and cooling effectiveness for a fully film cooled three dimensional nozzle guide vane(NGV) has been measured in a transonic annular cascade using wide band liquid crystal and direct heat flux gases(DHFG).These techniques have been used to measure the heat transfer coefficient and film cooling effectiveness in a transient blow down tunnel under extreme conditions of transonic flow and high heat transfer coefficient(400-1600 W/m2K). The roughness is shown to increase the heat transfer coefficient significantly, particularly in regions near the rear of pressure and suction surfaces where the non-dimensional roughness Re reaches to a value of high as 40. The differences in heat transfer to the rough and smooth point to a requirement to conduct further research including the effect of roughness shape, height, and pattern.

Rossano Comunelo[10](2005),his work deals with heat transfer coefficient ―h‖ of a isothermal vertical plate with H = 0.15 m. The neighbourhood surfaces influence in that coefficient is aimed with simulation and standard experimentation. A novel technology to measure the heat flux, calling ―Tangential Heat Flux meter‖ is applied and simulation with a CFD commercial code was performing. Five heat fluxmeters were glued on the vertical plate, heated 20 0C over the air temperature. The neighbourhood and air temperature were maintained constants. The distance between the plate and base wall (floor) was changed as well as the distance between the plate and backside wall. Through simulation results will be compare with experimental. The result expected is an increasing of heat transfer coefficient, very usefully in heat exchange devices.

J.W. Zhou , Y.G. Wang [11](2008) studies that determine the influence of unsteadiness on flat plate impinging jet heat transfer implicitly assume that the effect of unsteadiness found on smooth impingement surfaces also holds on surfaces with certain obstacles on them. To test this assumption a single roughness element was added to an otherwise smooth surface, and it was found that the steady heat transfer was almost the same as that for a totally smooth surface. The effect of unsteadiness, however, can be fundamentally different when roughness elements are added to a smooth surface. Slight changes in the surface geometry thus can have strong impact with respect to the effect of unsteadiness on heat transfer under impinging jets and cannot be neglected a priori. Varol et al. [21] studied natural convection in a triangular enclosure with flush mounted heater on the wall. The study of natural convection heat transfer in triangular enclosure was analysed numerically for different parameters, including the aspect ratio of triangle (AR = 1 and 0.6), Rayleigh number and both length and position of heater. The results showed that the flow and temperature fields are affected by the shape of enclosure and Rayleigh numbers play an important role on them. Both position and location of heater affect the flow circulation and heat transfer.

3. MATERIALS AND METHODS

In the light of the review of literature studied on tube surfaces, it is seen that the problem of heat transfer by natural convection from surface has been studied theoretically and experimentally by several investigators. It has been reported that provision of a dimple tube and corrugated tube enhances the heat transfer performance. parametric study of dimple tube and corrugated tube, but no clear conclusions were drawn because of too many influencing parameters involved. Review of the existing literature shows that the experimental work that has been done on tube surfaces have incorporated many non-realistic conditions such as the assumption of isothermal condition of tube surface.

Investigation for exploring the possibility of modified tube surfaces was suggested by many researchers. However, tube with notch were not investigated thoroughly. It is observed in the literature that very few investigators have reported experimental results pertaining to modified tube surfaces. Thorough investigation of the problem of natural convection heat transfer in laminar flow by dimple tube on brass has not been done so far. Therefore, it is decided to investigate various types of tubes such as dimple, corrugated and plane. Both experimentally as well as theoretically under present work. In the next chapter experimentation carried out in this project work is presented.

From the early research work and literature survey, there is establishment of tubes with modifications for lengthwise. There is rectangular enclosure for entry of air in case of natural convection. The air coming inwards gets heated as it moves towards the end of the tube, as well as it rises due to decrease in density. This thing is checked for other tubes like dimple and corrugated. With this view point, it is proposed to investigate the following types of tubes configurations which are shown in Fig. 3.1 (a) Dimple (b) Corrugated (c) Plane. Following methodology is decided to investigate thoroughly the possibility of optimization of tube surface.

- Modification of the tube's shapes for maximum heat transfer enhancement.
- Evaluating performance of such different types of tubes.
- Comparison with the plane and other shapes of tubes.
- Study of flow patterns for dimple shapes tube.

Proposal of optimum shape of tube for the given range of base heat flux, based on numerical results and experimental validation of the same.

Fig 3.1 Types of tubes

These all are the components of the test set up. The line diagram of the test set up is shown in fig. 3.2.

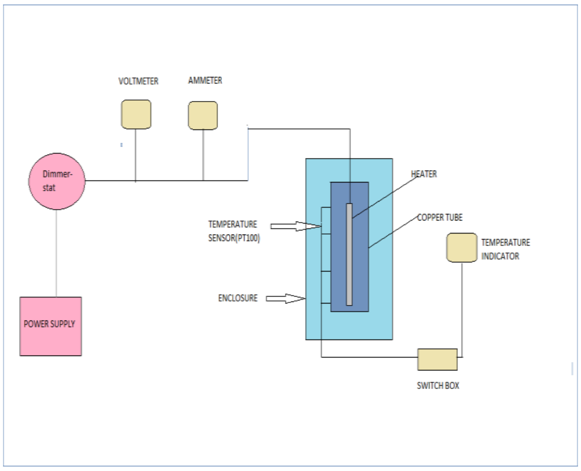


Fig 3.2 Schematic of experimental set up

4. CFD SIMULATION

4.1 Introduction

Computational fluid dynamics (CFD) is a computer-based simulation method for analysing fluid flow, heat transfer, and related phenomena such as chemical reactions. This dissertation uses CFD for analysis of flow and heat transfer. It will be advantageous to use CFD over traditional experimental-based analyses, since experiments have a cost directly proportional to the number of configurations desired for testing, unlike with CFD, where large amounts of results can be produced at practically no added expense. In this way, parametric studies to optimize equipment are very inexpensive with CFD when compared to experiments.

4.2 CFD Analysis Process

To perform a CFD analysis, the analyst will state the problem and use scientific Knowledge to express it mathematically. Then the CFD software package will embody this knowledge and expresses the stated problem in scientific terms. Finally, the computer will perform the calculations dictated by CFD software and the analyst will inspect and interpret their results. In principle, three different major tasks should be done to perform a CFD simulation.

4.3 Problem Identification

The final goal is to develop CFD-methods for realistic prediction of the overall heat transfer coefficient in a passage containing in-line array of pin fins, to improve the heat transfer efficiency. Therefore, generic flow cases with three different fin geometries that have the typical flow-heat transfer characteristics are investigated. The focus of this work is on the investigation of the effects of fin morphology in prediction of the flow and heat transfer in a typical heat exchanger passage. Also, the study of the underlying physics of the flow-heat transfer processes in these cases is included.

4.4 Model Development in ANSYS

The ANSYS Design Modeler is a gateway to geometry coping with for an ANSYS analysis or we can import the model file of other software like CAD, SOLIDWORKS. The geometry consists the physics or physical structure. In this we have developed our model in SOLIDWORKS 2017 and then imported it to ANSYS software. the reason for selecting SOLIDWORKS is that we are familiar with it and secondly for ease in making small protrusions on the tube which is very tough to draw in ANSYS.

Create a Geometry - All engineering simulations start with geometry to represent the design, be it a solid component of a structural analysis or the air volume of a fluid or electromagnetic field. The engineer either has geometry that has been created in a SOLIDWORKS system or builds the geometry from scratch. The ANSYS Design Modeler is a gateway to geometry handling for an ANSYS analysis. Geometry created using ANSYS Design Modeler software which is specifically designed for the creation and preparation of geometry for simulation. In engineering simulations, the geometry includes details not needed for simulation. Only the physics involved is to be included, simulating such a fully detailed model.

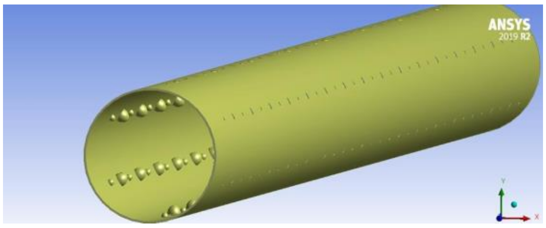


Fig. 4.1 Dimple Tube

4.5 Selection of Mesh and Reason

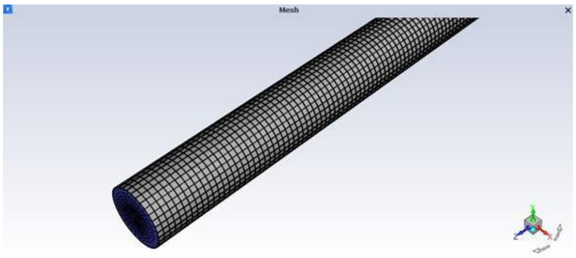


Fig. 4.2 Meshing of smooth Tube

Tetrahedron- A tetrahedron has 4 vertices, 6 edges, and is bounded by 4 triangular faces. In most cases a tetrahedral volume mesh can be generated automatically. When geometries are complex or the range of length scales of the flow is large, a triangular/tetrahedral mesh can be created with far fewer cells than the equivalent mesh consisting of quadrilateral/hexahedral elements. This is because a triangular/tetrahedral mesh allows clustering of cells in selected regions of the flow domain. Structured quadrilateral/hexahedral meshes will generally force cells to be placed in regions where they are not needed. Unstructured quadrilateral/hexahedral meshes offer many of the advantages of triangular/tetrahedral meshes for moderately-complex geometries.

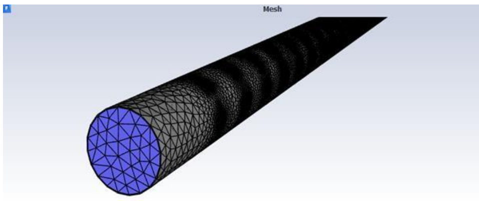


Fig. 4.3 Meshing of dimple tube

Choosing a Turbulence Model –

The eight RANS turbulence models differ in how they model the flow close to walls, the number of additional variables solved for, and what these variables represent. All of these models augment the Navier-Stokes equations with an additional turbulence eddy viscosity term, but they differ in how it is computed.

k-ε

The k-ε model solves for two variables: k, the turbulence kinetic energy; and ε (epsilon), the rate of dissipation of turbulence kinetic energy. Wall functions are used in this model, so the flow in the buffer region is not simulated. The k-ε model has historically been very popular for industrial applications due to its good convergence rate and relatively low memory requirements. It does not very accurately compute flow fields that exhibit adverse pressure gradients, strong curvature to the flow, or jet flow. It does perform well for external flow problems around complex geometries. For example, the k-ε model can be used to solve for the airflow around a bluff body. The turbulence models listed below are all more nonlinear than the k-ε model and they can often be difficult to converge unless a good initial guess is provided. The k-ε model can be used to provide a good initial guess. Just solve the model using the k-ε model and then use the new Generate New Turbulence Interface functionality, available in the CFD Module with COMSOL Multiphysics version 5.3.

k-ω

The k-ω model is like the k-ε model, but it solves for ω (omega) — the specific rate of dissipation of kinetic energy. It is a low Reynolds number model, but it can also be used in conjunction with wall functions. It is more nonlinear, and thereby more difficult to converge than the k-ε model, and it is quite sensitive to the initial guess of the solution. The k-ω model is useful in many cases where the k-ε model is not accurate, such as internal flows, flows that exhibit strong curvature, separated flows, and jets. A good example of internal flow is flow through a pipe bend.

Low Reynolds Number k-ε

The low Reynolds number k-ε model is like the k-ε model, but does not need wall functions: it can solve for the flow everywhere. It is a logical extension of the k-ε model and shares many of its advantages, but generally requires a denser mesh; not only at walls, but everywhere its low Reynolds number properties kick in and dampen the turbulence. It can sometimes be useful to use the k-ε model to first compute a good initial condition for solving the low Reynolds number k-ε model. An alternative way is to use the automatic wall treatment and start with a coarse boundary layer mesh to get wall functions and then refine the boundary layer at the interesting walls to get the low Reynolds number models. The low Reynolds number k-ε model can compute lift and drag forces and heat fluxes can be modelled with higher accuracy compared to the k-ε model. It has **SST**

The SST model is a combination of the k-ε model in the free stream and the k-ω model near the walls. It is a low Reynolds number of model and kind of the “go to” model for industrial applications. It has similar resolution requirements to the k-ω model and the low Reynolds number k-ε model, but its formulation eliminates some weaknesses displayed by pure k-ω and k-ε models. In a tutorial model example, the SST model solves for flow over a NACA 0012 Air foil. The results are shown to compare well with experimental data.

5. RESULTS AND DISCUSSION

CFD Results

Smooth Tube at Re 2300: -

1) As we can observe from the table that if we increase the Reynolds no. and keep inlet temperature the same then the friction coefficient increases on the outlet side of a smooth tube.

2) Due to increase in friction coefficient the value of heat transfer coefficient (h) continuously decreases as a result the temperature on the outer side of the smooth tube increases.

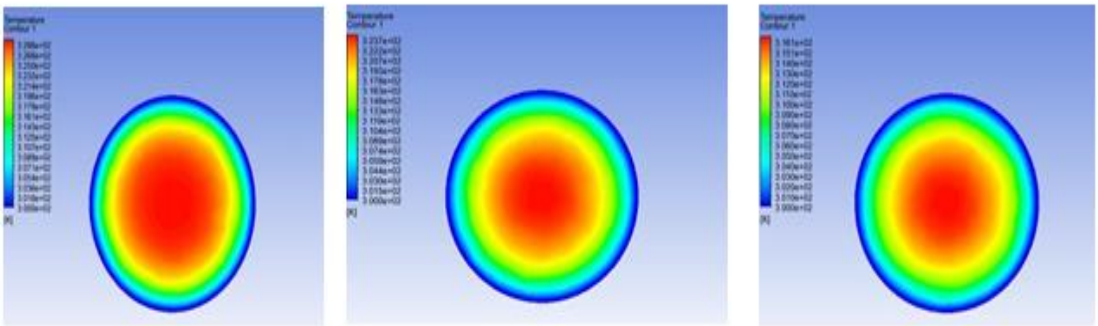
At Re 2300

Dimple tube Case 1

Temperature contour:

1) To observe temperature contour for a tube length of 380 mm we have considered 3 cross - sections at different lengths (i.e. 100 mm, 200mm and 380 mm).

2) While looking at temperature contour, we have observed that as we move from inlet to outlet the red circle diameter decreases which means that heat transfer is increased and maximum heat transfer is taking place at x = 600 mm

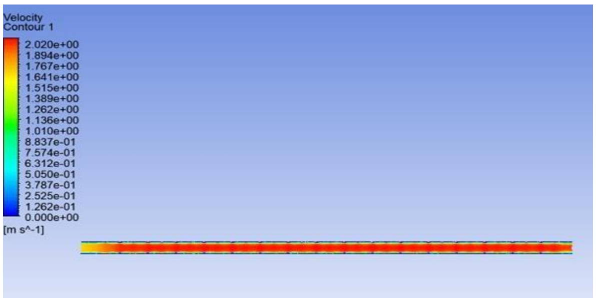


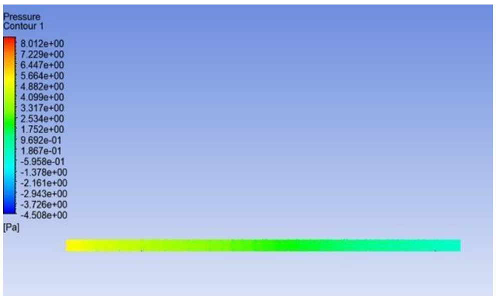
a) At x = 100 b) x = 200 c) x = 380

Fig 5.1 Temperature contour of dimple tube at different length

Velocity & pressure contour and the vector plot:

1. As we know at inlet boundary conditions the velocity of fluid is zero.
2. As the fluid moves inside the tube its velocity increases and its maximum at the outlet.





a) velocity b) pressure

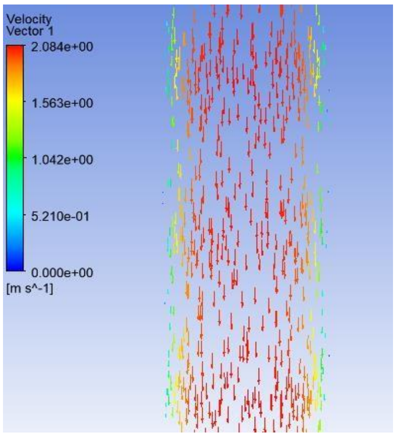


Fig 5.2 Velocity contour and vector diagram of dimple tube

Dimple tube Case 2

Temperature contour:

a) x = 100 b) x = 200 c) x = 380

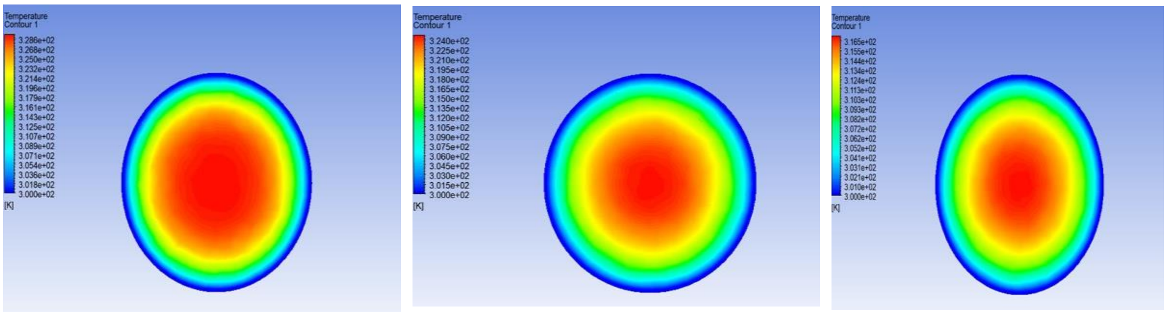
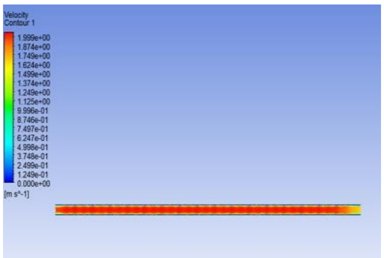


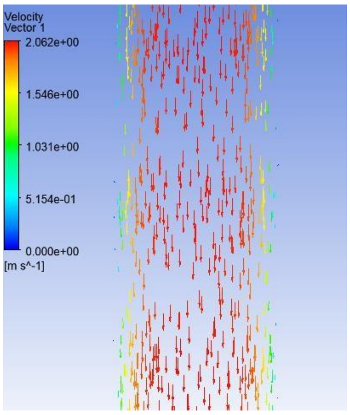
Fig 5.3 Temperature contour of dimple tube at different length

Velocity & pressure contour and the vector plot:



a) velocity

b) pressure



c) vector plot

Fig 5.4 Velocity contour and vector diagram of dimple tube

Dimple tube Case 3

Temperature contour:

a) x = 100 b) x = 200 c) x = 380

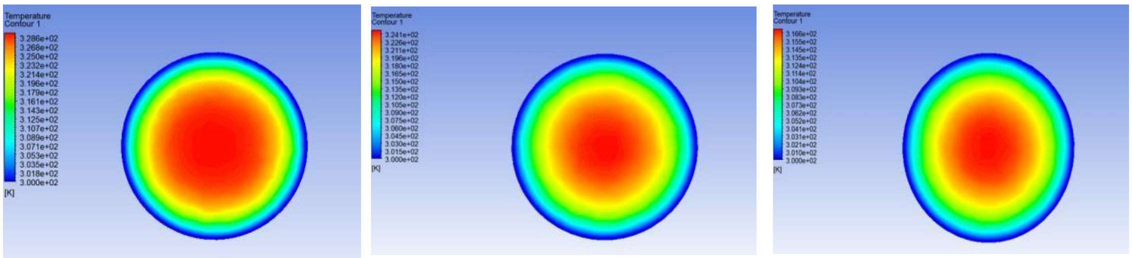


Fig 5.5 Temperature contour of dimple tube at different length

Velocity & pressure contour and the vector plot:-

a) velocity b) pressure

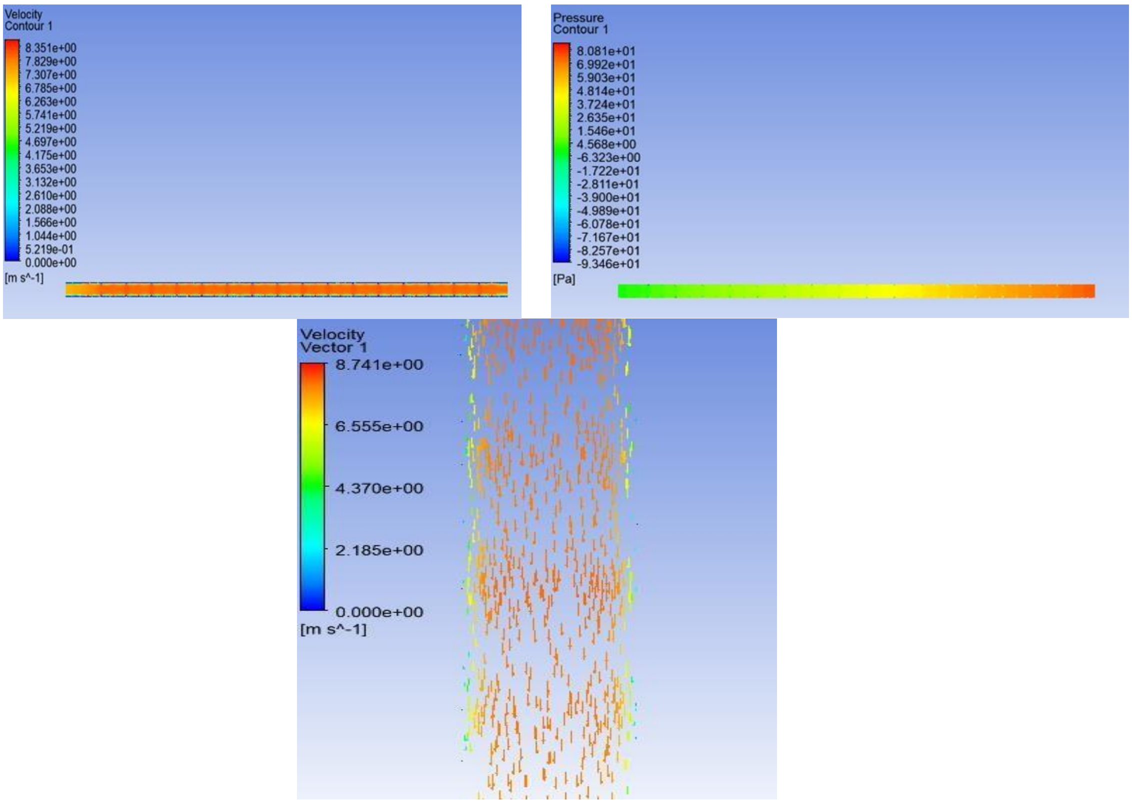


Fig 5.6 Velocity contour and vector diagram of dimple tube

To provide further insights into the evolution of the flow pattern, a succession of vector diagram was presented. The results of the vector diagram are shown in above figures. As we have developed protrusions on surface, we can clearly observe increase in amount of heat transfer. As flow increases there is enhancement in heat transfer due to presence of larger depth dimples. As, we can clearly observe that at the ends the red arrows quantity gets decreased and orange arrow quantity increased. With the continuous increase in Reynolds number recirculation increases more and more. At the highest Reynolds number, the inertial losses are dominant.

6. CONCLUSION

The Heat transfer enhancement of dimple tube, corrugated tube and plain tube is done in this done. Here, voltage is varying from 60 V to 100 V. It is seen that as the voltage increases the heat transfer is also increases. From the experiment outcomes we can conclude that the heat transfer is more from the corrugated tube as compared to others tube, this is because of the material property and small reduction in area as compared to dimple tube. It is seen that there is wide increase in heat transfer from corrugated tube after 80 Volts. Hence, we have high heat transfer in the corrugated tube at higher voltage.

From, the CFD result we can conclude that, Heat transfer coefficient increased with Reynolds number. Heat transfer enhancement was found to be more than 150% as compared to an equivalent smooth tube. Heat transfer enhancement was higher than pressure drop enhancement at any given operating condition, therefore resulting in higher PEC.The best performance (PEC = 1.5) was obtained at Reynolds number 2300.

REFERENCES

1. Tribus, M. and J. Klein., “Heat Transfer Symposium”, University of Michigan Press, pp. 211, 1953.
2. Cess R. D., “Applied scientific research”, vol.10 (1), pp 430-438, 1961.
3. Szewczyk, A. A., “J. Heat Transfer”, 86(4), 501-507, 1964.
4. Whitaker, S., “AIChE Journal”, vol.18 (2), 1972.
5. Shah, R. K., and London, A. L., “Academic Press”, San Diego, CA. 1978.
6. Cooper, P.I. Sheridan, J.C. and Flood, G.J., “Int. J. of Heat and Fluid Flow”, Vol.7 (1), pp61-68, 1986.
7. Copeland, D., “Synopsis report”, Sumitomo precision products, Dec-1998.
8. Kim, S. J., and Kim, D., Journal of heat transfer, vol.121, pp.639-645, 1999.
9. Cheng, W., T, and Lin. H.T., “International Journal of Heat and Mass Transfer”, 01,2002.
10. Sartori, E., Solar Energy, Vol.80 (9), pp-1063–1071, 2006.
11. Abreu, C.R.A., Alfradique, M.F., and Telles, A. S., Chemical Engineering Science, Vol. 61(13), pp. 4282–4289, 2006.
12. Yao, L.S., and Molla, M., “International Journal of Heat and Mass Transfer”, Vol.51 (21– 22), pp. 5154–5159, 2008.
13. Seyyedi, S.M., Bararnia, H., Ganji, D.D., Gorji-Bandpy, M. and Soleimani, S., “Int. J Thermal Sciences”, Vol. 61, pp 1–14, 2012.
14. Pantokratoras, A., “Journal of Mechanical Science and Technology”, 28 (5), pp.1909-1915, 2014.
15. L.A. Asmantas, M.A. Nemira, V.V. Trilikauskas, Coefficients of heat transfer and hydraulic drag of a twisted oval tube, Heat Tran Sov Res 17 (1985) 103–109.
16. L. Yang, Z.X. Li, Numerical analysis of laminar flow and heat transfer in twisted elliptic tubes, Eng Mech 20 (2003) 143–148.
17. Patil, P. and Deshmukh, P., 2014. Numerical study of flow and heat transfer in circular tube with almond shape dimple. International Journal of Engineering and Research, 3(8), pp.21-29.
18. Huang, Z., Yu, G.L., Li, Z.Y. and Tao, W.Q., 2015. Numerical study on heat transfer enhancement in a receiver tube of parabolic trough solar collector with dimples, protrusions and helical fins. Energy Procedia, 69, pp.1306-1316.
19. Wang, Y., He, Y.L., Li, R. and Lei, Y.G., 2009. Heat transfer and friction characteristics for turbulent flow of dimpled tubes. Chemical Engineering & Technology: Industrial Chemistry- Plant Equipment- Process Engineering- Biotechnology, 32(6), pp.956-963.
20. Rao, Y., Li, B. and Feng, Y., 2015. Heat transfer of turbulent flow over surfaces with spherical dimples and teardrop dimples. Experimental Thermal and Fluid Science, 61, pp.201-209.
21. Cheraghi, M. H., Ameri, M., & Shahabadi, M. (2020). Numerical study on the heat transfer enhancement and pressure drop inside deep dimpled tubes. International Journal of Heat and Mass Transfer, 147, 118845.
22. Albanesi, A.W., Daish, K.D., Dally, B. and Chin, R.C., 2018, December. Investigation of heat transfer enhancement in dimpled pipe flows. In 21st Australasian Fluid Mechanics Conference, Adelaide, Australia.
23. Wang, Y., He, Y.L., Lei, Y.G. and Zhang, J., 2010. Heat transfer and hydrodynamics analysis of a novel dimpled tube. Experimental thermal and fluid science, 34(8), pp.1273-1281.
24. Yadav, R.K. and Jain, V.N., 2016. Thermal Analysis of Heat Exchanger with the Help of Taguchi Method. International Journal of Advanced Research in Engineering and Technology, 7(1), pp.01-06.
25. Li, P., Xie, Y. and Zhang, D., 2016. Laminar flow and forced convective heat transfer of shear-thinning power-law fluids in dimpled and protruded microchannels. International Journal of Heat and Mass Transfer, 99, pp.372-382.
26. Ji, W.T., Fan, J.F., Zhao, C.Y. and Tao, W.Q., 2019. A revised performance evaluation method for energy saving effectiveness of heat transfer enhancement techniques. International Journal of Heat and Mass Transfer, 138, pp.1142-1153.

27. Kim, K.Y. and Choi, J.Y., 2005. Shape optimization of a dimpled channel to enhance turbulent heat transfer. Numerical Heat Transfer, Part A: Applications, 48(9), pp.901-915.
28. Mahmood, G.I. and Ligrani, P.M., 2002. Heat transfer in a dimpled channel: combined influences of aspect ratio, temperature ratio, Reynolds number, and flow structure. International Journal of Heat and mass transfer, 45(10), pp.2011-2020.
29. Liang, Z., Xie, S., Zhang, J., Zhang, L., Wang, Y. and Ding, H., 2019. Numerical investigation on plastic forming for heat transfer tube consisting of both dimples and protrusions. The International Journal of Advanced Manufacturing Technology, 102(1-4), pp.775-790.
30. Maithani, R. and Kumar, A., 2019. Correlations development for Nusselt number and friction factor in a dimpled surface heat exchanger tube. Experimental Heat Transfer, pp.1-22.