International Journal of Engineering and Applied Physics (IJEAP)
Vol. 1, No. 2, May 2021, pp. 90~95
ISSN: 2737-8071

Simulation of heat transfer augmentation during constant solar heat flux in circular pipe

Chandan D. Chaudhari[1], Kaustubh V. Chavan[2]
[1,2]Dept. of Mechanical Engineering, SIES Graduate School of Technology, University of Mumbai, India

Article Info

Article history:

Received Jan 19, 2021
Revised Apr 14, 2021
Accepted Apr 20, 2021

Keywords:

Nusselt Number
Solar flux
Computational Fluid Dynamics
Reynolds Number
Heat transfer augmentation

ABSTRACT

The aim of the current research is to evaluate the flow characteristics and heat transfer enhancement in a circular hose having a diameter of 12.5 mm and 1500 mm Length subjected to a constant solar heat flux of (1000 W/m2) using numerical simulation. The study evolves the changing Reynolds numbers (5000 < Re < 10500) and its effect on the Nusselt number during variable flow rate conditions from 0.45 m/sec to 0.90 m/sec. The heat transfer phenomenon was evaluated during controlled conditions with working fluid as water. The results obtained from numerical investigations were studied to understand the flow and heat transfer enhancement, Nusselt number phenomenon, outlet temperature, and fluid fully developed region. State the most important part of your findings and achievements

Corresponding Author:

Prof. Chandan D. Chaudhari,
SIES Graduate School of Technology,
Mumbai, India.
Email: chaudhari.chandan22@gmail.com

1. INTRODUCTION

The heat transfer augmentation has been urbanized and extensively advantageous in heat exchanger applications over the past decade in numerous submissions which may include refrigeration sector, automotive, process industries, nuclear reactors technologies, and solar technologies etc. [1,2]. There have been many attempts made by researchers to enhance the heat transfer with diminishing the size and the costs of the heat exchangers. The most protruding factors analyzed such as heat transfer coefficients and pressure drop, which generally lead to reduction of capital cost [3]. Energy is one of the important concerns for sustainable development for developing countries like India. The ample availability and truncated cost are the two tactical advantages of solar energy [4]. As solar energy is an imperative portion of renewable energy sources; to overcome energy difficulties issues, solar energy can be used with different configuration of collectors such as flat plate collector, heliostat field and compound parabolic collectors etc. For efficient energy conversion process, exact position of the receiver is required since It depends upon the various parameters such as ambient temperature, fluid inlet temperature, heat flux, working fluid, receiver material, fluid flow rate, surface friction between working fluid and receiver material, reflector material etc [7]. Many heat transfer improvement methods were examined and designated by various researchers in past decades. Although abundant studies have been investigated the thermal performance of solar receivers, the effect of water and other fluids with variable Nusselt number and Reynolds number during constant heat flux condition to advance thermal performance of the receiver is not available in the literature. The Nusselt number (NU) is found as the function of Reynold number and Prandlt number respectively this function based on energy and momentum transfer analogies to calculate heat transfer coefficient in pipe for turbulent flow Dittus-bolter equation expressed in Equation 2. Here exponent term associated with Prandlt number used 0.4 which is used for heating of fluid. For Uniform Heat Flux (UHF) for pipe domain with turbulent flow regime approximately thermal entry length would be ten times of the hydraulic diameter of the pipe domain. Research found that thermal entry length is 115 mm. The length of pipe section must be considered as per the standard norms i.e. L/D> 60.

2. SIMULATION METHOD

The following are the basic assumptions considered for the Numerical Simulation. The assumptions are listed for Numerical analysis:

- Condition of heat transfer: **-** Stead State
- Flow behavior of fluid in receiver: **-** Uniform with Fully developed condition.
- Fluid Nature: **-** Incompressible condition.
- Velocity Inlet and Outflow boundary Condition,
- Semi Implicit Pressure Linked Equations (SIMPLE) scheme is used

2.1 Geometric Aspects: The Commercial software was used for geometry creation and modeling. The 2D fluid geometry modeled was considered for analysis for simplicity since it can be validated easily. The Length of pipe is 1500 mm, diameter of 12.5 mm. The meshing of the geometry is done by use of the ICEM tool. The figure. 1 shows computation domain model with mesh.

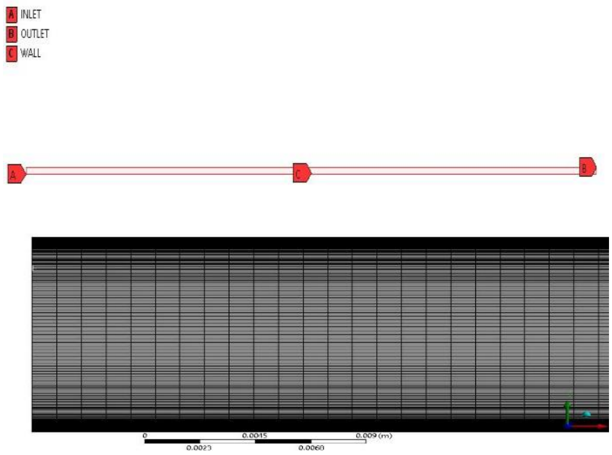


Figure 1: Meshing of Pipe Domain

The physical properties of water are considered for the numerical simulation and listed as shown in Table 1.

Table 1**.** Thermal properties of working fluid (water)

Properties	*Water*
Density	998.2 (kg/m^3)
Specific heat	4182 (J/kg.K)
Viscosity	0.001003 (kg/m·s)
Thermal conductivity	0.6 (W/m.K)

2.2 Meshing Approach for Modeling: The CFD analysis is carried out with uniform heat flux. In presented study constant heat flux of value 1000 W/m^2 is instigated on pipe wall surface using UHF i.e. uniform heat flux condition. Some assumption was considered during numerical simulation such as steady state condition, incompressible turbulent flow, no heat loss etc. Outlet flow and temperature fields are assumed fully developed as ((x/D) > 10) [11]. The velocity inlet is used for specifying the velocity for different flow conditions and Outflow boundary condition was used since this boundary condition specifies zero normal gradients for all flow variables excluding pressure [12]. The computational simulation follows basic governing equations such as continuity equation, Momentum equation and energy equation as expressed follows:

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho.v) = 0 \tag{1}$$

$$\frac{\partial}{\partial t}(\rho v) + \nabla.(\rho v v) = -\nabla p + \rho g + F \tag{2}$$

$$\frac{\partial}{\partial t}(\rho E)+\nabla.[v(\rho E+p)]=0 \quad (3)$$

The Navier-Stokes equation tied with energy equation and turbulent viscosity model is solved for fully developed water flow condition. SIMPLE Technique i.e. [Semi Implicit Pressure Linked Equations] is suggested for simulation for lesser computational efforts [17, 18]. Second order up winding scheme was selected for Pressure, Momentum, and Energy. To control the variables of each cell under relaxation factors keeping to a constant default value. As per the theory suggested by Rayleigh Ritz, the residue is an error for any computational method. So, keeping the value of residue minimum the accuracy of the solution is achieved. Hence Simulation run for the of 1x10-6 [19]. The workstation used is a i5 Processor (4 threads) processor, 2.3 GHz CPU with 16 GB RAM. Each iteration took average time of 4 hours. Grid sensitivity is checked for pipe domain for every iteration. The grid used in the present analysis was 120 × 1500, 120 in r-direction and 1500 in x-direction. We also tested 180×1600 and 200 × 1500. All gave similar values of velocity and temperature at the outlet. Therefore, 120×1500 was selected as the optimal grid size. The number of elements formed are 270000. This grid size was validated by our computational results as shown in Figure. 3 and 4. The Orthogonal Quality of mesh metric plays an important role during computation as orthogonal quality of mesh lies between 0 to 1. Higher the value of orthogonal quality results in less skewness of mesh i.e. less distortion of Mesh element. So, with the selected grid size the orthogonal quality of mesh is found to be 1 [20]. Hence Numerical Simulations were agreed on for different values of Reynold number. The inlet temperature Tin is considered as 300 K i.e. 27 °C.

2.3 Modeling Technique: In order to validate the computational model, the numerical results were compared with the theoretical data available for the conventional fluids [23].
The Darcy's friction factor equation given by Blasius as,

$$f = 4(0.0791)Re^{-0.255} \quad (4)$$

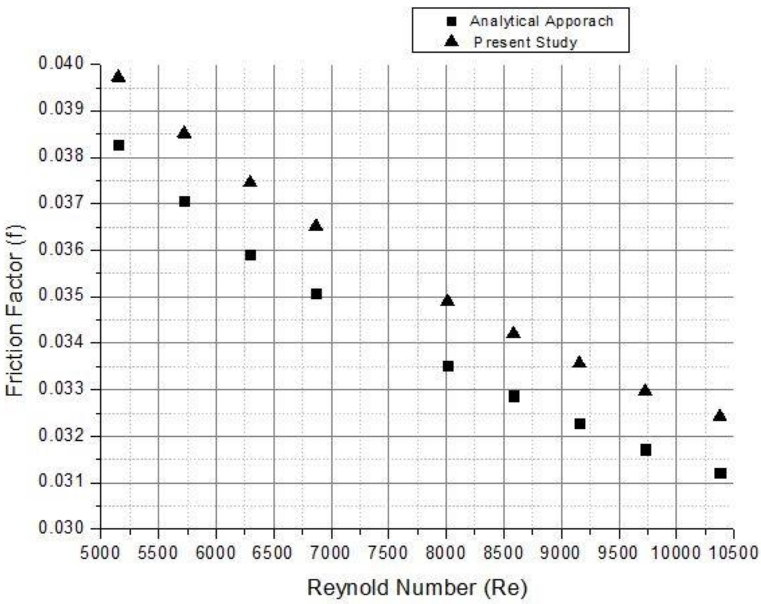


Figure 2: Friction Factor vs Reynold Number

The Figure.2 shows the comparison of Darcy friction factor from Blasius formula and computed values from simulation. An excellent agreement is observed with maximum deviation and average deviation of computed values from theoretical equation being 3.2 and 1.9%, respectively, over the range of Reynolds numbers studied.

3. RESULT AND DISCUSSION

The detail study was carried out to understand the effect of change in Reynolds Number and Nusselt number. The Reynolds number plays important role in heat transfer process and temperature at the outlet of the pipe. The different equations are used to understand the Nusselt Number, Reynolds Number, Turbulent kinetic energy and diameter etc. was studied [24].

Dittus Bolter equation

$$N_U = 0.024\ R_e^{0.8} P_r^{0.4} \tag{5}$$

Pak and Cho equation

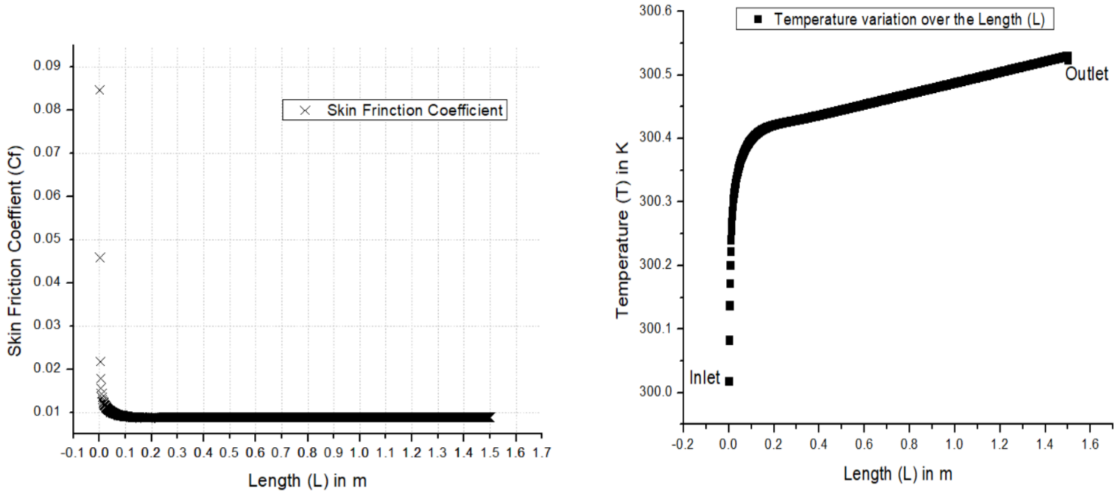
$$N_U = 0.021\ R_e^{0.8} P_r^{0.5} \tag{6}$$

Maiga equation

$$N_U = 0.085\ R_e^{0.71} P_r^{0.35} \tag{7}$$

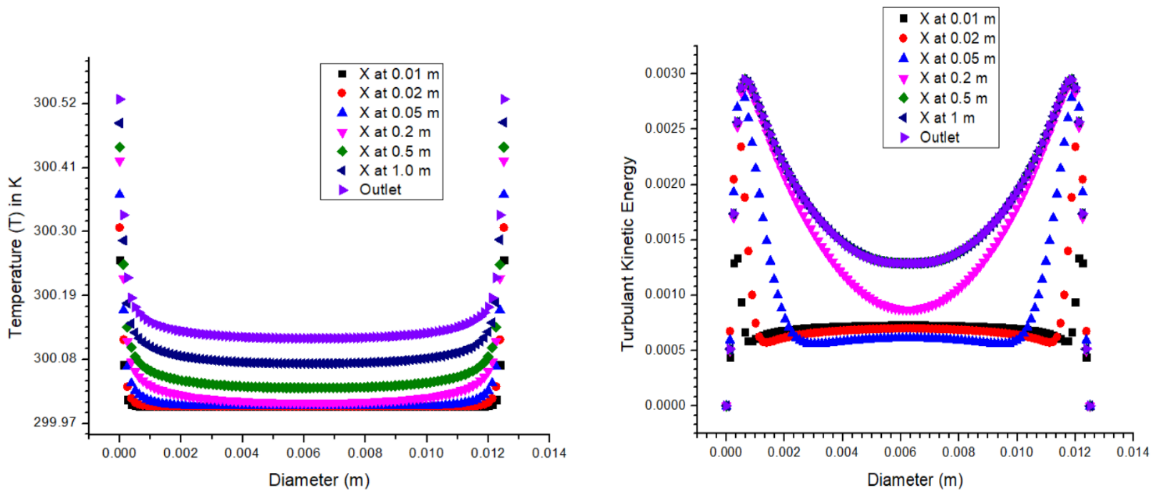
Gnieinkski equation

$$Nu = 0.012(\ Re^{0.87} - 280)\ Pr^{0.4} \tag{8}$$



[a] [b]

Figure 3: [a] Skin friction coeffiecient vs Length; [b] Tempreture vs Length



[a] [b]

Figure 4: [a] Temperature variation at interval lengths; [b] Turbulant kinetic energy variation at interval length

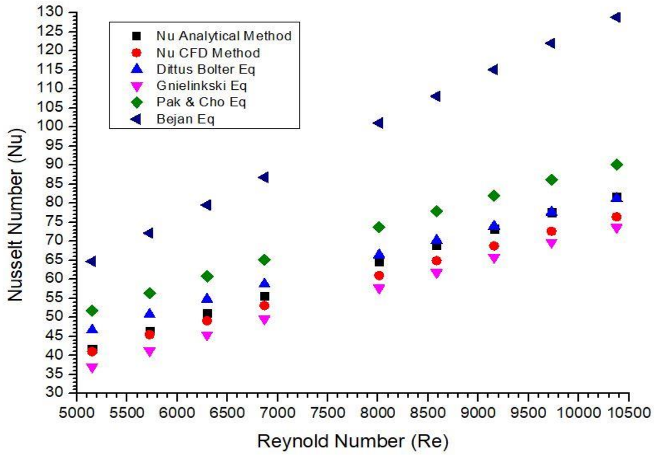


Figure 5: Variation of Nussel number w.r.t Reynold number

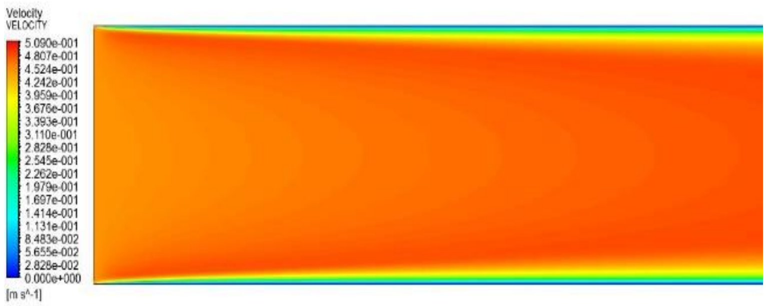


Figure 6: Velocity plot in circular hose pipe

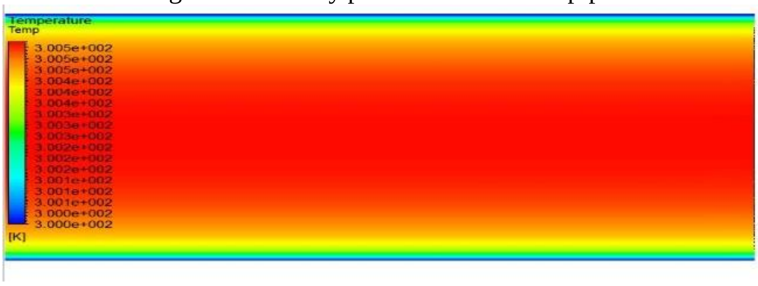


Figure 7: Temperatuer plot in circular hose pipe

The Figure 3 [a] and [b] shows a comparison between the numerically results and the Nusselt number correlations. The results compared with lower range of Reynolds number shows predicted results of Nusselt number is very close to all correlations at lower velocity as compared to higher range of Reynolds number. Exactly the same trend was observed for all computed concentrations.

4. CONCLUSION & FUTURE PERSPECTIVE

In this research paper, the detail analysis of effect of numerical parameters on the dimensionless numbers like Nusselt number, Reynolds number and Prandtl number. Various Nusselt number equations are identified and compared to study the effect and result analysis. The Dittus Bolter equation shows close results as compared to other Nusselt number equations and theoretical results. Also, the study was carryout to understand turbulent kinetic energy verse diameter of pipe shows the proper flow distribution in case of Dittus Bolter equation. The percentage error between the theoretical and numerical results in case of Dittus Bolter equation results was 6 to 8 percentage. The results were found to be in good agreement between the numerical simulation and other numerical equations. The numerical study can be further extended to investigate another configuration and optimized.

REFERENCES

[1] Chavan Kaustubh, Satpute Jitendra, Gulhane Nitin, (2015). "Experimental and fluid structure interaction study of triangular receiver with square fin arrangement using compound parabolic collector", RESD2015-Proceedings of the International Seminar on Renewable Energy and Sustainable Development, 2, 200-206

[2] Chew, T. C.,Tay, A. O.,Wijeysundera, N. E., (1989). "A Numerical Study of the Natural Convection in CPC Solar Collector Cavities with Tubular Absorbers", Journal of Solar Energy Engineering,111, 16-23
[3] Jadhav, A. S., Gudekar, A. S.,Patil, R.G., Kale, D. M., Panse, S. V., (2013), "Performance analysis of a novel and cost effective CPC system", Energy Conversion and Management, Vol. 66, pp. 56-65
[4] Javier Muñoz, Alberto Abánades, (2011), "Analysis of internal helically finned tubes for parabolic trough design by CFD tools", Applied Energy, Vol. 88, pp. 4139–4149
[5] Scampoli Steve,(2012) " ANSYS advantage", Vol. 6 , No. 1, pp. 14-16
[6] John W. Allen, Norman M. Levitz , Ari Rabl, Kent A, Reed, William W, Schertz, George Thodos, and Roland Winston, (1975),"Development And Demonstration Of Compound Parabolic Concentrators For Solar Thermal Power Generation And Heating And Cooling Applications", Progress Report.
[7] SA Waghmare, KV Chavan, PB Muntode, NP Gulhane, (2017), "Experimentation of surface areal irradiance method for solar flux measurement in compound parabolic collector", Optik, Vol. 147, pp. 373-384.
[8] Romero, V.M., Cerezo, E., Garcia, M.I., Sanchez, M.H., (2014), "Simulation and validation of vanilla drying process in an indirect solar dryer prototype using CFD Fluent program" ,Energy Procedia, 57, 1651-1658.
[9] NP Gulhane, KV Chavan, (2014), "A critical technological literature review of–photovoltaic/thermal solar collectors system", Int J Current Eng Tech, Vol. 3, pp. 258-264"
[10] Roldán, M I; Valenzuela, L; Zarza, E; (2013), "Thermal analysis of solar receiver pipes with superheated steam", Applied Energy, 103, 73-84.
[11] K Chavan, N Gulhane, (2017), "Performance analysis of compound parabolic collector with extended surfaces inside the triangular receiver", International Journal of Renewable Energy Technology, Vol. 8 (1), pp. 24-46
[12] Braga, C.V.M., Saboya, F.E.M., (1999)."Turbulent heat transfer, pressure drop and fin efficiency in annular regions with continuous longitudinal rectangular fins", Experimental Thermal and Fluid Science, 20(2), 55-65
[13] SA Waghmare, KV Chavan, NP Gulhane, (2018), "Numerical simulation of tracking modes for compound parabolic collector with tubular receiver", IEEE Transactions on Industry Applications, Vol. 55 (2), pp. 1882-1889
[14] Mark, J. C., (1998), "Demonstration of the effect of flow regime on pressure drop", Mathematica- Introduction to multifluid, 1-69
[15] Sukumar, R Sam, Sriharsha, G, Arun, S Bala, Dilip, P, Sanyasi, Ch, (2013)."Modelling And Analysis Of Heat Sink With Rectangular Fins Having Through Holes", International Journal of Engineering Research and Applications, 3(2), 1557-1561
[16] MersihaŠumić, (2014). "Thermal Performance of a Solarus CPC-Thermal Collector", Master Level Thesis, European Solar Engineering School, 186, 1-54.
[17] Stefanovi, Velimir; Pavlovi, Saša; Apostolovi, Nenad; (2011), "Mathematical model and numerical simulation of cpc-2v concentrating solar collector", Automatic Control and Robotic, Facta Universitatis, 10(2),153 – 166.
[18] K Chavan, N Gulhane, (2016), "Performance investigation of compound parabolic collector system with triangular receiver with and without internal fins, International Journal of Energy and Statistics, Vol. 4 (01), pp. 1650004
[19] Soteris A. Kalogirou, (2004), "Solar thermal collectors and applications", Progress in Energy and Combustion Science Vol.30, pp 231–295.
[20] S P Sukhatme, J K Nayak, (2008), "Solar Energy Principles of Thermal Collection and Storage", Third Edition, Delhi
[21] Nasif Shams, (2013), "Design of a Transpired Air Heating Solar Collector with an Inverted Perforated Absorber and Asymmetric Compound Parabolic Concentrator", Doctoral Thesis report, Dublin Institute of Technology.
[22] Yunus Cengel and Afshin Ghajar, (2014), "Heat and Mass Transfer: Fundamentals and Applications", Fifth Edition, Mc Graw Hill Education.
[23] Praveen K. Namburu, Debendra K. Das, Krishna M. Tanguturi, Ravikanth S. Vajjha, (2009), "Numerical study of turbulent flow and heat transfer characteristics of nanofluids considering variable properties", International Journal of Thermal Sciences, Vol. 48, pp. 290–302
[24] A. Akbarinia, A. Behzadmehr, (2007), "Numerical study of laminar mixed convection of a nanofluid in horizontal curved tubes", Applied Thermal Engineering, Vol. 27, pp. 1327–1337