INTERNATIONAL JOURNAL OF ADVANCE RESEARCH, IDEAS AND INNOVATIONS IN TECHNOLOGY

ISSN: 2454-132X
Impact factor: 4.295
(Volume 4, Issue 4)
Available online at: www.ijariit.com

CFD analysis of heat transfer enhancement in a pipe with or without circular insert

Krit Kumar
kritkumarjanghel2011@gmail.com
Patel College of Science & Technology, Bhopal, Madhya Pradesh

Amit Kumar
amitavit689@gmail.com
Patel College of Science & Technology, Bhopal, Madhya Pradesh

ABSTRACT

This work presents a CFD analysis of heat transfer problem. The enhancement of heat transfer with mass in a tube has been investigated without and with insert. The combinations of without inserts, two inserts, four inserts, six inserts, eight inserts and ten inserts has been used. An 800 mm long pipe with 26 mm inner diameter and 30 mm outer diameter is considered for our simulation. 5. It is observed that outlet temperature increases to 313.15 K when two inserts are being fitted into tube. We get the lowest outlet temperature for ten inserts. Therefore after 8 inserts outlet temperature decreases.

***Keywords**: Inserts, Turbulent flow, Heat transfer coefficient, Reynolds number, exit temperature*

1. INTRODUCTION

High performance heat transfer system is of great importance in many industrial applications. The performance of conventional heat exchangers can be substantially improved by a number of heat transfer enhancement techniques. A variety of complex, highly viscous liquids are involved that undergoes heat exchange process while flowing through heat exchangers. Because of their viscous nature, they tend to have low flow rates and generally represent the dominant thermal resistance due to their viscous nature, adversely affecting size and cost of heat exchanger. The process industry is continuously working to incorporate enhancement in heat transfer. Enhancement techniques can be classified as active methods which require external power and passive methods, which require no direct application of external power. The enhanced surfaces are routinely used to improve thermal and hydraulic performance of heat exchangers. These heat exchangers are used in process industries, air conditioning, refrigeration, power generation etc. Secondary flows are created by these techniques, causing the separation of the boundary layer results in better bulk fluid mixing which reduces the temperature gradients in the fluid flow. It is expected that a heat transfer augmentation device should produce significant increase in heat transfer.

2. PROBLEM FORMULATION

This work presents a FEM based heat transfer problem. The heat transfer enhancement technology has been developed and applied to heat exchanger applications such as refrigeration, process industry etc. There have been many techniques proposed over the years for enhancing the heat transfer with mass in tubes by using different types of insert. Hence the present work is concerned with carrying out three-dimensional simulations on a circular copper tube with inserts, through which water flows.

3. COMPUTATIONAL DOMAIN

In our work we designed a horizontal circular pipe without insert, two inserts, four inserts, six inserts, eight inserts and ten inserts with different orientation and different shapes. The tube has total length of 800 mm, inner diameter and outer diameter of 26 mm and 30 mm [Sabbir Hossain et al. (2015)].

The ultimate goal of this study is to improve understanding of the enhancement of heat transfer in a circular pipe using insert and without insert. Our model has been designed as the geometry creates eddies across the flow which enhances the heat transfer. The purpose of using inserts in our design is to decrease layer to layer friction and to scatter the fluid particles carrying heat from the boundary wall. Different design of circular pipe with inserts as shown below.

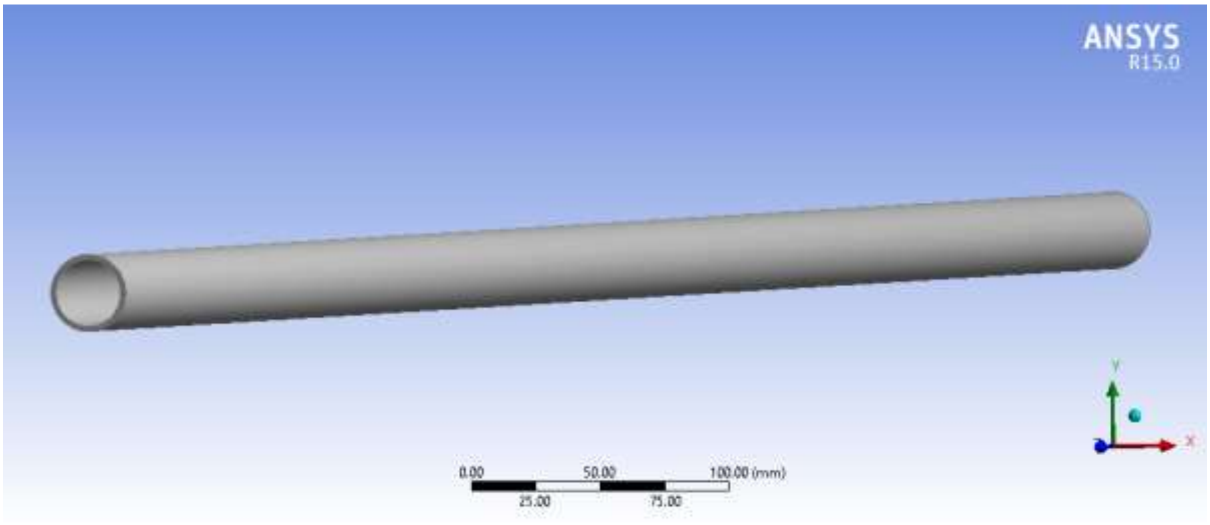


Fig. 1: Domain with no insert

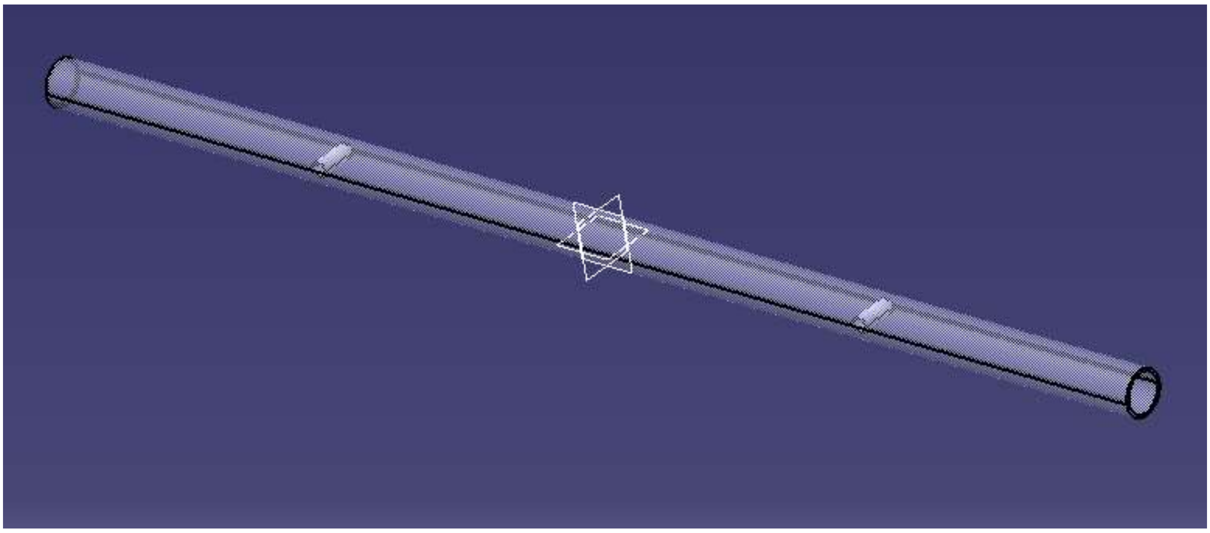


Fig. 2: Domain with 2 inserts

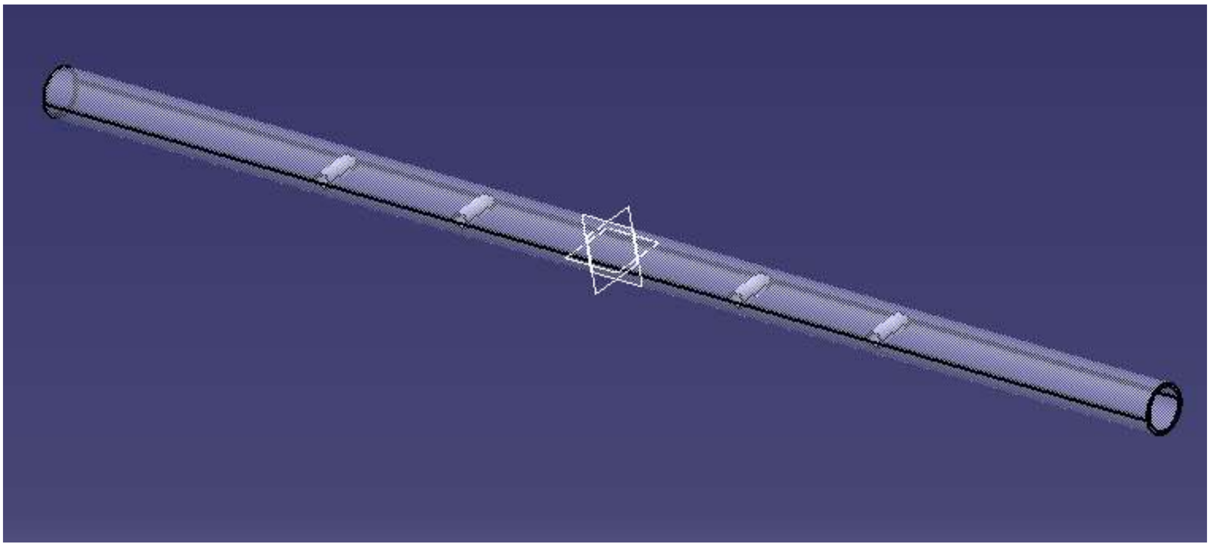


Fig. 3: Domain with 4 inserts

Table 1: Operating and geometrical parameters used for CFD analysis

Operating and Geometrical parameters	Value / Range
Length of pipe	800 mm
Outer diameter	30 mm
Inner diameter	26 mm
No. of insert	2, 4, 6, 8, 10
Shape of insert	circular
Constant heat flux, q	32087 w/m^2
Range of Reynolds number	2000-10000

Reynolds number was varied from 2000-10000. Constant heat flux of value approximately 32087 W/m2 was supplied only on the pipe. Simulations were performed assuming the flow to be steady. The operating and geometrical parameters used for computational analysis are listed in Table 1.

4. BOUNDARY CONDITIONS

On the pipe wall, no-slip boundary conditions were assigned. Constant heat flux of 32087 W/m^2 was decided to be the boundary condition at the wall of the pipe. At the inlet, different velocity with an inlet temperature of 293.15 K and at the exit, invariable pressure (atmospheric pressure) boundary conditions were assigned as shown in Figure 4.

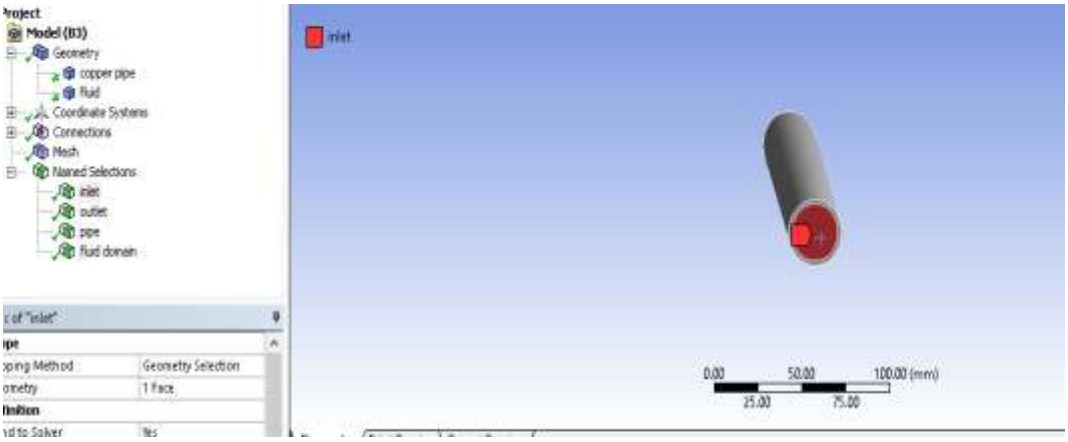


Fig. 4: Boundary conditions

5. MESHING OF THE DOMAIN

The meshing work was accomplished on commercially available ANSYS meshing software. The geometry created was imported in ANSYS meshing. The required number of divisions and the type of "bias" were assigned to each edge. In order to obtain regular shaped mesh cells with the best orthogonal quality, mapped facing option was activated. Finally, mesh was generated by clicking on "Generate Mesh" button. Figure 5 shows the meshed domain for different cases.

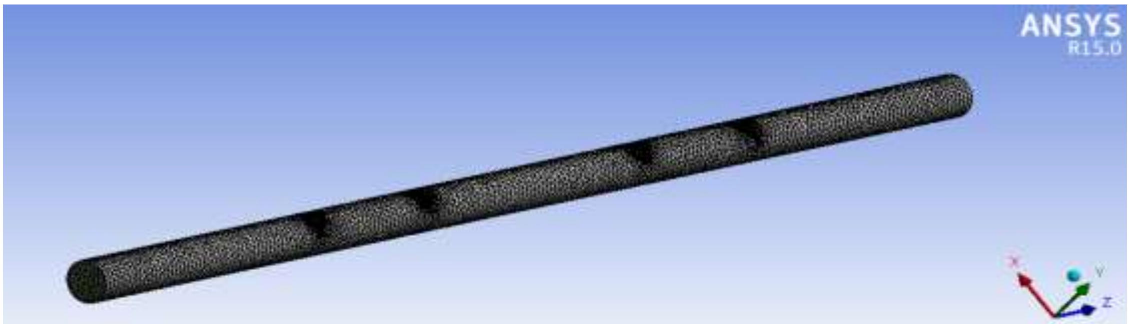


Fig. 5: Mesh model

6. RESULTS WITH NO INSERT

6.1 Effect of Reynolds number on convective heat transfer coefficient

Figure 6 shows the variation of convective heat transfer coefficient with Reynolds number. As the Reynolds number increases, the heat transfer coefficient also increases. The variation in heat transfer coefficient is low at small Reynolds numbers while it is large at higher Reynolds numbers. This behaviour seems due to increased turbulence at higher Reynolds numbers and also due to breakage of thermal boundary layer at higher Reynolds numbers.

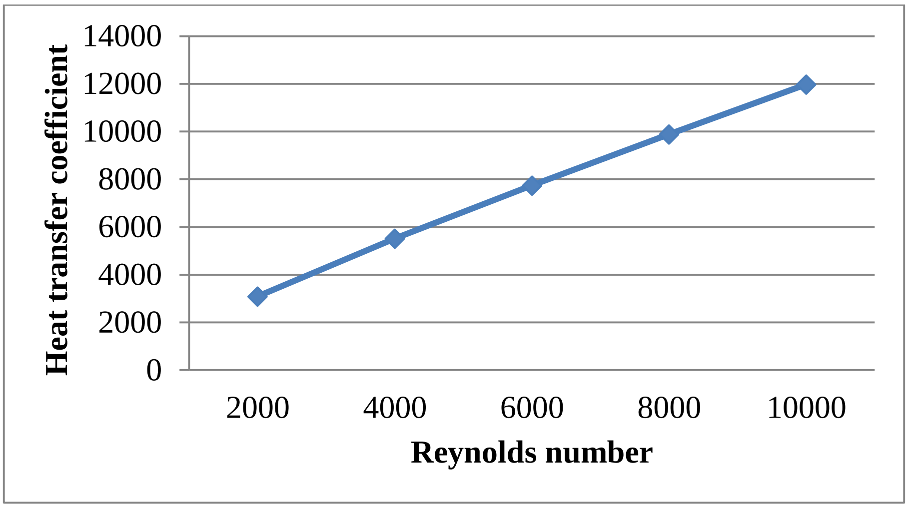
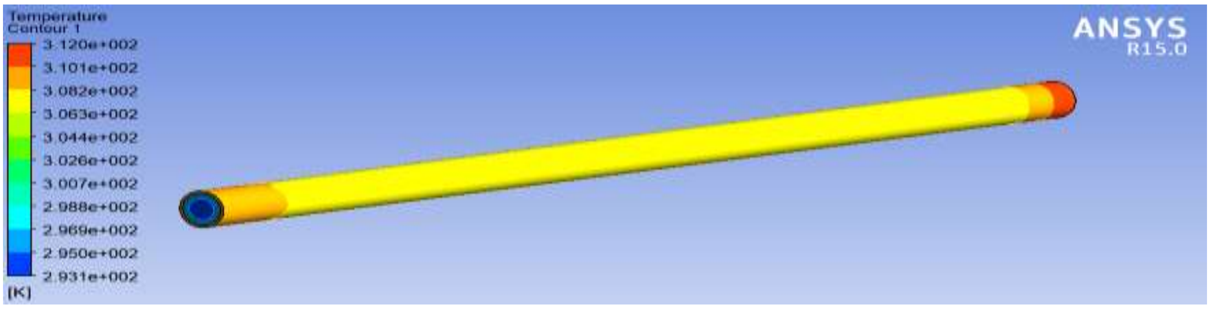


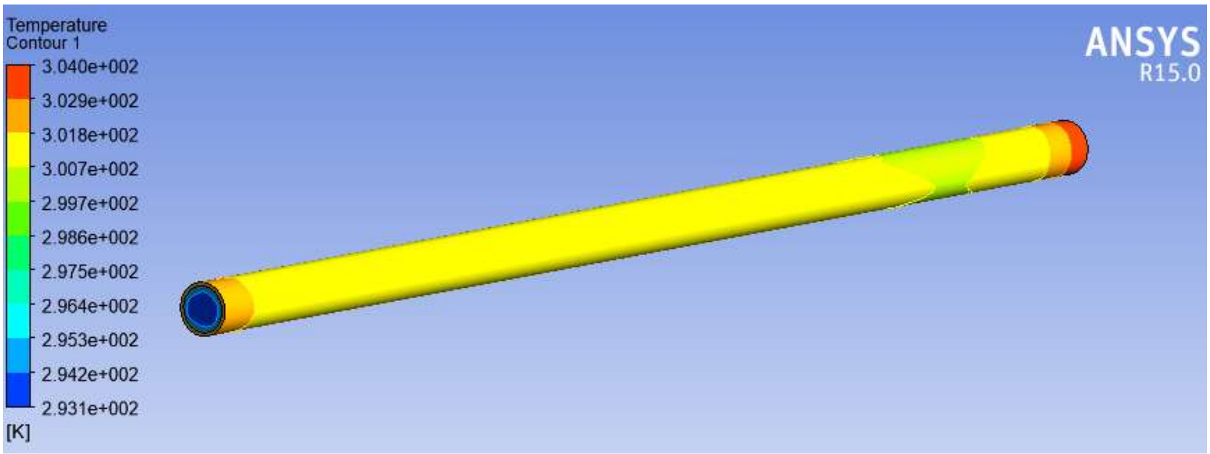
Fig. 6: Variation of heat transfer coefficient for different values of Reynolds number

6.2 Effect of Reynolds number on outlet temperature

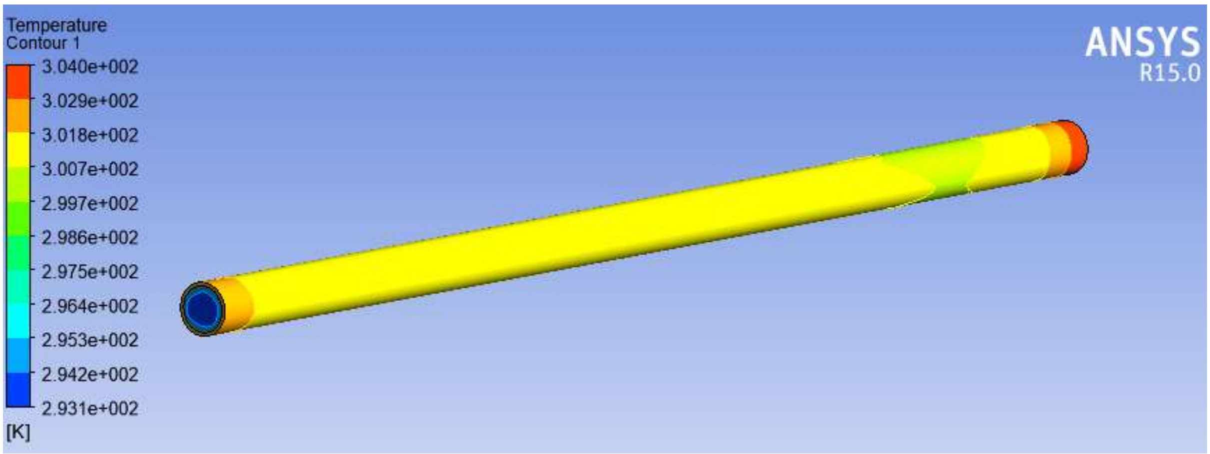
Most of the flow which occurs in practical applications is in general turbulent in nature. In the turbulent region, the velocity of the particles very near to surface becomes almost zero. In this region the particle have very low kinetic energy. This region is called laminar sub-layer. These laminar sub-layer acts as a barrier of heat transfer from heated surface to fluid medium.



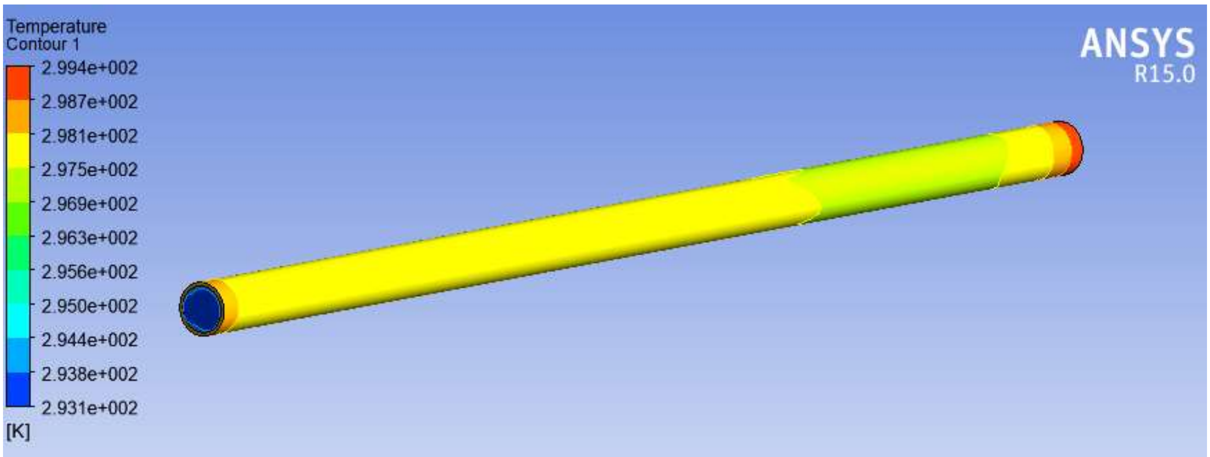
(a) Re= 2000



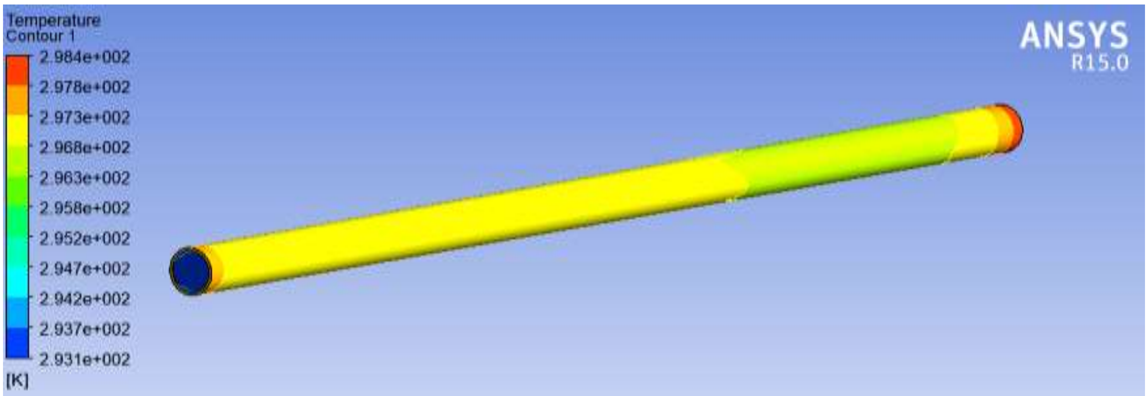
(b) Re= 4000



(c) Re = 6000



(d) Re = 8000



(e) Re = 10000

Fig. 7: Temperature distribution of water domain

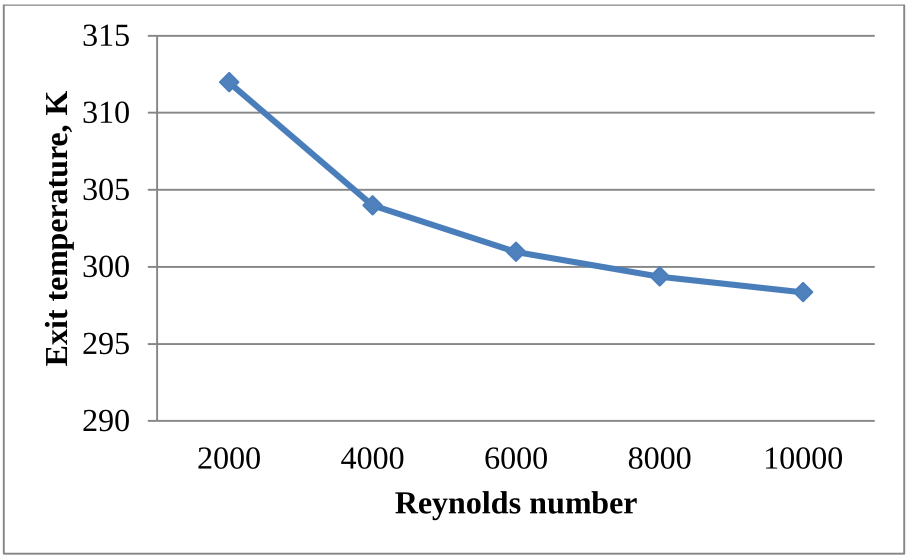


Fig. 8: Variation of exit temperature for different values of Reynolds number with no insert

Figure 8 shows the variation of temperature of air at exit with Reynolds number. As the Reynolds number increases, the heat carrying capacity of air increases. Because of this heat utilization also increases, but the rate of increase of heat capacity is more significant than the rate of increase of heat utilization. Therefore the temperature of air at exit decreases as a result increase in Reynolds number.

7. RESULTS ON EFFECT OF INSERT

7.1 Effect of Reynolds number on convective heat transfer coefficient

Figure 9 shows the variation of convective heat transfer coefficient with Reynolds number. As the Reynolds number increases, the heat transfer coefficient also increases for different number of insert. The variation in heat transfer coefficient is low at small Reynolds numbers while it is large at higher Reynolds numbers. This behaviour seems due to increased turbulence at higher Reynolds numbers and also due to breakage of thermal boundary layer at higher Reynolds numbers.

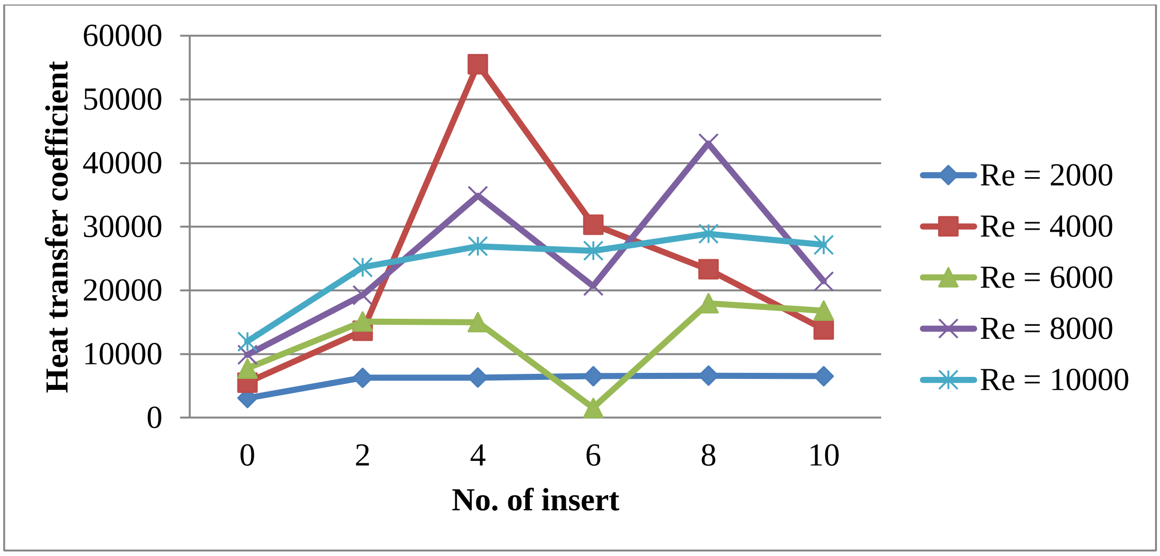


Fig. 9: Variation of heat transfer coefficient for different values of Reynolds number with inserts

7.2 Effect of Reynolds number on outlet temperature

Most of the flow which occurs in practical applications is in general turbulent in nature. In the turbulent region, the velocity of the particles very near to surface becomes almost zero. In this region the particle have very low kinetic energy. This region is called laminar sub-layer. These laminar sub-layer acts as a barrier of heat transfer from heated surface to fluid medium.

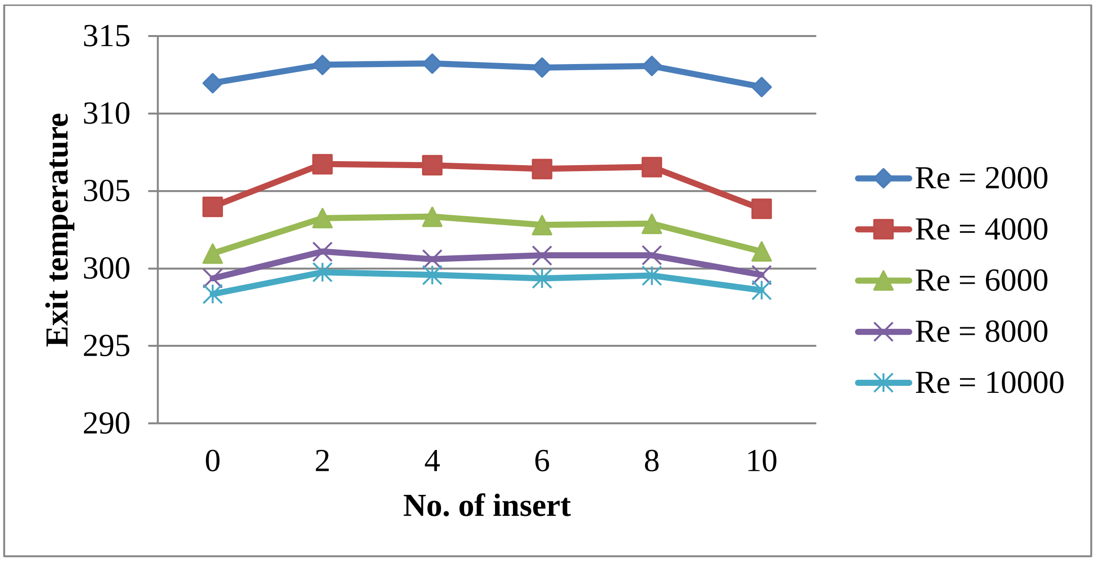


Fig. 10: Variation of exit temperature for different values of Reynolds number with inserts

Figure 10 shows the variation of temperature of air at exit with Reynolds number. As the Reynolds number increases, the heat carrying capacity of air increases. Because of this heat utilization also increases, but the rate of increase of heat capacity is more significant than the rate of increase of heat utilization. Therefore the temperature of air at exit decreases as a result increase in Reynolds number.

It is observed that outlet temperature increases to 313.15 K when two inserts are being fitted into tube. We get the lowest outlet temperature for ten inserts. Therefore after 8 inserts outlet temperature decreases.

8. EFFECT OF ORIENTATION OF INSERT

In this case, we fit inserts in pipe at 90º and 0º alternatively. The distance among these inserts are also varied. Then we run our simulation for these domains.

8.1 Effect of Reynolds number on convective heat transfer coefficient

Figure 11 shows the variation of convective heat transfer coefficient with Reynolds number. As the Reynolds number increases, the heat transfer coefficient also increases for different number of insert. The variation in heat transfer coefficient is low at small Reynolds numbers while it is large at higher Reynolds numbers. This behaviour seems due to increased turbulence at higher Reynolds numbers and also due to breakage of thermal boundary layer at higher Reynolds numbers.

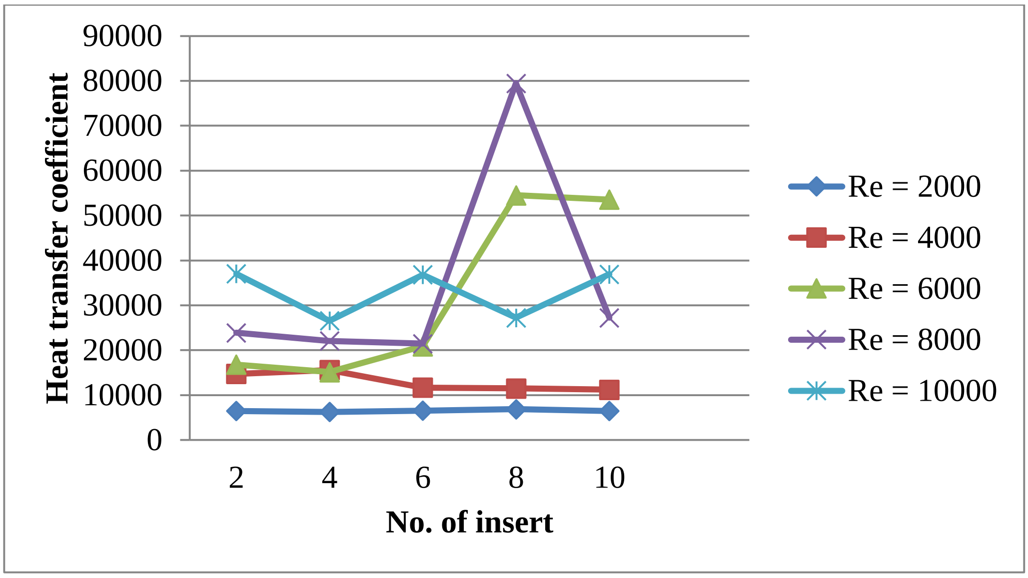


Fig. 11: Variation of heat transfer coefficient for different orientation with inserts

8.2 Effect of Reynolds number on outlet temperature

Most of the flow which occurs in practical applications is in general turbulent in nature. In the turbulent region, the velocity of the particles very near to surface becomes almost zero. In this region the particle have very low kinetic energy. This region is called laminar sub-layer. These laminar sub-layer acts as a barrier of heat transfer from heated surface to fluid medium

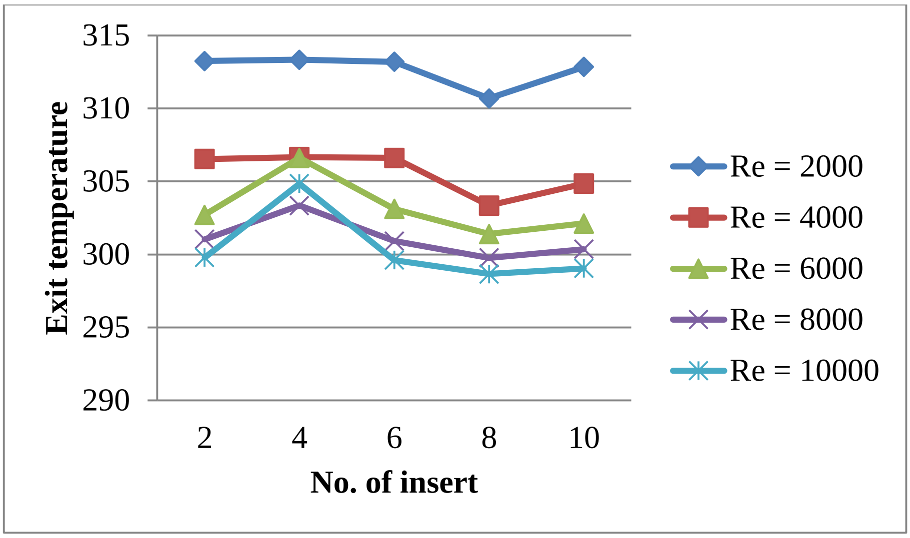


Fig. 12: Variation of exit temperature for different orientations with inserts

Figure 12 shows the variation of temperature of air at exit with Reynolds number. As the Reynolds number increases, the heat carrying capacity of air increases. Because of this heat utilization also increases, but the rate of increase of heat capacity is more significant than the rate of increase of heat utilization. Therefore the temperature of air at exit decreases as a result increase in Reynolds number.

It is observed that outlet temperature increases to 313.25 K when two inserts are being fitted into tube. We get the highest outlet temperature for four inserts. Therefore we noticed that there is an irregular variation of outlet temperature and cannot come to certain decision to find out the reasons of such irregularities.

9. CONCLUSIONS

A CFD simulation study on heat transfer characteristics of fluid in a circular tube without and with inserts under constant boundary heat flux for turbulent flow has been presented. The conclusions from the simulation are as follows:

1. We found that heat transfer rate is obtained after using inserts compare to without inserts.
2. As the Reynolds number increases, the heat transfer coefficient also increases. The variation in heat transfer coefficient is low at small Reynolds numbers while it is large at higher Reynolds numbers. This behaviour seems due to increased turbulence at higher Reynolds numbers and also due to breakage of thermal boundary layer at higher Reynolds numbers.
3. As the Reynolds number increases, the heat carrying capacity of air increases. Because of this heat utilization also increases, but the rate of increase of heat capacity is more significant than the rate of increase of heat utilization. Therefore the temperature of air at exit decreases as a result increase in Reynolds number.
4. The heat transfer increases with increase in Reynolds number due to increase in axial convection. Increase in Reynolds number increases the heat transfer due to disturbance in boundary layer causing increased convection heat transfer from wall to fluid.
5. It is observed that outlet temperature increases to 313.15 K when two inserts are being fitted into tube. We get the lowest outlet temperature for ten inserts. Therefore after 8 inserts outlet temperature decreases.
6. It is observed that outlet temperature increases to 313.25 K when two inserts are being fitted into tube. We get the highest outlet temperature for four inserts. Therefore we noticed that there is an irregular variation of outlet temperature and cannot come to certain decision to find out the reasons of such irregularities.
7. After changing inserts position, greater outlet temperature is obtained.
8. Optimum distance among the inserts causes greater efficiency of the heat transfer rate.
9. The optimum distance among the inserts should be considered for executing the design which makes it economic and easy to manufacture.

10. REFERENCES

[1] Wei Yuan (2018), Heat Transfer and Friction Characteristics of Turbulent Flow through a Circular Tube with Ball Turbulators, Applied Sciences, 8(6), 1-13.
[2] Virendra Patidar (2018), Enhancement of Heat Transfer Through Pipe with the Help of Various Types of Turbulators – A Review, 5(1), 698-704.
[3] Sachin S. Surywanshi (2017), Numerical Heat Transfer Analysis of Flowing Fluid through a Circular Tube with Helical Twisted Strip Inserts, Journal of Advances in Science and Technology, 13(1), 88-94.
[4] Durgesh V. Ahire (2017), Experimental Investigation of Heat Transfer Enhancement in Circular Pipe using Conical Ring Inserts, International Journal of Mechanical And Production Engineering, 5(6), 83-88.
[5] V. Tirupati Rao (2017), Augmentation Heat Transfer in a Circular Tube Using Conical Ring And Twisted Tape Insert, International Journal of Scientific Research Engineering & Technology, 6(1), 15-22.
[6] M. Ashham (2017), Simulation of Heat Transfer in a Heat Exchanger Tube with Inclined Vortex Rings Inserts, International Journal of Applied Engineering, 12(20), 9605-9613.
[7] Mukesh P. Mangtani (2016), Performance Analysis of Copper Twisted Tape Inserts for Heat Transfer in a Circular Tube, Int. Journal of Engineering Research and Application, 6 (11), 25-28.

[8] Yogesh B. Rakate (2016), Experimental and Numerical Analysis of Heat Transfer Augmentation through a Pipe using Twisted Tapes, International Journal for Innovative Research in Science & Technology, 2(10), 65-71.
[9] Avinash Savekar (2015), Analysis of Heat Transfer in Pipe with Twisted Tape Inserts, International Conference on Fluid Flow, Heat and Mass Transfer, Ottawa, Ontario, Canada, April 30 – May 1, 143-1.
[10] Dipak Sen, Rajdeep Ghosh (2015), A Computational Study of Very High Turbulent Flow and Heat Transfer Characteristics in Circular Duct with Hemispherical Inline Baffles, International Journal of Mechanical and Mechatronics Engineering, 9(6), 1046-1051.
[11] K. Gowrisankar (2015), Augmentation Heat Transfer in a Circular Tube Using Divergent Nozzle as Insert, International Journal for Research in Applied Science & Engineering, 3(9), 143-149.
[12] Vinay Kumar Patel (2015), Heat Transfer Enhancement using Passive Enhancement Technique, International Journal of Engineering Sciences & Research Technology, 4(9), 394-399.
[13] Shivaji V. Mundhe (2014), Experimental Analysis of Heat Transfer for Turbulent Flow through Circular Pipe, International Journal of Innovative Research in Advanced Engineering, 1(3), 70-75.
[14] Rupesh G. Telrandhe, T.S. Karhale, B.B. Deshmukh (2013), Thermal Analysis of Circular Inclined Pipe Subjected To Natural Convection Using CFD, Int. Journal of Engineering Research and Applications, 3(5), 1911-1913.
[15] Sami D Salman (2013), CFD analysis of heat transfer and friction factor characteristics in a circular tube fitted with horizontal baffles twisted tape inserts, International Conference on Mechanical Engineering Research, 3(5), 1-7.
[16] Pawan A. Sawarkar (2013), Experimental Analysis of Augmentation in Heat Transfer Coefficient Using Twisted Tape with Semi-Circular Cut Insert, International Journal of Science and Research, 4(4), 1174-1179.

