International Journal of Engineering Applied Sciences and Technology, 2019
Vol. 4, Issue 5, ISSN No. 2455-2143, Pages 529-535
Published Online September 2019 in IJEAST (http://www.ijeast.com)

NUMERICAL INVESTIGATION OF THERMAL AND FLUID CHARACTERISTICS OF FLUID FLOW INSIDE A SMOOTH TUBE HEAT EXCHANGER

Mr. P. T. Date, Mr. P. P. Shirpurkar
Department of Mech. Engg
Priyadarshini College of Engineering,
Nagpur, Maharashtra, India India

***Abstract*— The analysis of the thermal-fluid characteristics of smooth heat exchanger tube has been carried out numerically under the constant heat flux boundary condition. The Reynolds number was varied in the range from 3000 to 20000 for the air as fluid for analysis. The analysis results are discussed in terms of various parameters commonly used in such kind of analysis. The temperature profile along the wall of the heat exchanger tube is plotted and is being briefly discussed. The variation in Surface Heat Transfer Coefficient (HTC), total Surface Heat Flux are studied in the mentioned test condition. Temperature distribution is also shown over the 3D view for the fluid flow in the tube. Also the variation of skin friction is shown graphically. Fluid flow pattern are discussed with the help of boundary layer with respect to the inlet and outlet condition of the tube. Streamlines shows a convincing picture while analyzing the fluid flow nature from center of the tube to the inner wall of the tube.**

Keywords— **Heat exchanger, pipe flow, Numerical analysis, Ansys fluent.**

I. INTRODUCTION

Heat exchangers are commonly used heat exchanging device in many engineering application. Air conditioners, heat exchangers in power plant, chemical processes, waste heat recovery system and in industries like steel production etc. Due to wide applications of heat exchangers researchers are working from decades on the thermal performance of this device. Increasing thermal performance will give us more efficient heat exchanger, which will result in reduction in its size and overall operation cost. Heat transfer enhancement techniques are classified as active and passive type. In active type of heat transfer technique, heat transfer rate is enhanced by using some external power source. Whereas in passive type of heat transfer technique no external power source is needed and hence have become a point of interest by many researchers. Passive kind of techniques are easy to install, easy to maintain and are cost effective. Inserts in the tube of heat exchanger is one such technique of passive kind of heat transfer enhancement. Inserts like helical wires, angular blades, twisted tape etc and their various combinations is the most popular research section. These types of inserts create chaos in the usual fluid flow generating vortex or swirl and enhance the thermal interaction. While doing the numerical analysis of tubes of heat exchangers, we have to correlate the result for the fluid flow in smooth tubes with that of experimental results. And if the results get validated with the smooth tube case we can use the same setup with inserts in the tube for numerical analysis. So the analysis of tubes of heat exchangers with smooth tubes tube is the basic need of this kind of research. Also the results for inserts in tubes are mostly compared with that of smooth tubes. Hence for comparative study also the analysis of fluid flow through smooth tubes is essential. Based on several such research on smooth tubes and their validation, one such case is considered from the literature and analyzed in Computational Fluid Dynamics (CFD) software in this study (Promvonge, 2018). In this work the focus is on the actual interpretation of the result section of the CFD analysis which will be helpful for the beginners entering in this research domain. These interpretation may be simple and not worthy for most of the experts in this field but will definitely act as a guide for the amateur in the field of numerical analysis. For the purpose of simplicity and basic understanding the case of inserts in the tube of heat exchanger are not covered in this work. The visual results such as velocity profiles, velocity vectors, streamlines to explain the physics of fluid movement is used in several literature (Kumbhar D. G, et. al, 2015). Computational fluid dynamics software is useful tool in simulating the cases of fluid flow including the heat transfer. According to the computation facilities, the various inserts can be modeled in a tube of heat exchanger and simulated for variable flow conditions. Fluent is one such module which is used for observing the various fluid flow governing parameters (Altaie A., et al, 2015). The analysis of smooth

International Journal of Engineering Applied Sciences and Technology, 2019
Vol. 4, Issue 5, ISSN No. 2455-2143, Pages 529-535
Published Online September 2019 in IJEAST (http://www.ijeast.com)

tubes with insert and result in the form of velocity vectors, temperature contours and its comparison with corrugated tubes is shown in one such study (Yang L., et. al. 2016). The fluid entering the tube is fully developed or not is the important phenomenon to be considered while establishing the results. Similar is the situation for thermal boundary layer, it also need some distance of flow to get fully developed. The boundary condition for caring heat transfer from pipe wall to the fluid are of two types, the constant wall temperature and constant heat flux (Chamoli S, et. al. 2017). The CFD code embedded in Ansys software is popularly used for fluid flow analysis. Ansys workbench is the platform used which include modeling of the physical problem, meshing of the model, different algorithm for the solution of problem and post processing module for the analysis of results (Huminic G., et. al, 2011).

Physical Model: (Fig.1)
Length of Pipe = 2.5m
Diameter of Pipe; Inner diameter = 47mm, Outer diameter = 50 mm
Heated Section length = 1.5m in the centre
Section 1 = 0.5 m which is the inlet section for the fluid before entering the test section.
Section 2 = 0.5 m which the outlet section for the fluid after passing the test section.

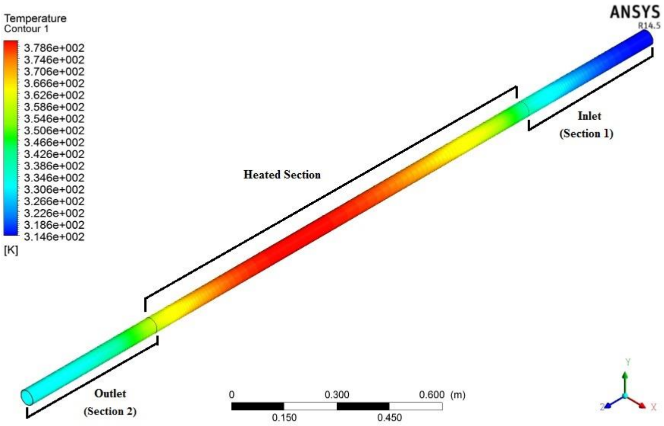


Fig. 1

Material Properties:
Pipe material - Copper, Fluid in the pipe - Air

Boundary Conditions:
Inlet condition of air, that is velocity is monitored from 1 m/s to 5 m/s corresponding variation in Re from 3000 to 20000. Inlet temperature of air is monitors at 300 K for all the cases. Heated section is given the boundary condition of constant heat flux which we have considered as 500 W/m2 for each case. Outlet condition for fluid is give as pressure outlet. For all the other geometrical and fluid based parameters the copper material is selected for the solid components and coupled or via system coupling thermal boundary condition is used.

Grid System:
The default mesh is selected for the computational domain for the flow in pipe using the software ICEM CFD 14.5.0.

Numerical Method:
Commercial software Fluent 14.5.0 is used for the fluid simulation. The governing conservation equations accompanied by the boundary conditions are discretized using the finite volume formulation. The standard pressure and second order upwind discretization schemes for the momentum and energy equations are employed in the numerical model. To achieve pressure-velocity coupling, the Semi-Implicit Method for Pressure Linked Equations-Consistent (SIMPLEC) algorithm is selected because it is more stable and economical compared to other algorithms. The computation is assumed to be converged when the relative residual values are less than 10^-6 for all variables.

Results and Discussions:

Verification of smooth tubes:
The current numerical results are first validated in terms of the Nusselt number. The calculated Nusselt number from the smooth tube case are compared to those of the well-known theoretical formulas. Gnielinski Correlation mentioned below is for Fully developed turbulent flow without considering the effect of viscosity.

$$Nu = \frac{(f/8)(Re-1000)Pr}{1+12.7\,(f/8)^{\frac{1}{2}}\,(Pr^{\frac{2}{3}}-1)}$$

for $3000 \leq Re \leq 5 \times 10^6$

The friction factor (f) is given as below which is then used in Gnielinski Correlation of Nusselt number.
The below mentioned equation is for the Transitional turbulent flow.
Blasius correlation,

for $3000 \leq Re \leq 5 \times 10^6$

The friction factor for ReD > 2300 and smooth tubes given by
$f = 0.25\,(1.82 \; x \; log_{10} Re_D - 1.64)^{-2}$
and for $Re_D > 10^4$
$f = 0.184 \; x \; Re_D^{\;-0.2}$
Friction factor (Darcy Friction factor) is also given by the formula mentioned below

$$f = \frac{\Delta P}{(L/D)\,\rho U^2/2}$$

Here,
- Pressure drop due to friction
U - Mean velocity in tube
For $Re_D < 2300$
$f = 64 / Re_D$

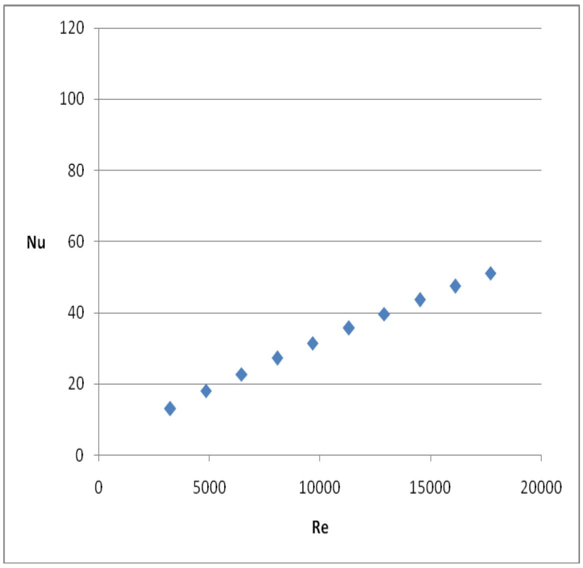
Dittus - Boelter correlation mentioned below is used for Fully Developed flow. Where Pr^0.4 for heating of the fluids and Pr^0.3 for the case of cooling of fluids. This equation is valid for 0.6 <Pr<100, 2500 < Re < 1250000 and L/D > 60
Dittus - Boelter correlation,

$$f = 0.023 Re^{0.8} Pr^{0.4}$$

for Re ≥ 10000
The length of pipe is 2.5m and diameter of 47mm. It gives L/D = 53.19 and literature suggest, the entry region of the turbulent zone. Since for fully developed region the condition is L/D > 60. Assuming the value is close to 60, it would be convenient to use Dittus Boelter correlation for further calculation.
This study have evaluated the fluid flow for the following velocities and Reynolds number:

Sr.no	Velocity [m/s]	Reynolds number
1	1	3218
2	1.5	4830
3	2	6440
4	2.5	8050
5	3	9650
6	3.5	11300
7	4	12900
8	4.5	14500
9	5	16100
10	5.5	17700

Fig. 2 shows that the agreement between the computational results and the theoretical values is satisfactory (the relative discrepancy is within 6%), which demonstrates that the present numerical predictions have a reasonable accuracy.



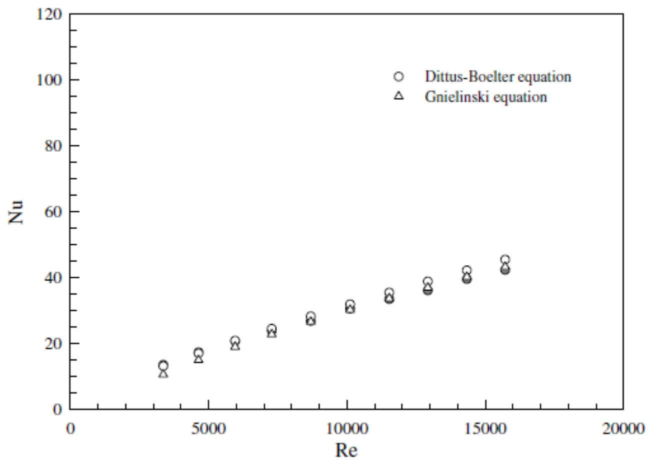


Fig. 2

Result and Discussion:

Verification of properties for flow through Smooth tube with respect to the standard literature available:
The temperature distribution (along the tube wall) for the pipe for its entire length of 2.5 m is shown below in Fig. 3. Note that the portion is red is the inlet section of the pipe and hence lower temperature is observed. The portion of the curve in black represents the heated section with constant heat flux boundary condition (Q = 500 W/m2). One can observe that the temperature of the wall keep on increasing along the length of the test section upto 2/3 of its length. Approximately at a distance of 1.5 m from the inlet (1m from the inlet of test section) the maximum value of temperature is observed.

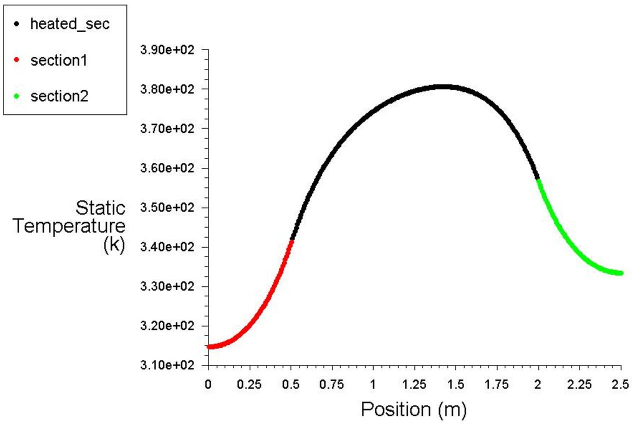


Fig. 3
TheFig.4 below gives the exact temperature at the outer surface of the tube wall (that is at a distance of 25mm from the central axis of the tube). For the length of tube in the range 1.25m - 1.45m the maximum value of temperature is observed which is around 380K. This value again get confirmed with the 3D temperature contour image.

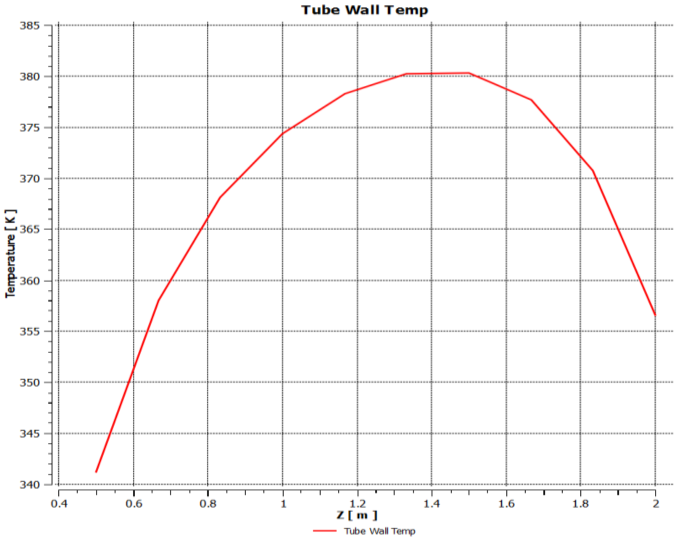


Fig. 4

The Fig.5 shown below gives the idea about the spread of heat transfer coefficient (HTC) for the fluid along the entire length of pipe. The curve in black represents the heat transfer coefficient for the pipe wall for the heated section. One can observe that the HTC have the higher value at the entry region of the heated section.

This is because, at the entry region the temperature difference between the fluid and the wall of the pipe is maximum, and hence the heat transfer rate is more at this section. With the progress in flow through the pipe the heat transfer get less effective and hence we can see the gradual drop in HTC.

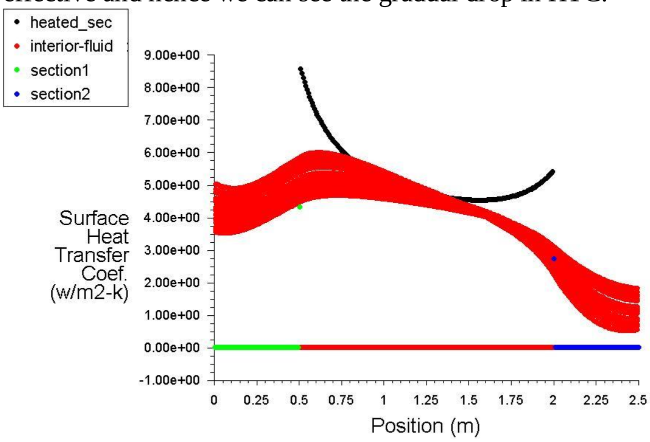


Fig. 5

Heat Flux:

The boundary condition at the tube wall is of constant heat flux. For this analysis we have maintained the heat flux at 500 W/m2. Black line represents the heat flux at the pipe wall which is constant. The heat flux reading for the fluid increases gradually and reaches it maximum value at 1/3 of the heated section as shown in Fig. 6.

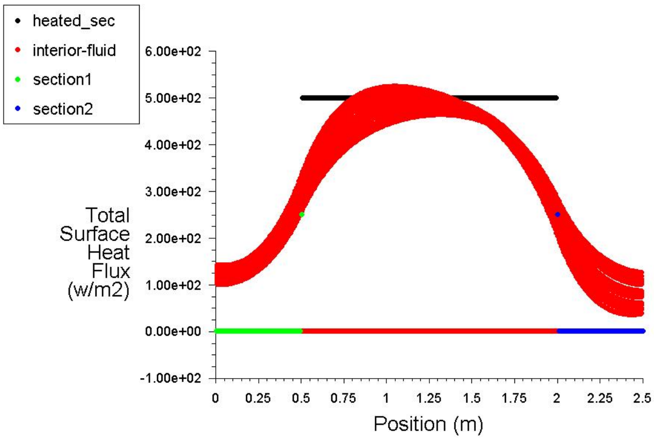


Fig. 6

Skin Friction Coefficient:

Skin friction coefficient (SFC) is the defining parameter of the resistance in the flow. More the SFC more the resistance. From the graph Fig. 7 it is clear that at the beginning of the fluid flow the skin friction at the wall is higher and goes on decreasing along the length of pipe. This is because at the beginning of the flow the due to no slip phenomenon more resistance to the flow is observed. As the flow progress the hydraulic boundary layer get fully developed and thus the resistance to the flow reaches the steady value.

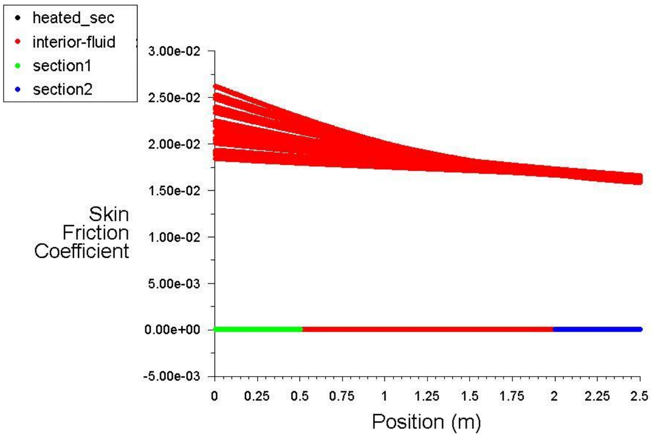


Fig. 7

Temperature distribution along the length of pipe:

From Fig. 8 since Section 1 is our inlet of the fluid the lowest temperature zone is observed at this location. In Heated section the heating begins and higher temperature zone can be spotted. At the region approx. 2/3 of the test section, maximum temperature region at the pipe wall is observed. At the outlet of the heated section the lower temperature zone is observed since it is close to the exit where the normal temperature air coexists with the heated air.

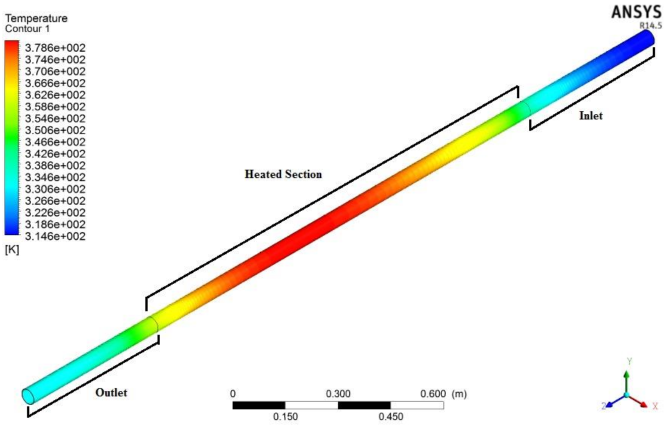


Fig. 8

Fluid flow temperature profile:

The Fig. 9 below shows the temperature profile of the fluid exactly at the centre of the pipe. At the centre of the pipe the fluid will be rarely effected by the heated wall and hence the temperature of the fluid will not change noticeably.

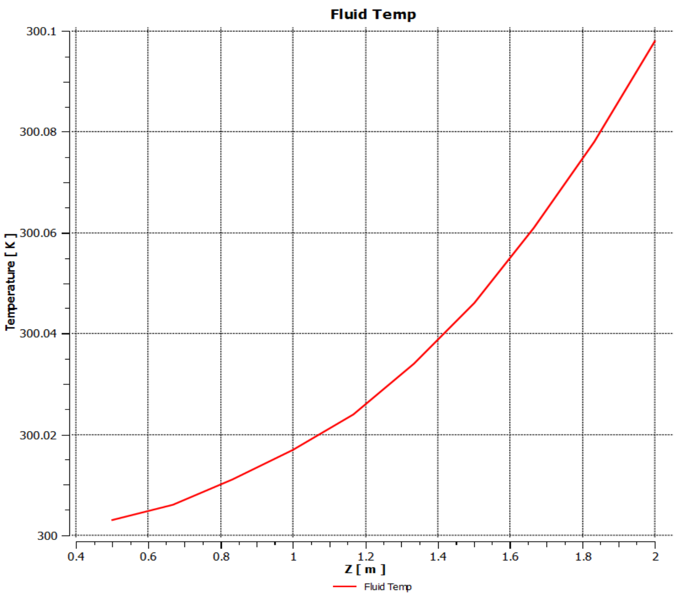


Fig. 9

To note down the behavior of the fluid from the centre of the pipe to the wall of the pipe, the stream line command is used and the temperature profile is plotted for the same shown in Fig. 10. The temperature profile showing maximum deviation from the temperature of 300K are the streamlines close to the pipe wall. The streamlines which are away from the pipe wall and at the centre of the fluid flow shows very little change in the temperature along its flow path.

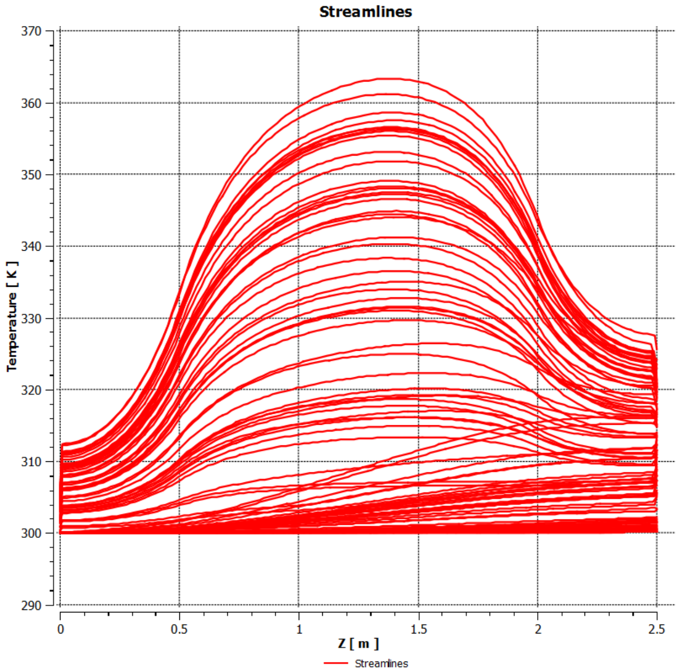


Fig. 10

Inlet Temperature Profile to the Heated Pipe:

At the heated section the constant heat flux condition is maintained. Hence the temperature interaction starts at the heated section of the pipe as shown in Fig. 11. One can clearly see that the temperature profile is in building stage at the inlet. Whereas the condition at the outlet is showing quite a stable temperature profile as shown in Fig. 12. Stable temperature profile also resembles a steady state of heat transfer.

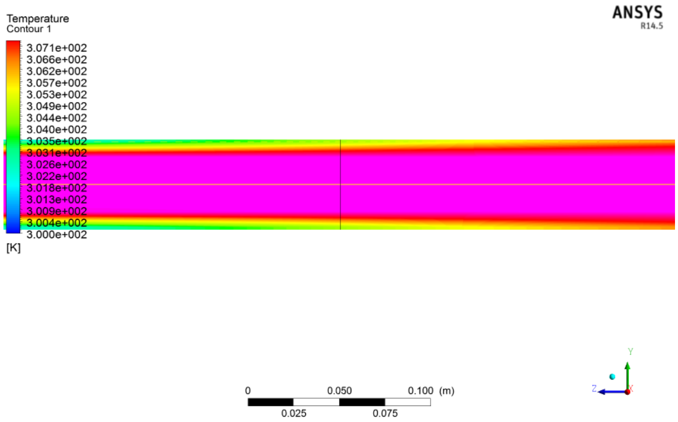


Fig. 11

International Journal of Engineering Applied Sciences and Technology, 2019
Vol. 4, Issue 5, ISSN No. 2455-2143, Pages 529-535
Published Online September 2019 in IJEAST (http://www.ijeast.com)

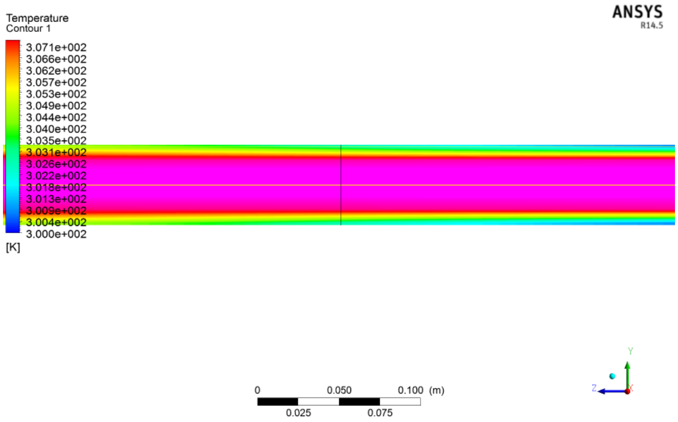


Fig. 12

Inlet Velocity profile to the heat pipe:
The inlet velocity shows a stable profile as shown in Fig. 13 at the boundary of the pipe. This also means that the hydrodynamic boundary layer is fully developed. The similar profile is observed at the outlet of the heated section in Fig. 14. But the temperature range is higher as compared to the inlet condition.

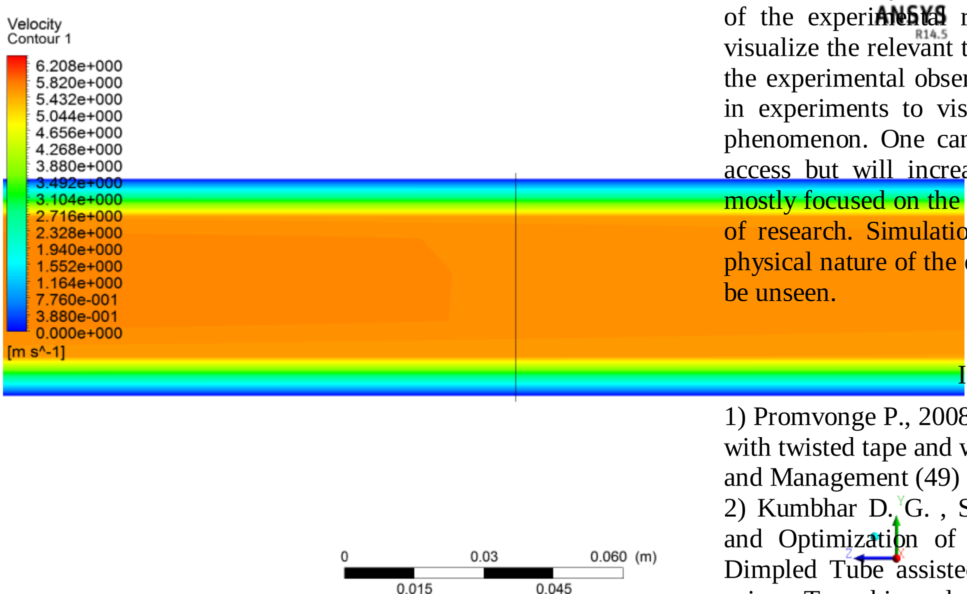


Fig. 13

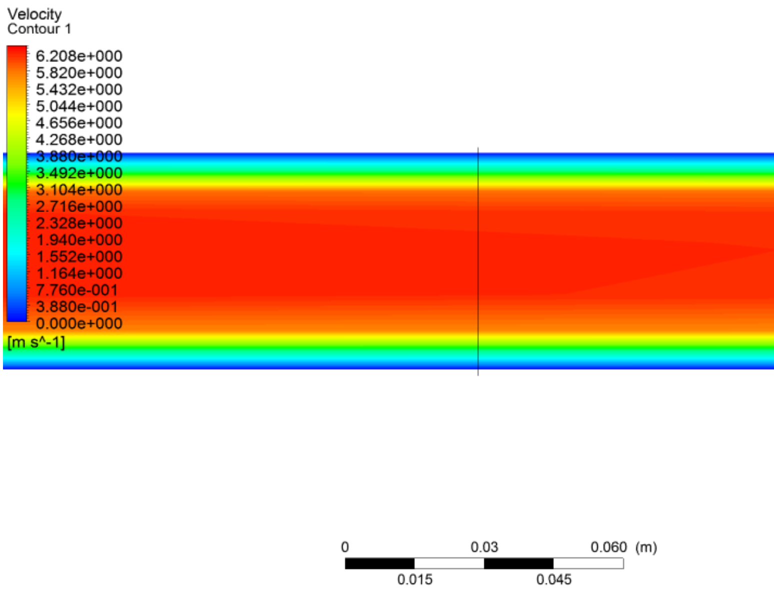


Fig. 14

II. CONCLUSION

Numerical analysis as in case of CFD gives us a broader view of the experimental results. It is easier to understand and visualize the relevant theory on CFD software as compared to the experimental observations. Most of the time it is difficult in experiments to visually observe the physical or thermal phenomenon. One can definitely built the setup with visual access but will increase the setup cost. Experimentation is mostly focused on the end results and not on the visual aspects of research. Simulation software helps us to understand the physical nature of the experimentation which would otherwise be unseen.

III. REFERENCE

1) Promvonge P., 2008, Thermal augmentation in circular tube with twisted tape and wire coil turbulators, Energy Conversion and Management (49) (2949-2955)
2) Kumbhar D. G. , Sane N. K., 2015, Numerical Analysis and Optimization of Heat Transfer and Friction factor in Dimpled Tube assisted with regularly spaced Twisted tapes using Taguchi and Grey Relational Analysis. Procedia Engineering (127) (652-659)
3) Altaie A, Hasan M. R., Rashid F. L., N
Arkan Altaie, Moayed R. Hasan, Farhan Lafta Rashid, 2015, Numerical Investigation of Heat Transfer Enhancement in a Circular Tube with Rectangular Opened Rings. Bulletin of Electrical Engineering and Informatics, 2089-3191(18-25).
4) Yang L, Han H, Li Yanjun, Li Xiaoming, 2016, A Numerical Study of the Flow and Heat Transfer Charateristics of Outward Convex Corrugated Tubes with Twisted Tape Insert, Journal of Heat Transfer, (024501) (1-8)

5) Chamoli S, Lu Ruixin, Yu Peng, 2017, Thermal Characteristic of a turbulent flow through a circular tube fitted with perforated vortex generator inserts, Applied Thermal Engineering (121) (1117-1134)
6) Huminic G, Huminic A, 2011, Heat Transfer Characteristics in Double tube helical heat exchanger using nanofluids, International Journal of Heat and Mass Transfer, (54) (4280-4287)