Numerical Solution of the Flow of Air Inside a Circular Duct Encountering Series of Vertically Hanging Heated Circular Baffled Discs Provided at Different Angles

Bamaniya Jayesh[1*], *Kevat Mayur*[1], and *Patel Amit*[1]

[1]Department of Mechanical Engineering, Faculty of Technology and Engineering, The M S University of Baroda, Vadodara-390001, Gujarat, India

Abstract. Efficient vortex generators are the key for heat transfer augmentation. In the present work, a novel approach of generating vortex in the proximity of the heat transfer surface is examined using CFD analysis. This novel approach combines the vortex generator and heat transfer component of heat transfer augmentation. The fluid flow in a circular duct around a series of heated circular baffled discs is considered for this. The CFD analysis is carried out in Ansys Fluent by varying Reynolds number (Re) for a particular heat flux and for different angles of discs as 15˚, 30˚, 45˚, 60˚ and 75˚. Present work includes thermo-hydraulic analysis by calculating temperature difference, pressure drop and Nusselt number (Nu), for different flow configurations. It is observed that the pressure drop increases with the increase in the angle of the discs from 15˚ to 75˚. Thermal analysis through the trend of Nu v/s Re is carried out and the heat transfer performance is compared with Dittus-Boelter correlation.

1 Introduction

Vortex generators are used as a common heat transfer augmentation technique. These techniques are basically utilized to increase the turbulence levels in the flow inside an enclosed duct regime. The basic principle of these techniques is to increase the contact area of the cold flow with the heated surfaces which increases the enthalpy of the flowing fluid for various purposes. In these techniques, the heating is done through the surfaces of the duct, so these vortex generators are designed in such a way that the cold fluid away from the duct surface at the axis of the geometry gets contact from the surface of the duct. These techniques utilize methods like insertion of twisted tapes [1], winglets [2], baffles [3], wire coil [4], and many others.

In a novel method for heating cold fluid, the authors have found a novel approach to this heat transfer mechanism. In this technique, the authors utilized heated circular discs to increase the enthalpy of the cold fluid using resistance heating through electrical energy supply. In

* Corresponding author: jayeshbamaniya448@gmail.com

this study, the heated circular discs themselves work as a vortex or turbulence generator for the flow inside the duct. The method utilizes internal heating instead of external heating via duct surface. There are numerous studies available for the vortex generators in which the surface of the duct is heated [5]. But as per author's knowledge, no such study is available for this method. In an analytical investigation of the entropy generation rates for the optimal designs for laminar forced convection by Mohammad Ahadi and Abbas Abbassi [6], the authors utilized the heated coil as heat transfer component for the flow passage. In the author's previous works, an attempt was made for the investigation of the performance of the flow of air around circular heated discs like obstructions [7]. The CFD analysis was carried out in which the heated circular discs were placed perpendicular to the flow of air. It was observed that the flow profile was symmetric, and the pressure drop was significant. By taking this study further, the authors have done the CFD analysis of flow of air around a series of heated circular discs at a different angle [8]. The angle of them varied from 0˚ to 45˚ where 0˚ means the axial flow perpendicular to the flow of air. Pressure drop and temperature gain for different flow configurations were analysed. It was observed that the heat transfers from the discs to air increased by 45% with an increase in the angle of heated circular discs for a fixed Reynolds number (Re). Whereas the overall pressure drop decreased by 70% as the angle increased. To study the effect of geometrical variations for the proposed technique, numerical study was carried out [9]. Different geometries of different diametrical variation and pitch variation were studied numerically for the heated circular discs perpendicular to the flow of air. It was observed that diametrical ratio of 1.2 and pitch to diameter ratio of 0.7 was best suited for the maximum heat transfer with much less pressure drop. In the present study, the authors are further investigating the effects of these heated circular discs by varying the angle of it. The effects of angle variation of heat transfer are analysed by estimating the Nusselt number (Nu) by varying the Re for the cases of 15˚, 30˚, 45˚, 60˚ and 75˚.

2 Modelling and Meshing

2.1 Geometry

The geometry was created in Solid works CAD modelling software. 12 number of heaters at an equal distance of approximately 0.15 m was modelled. The duct diameter was taken as 6 inches (0.1524 m), and the diameter of the heated circular discs was taken as 5 inches (0.127 m). The CAD model file was converted to STEP file and imported into the ANSYS Design Modeler for the further CFD analysis. A basic geometry is shown in Fig. 1.

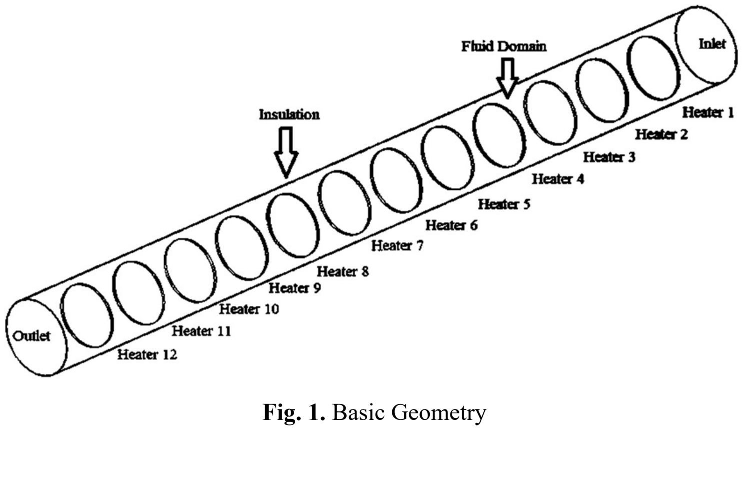


Fig. 1. Basic Geometry

In the present work, 12 circular discs at different angles were used as a turbulence generator. The nomenclature adopted for the angular position of the heated circular discs are described in Fig. 2. The change in the angular position of the discs is shown by a 2D geometry in Fig. 2. The angles were specified with respect to the direction of the flow. The angular position of the heated circular discs normal to the direction of the flow was considered as 90˚ and the position parallel or axial to the flow was considered as 0˚. As shown in Fig. 2, from 90˚, the angle was changed clockwise when seen from the top view. The change in the angle considered to be 15˚, starting from 90˚. The cases considered in the present study were 15˚, 30˚, 45˚, 60˚ and 75˚. The discussion of cases of 0˚ and 90˚ were ignored from the numerical simulation due to the worst-case scenario in terms of heat transfer performance. The direction of the flow for the figure was from bottom to top as shown.

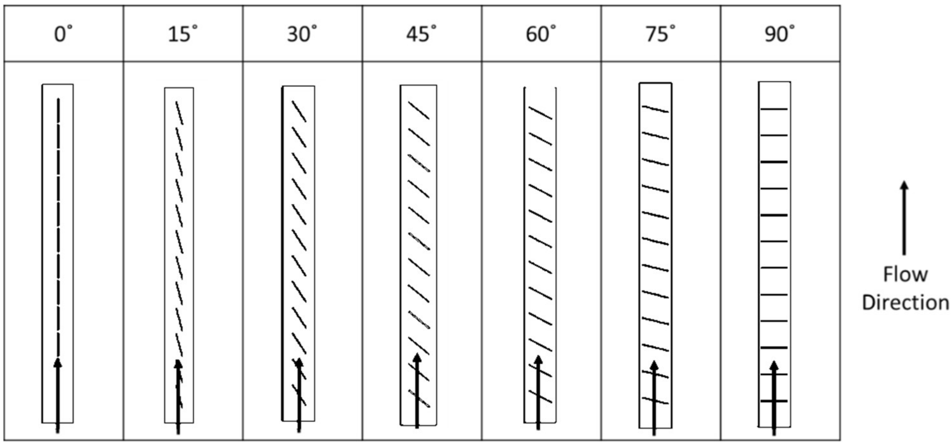


Fig. 2. Top view of the discs at different angles

2.2 Meshing

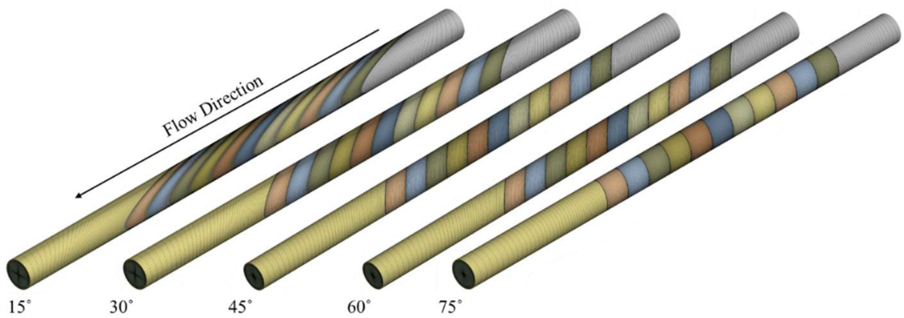


Fig. 3. Meshing of the heater circular discs at different angles

Meshing is the process of dividing the computational domain into number of computational elements. This process is also known as discretization. Different sub volumes were created in the geometry modelling meshed simultaneously and methodically to find the uniform quad mesh. The main objective of the meshing methodology adopted in the study was to find the best possible configuration of the meshing inputs to make quadrilateral mesh elements for all the cases. The reason for this was the fast convergence time and accurate solutions. The final mesh output can be seen in Fig. 3. A fine mesh near the heater discs were facilitated by using the bias option available in the edge sizing. The reason was to capture the accurate turbulence zones near the heated circular discs. These discs are responsible for the onset of turbulence generation. The same edge sizing and biasing options were utilized for the further meshing all the elements.

3 Solution Methodology

3.1 Boundary Conditions

After achieving the uniform meshing, the model was sent to ANSYS Fluent solution module where the following boundary conditions and the solution methodology were adopted. The angular positions of heated circular discs considered were 15˚, 30˚, 45˚, 60˚ and 75˚as mentioned earlier. The velocity at inlet was applied with different values. At the outlet, the atmospheric conditions were achieved through zero-gauge pressure input. Constant heat flux of 149.46 W/m^2 (for grid sensitivity analysis) and 336.28 W/m^2 (for performance analysis) was applied to all the heater surfaces at the front and back side. The side surfaces of the heater were kept insulated along with insulated duct walls. The material chosen for the wall as steel and air as the fluid medium from the fluent database.

3.2 Solution Methods

The energy equation and SST k-ω model are selected as solution models. Pressure-based steady state solver is employed. For pressure-velocity coupling, coupled scheme was used. Other considerations include spatial discretization, Least Squares Cell based gradient, Standard Pressure and Second Order Upwind Momentum, Turbulent Kinetic Energy and Turbulent Dissipation Rate were considered. For all flows, ANSYS Fluent solves conservation equations for continuity and momentum. For flows involving heat transfer, an additional equation for energy conservation is solved. The residual value of 10^{-3} was taken as convergence criteria for continuity, x, y and z momentum equations. Whereas 10^{-6} was taken as convergence criteria for the energy equation.

4 Results and Discussion

After investigating a uniform meshing methodology for all the cases, the grid sensitivity test was carried out by uniformly varying all the meshing parameters. After arriving at the grid independence solution, the numerical solutions were carried out by considering the solution methodology discussed in the previous section. Thermo-hydraulic performance analysis was carried out by finding the temperature difference, pressure drop as well as the change in Nusselt number (Nu) for all the heated circular discs angular positions. A comparative analysis was carried out to find the best configuration of the heated circular discs in terms of angular position with respect to the direction of the flow.

4.1 Grid Independence Test

The numerical solutions are largely influenced by the meshing. The number of elements, grid sizes and the fine mesh influences the numerical simulations. For getting grid independent solutions, temperature values were calculated numerically by varying grid sizes. Seven number of grid sizes were considered which were 368000, 478000, 588000, 738000, 848000, 896000 and 986800. For the same input parameters as discussed in the previous section, the output thermal parameter was calculated.

The temperature at the front side of heater number 7 was calculated for different grid sizes. The Fig. 4 shows the change in temperature with respect to the change in number of meshing elements for all the cases of angular positions. As per the data, it was observed that there were almost constant values of temperature after 848000 number of elements. Hence, the grid size of the 848000 number of elements could be taken as a grid independent meshing solution.

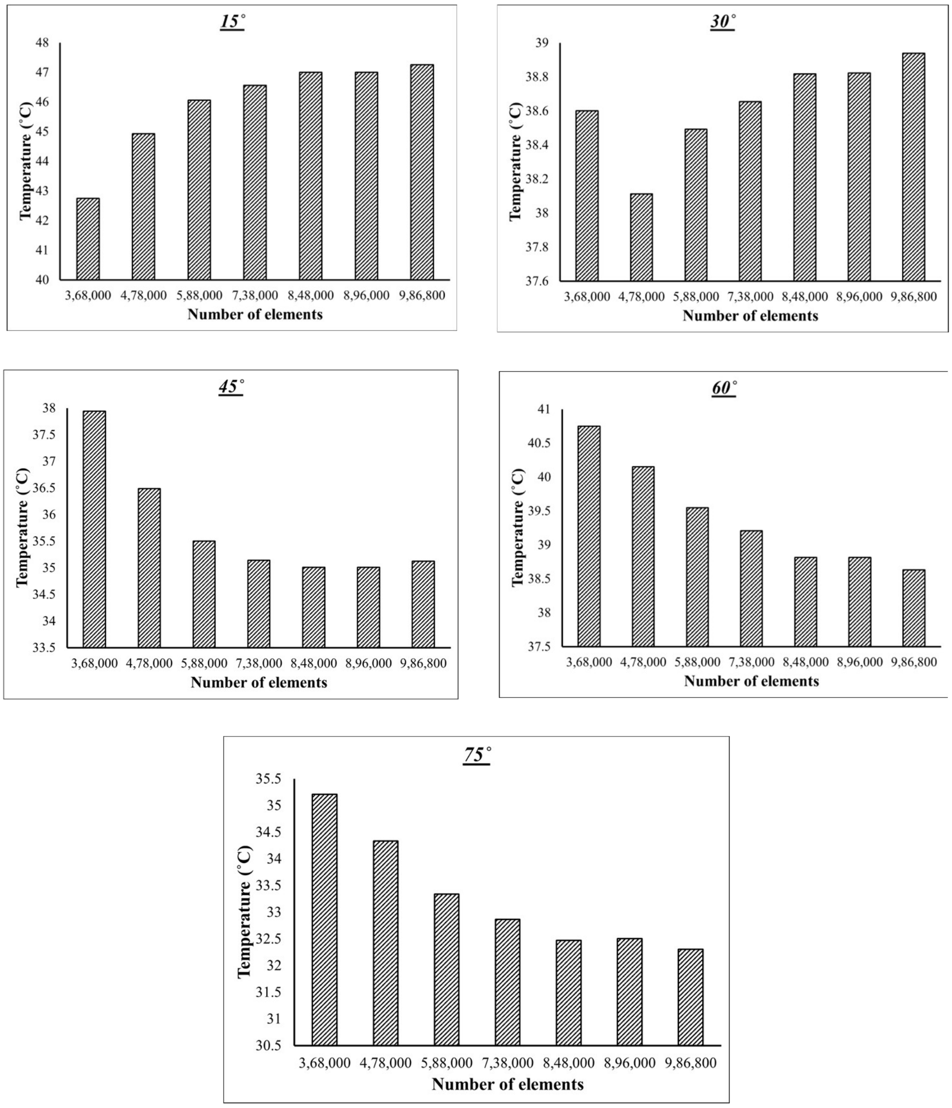


Fig. 4. Grid sensitivity analysis at different angles

4.2 Effect on Temperature Difference

The meshing from the grid sensitivity analysis was exported to ANSYS Fluent module for the numerical simulations. The temperature difference was calculated numerically for all the various Reynolds number (Re) by varying the velocity magnitude. The temperature difference data was noted for a constant heat flux of 336.28 W/m^2. Fig. 5 shows the variation in the temperature difference values with respect to Re for all cases. As shown in the figures, the trend of the line remains the same for all cases of numerical simulation. As shown in figure, the temperature difference decreased almost exponentially with increase in Re. For the case of 15˚, 30˚, 45˚, 60˚ and 75˚, maximum temperature difference of 9.75 ˚C, 9.82 ˚C, 8.2 ˚C, 5.96 ˚C and 9.86 ˚C was noted at Re of 4567, 4567, 5481, 7582 and 4659, respectively. While the minimum temperature difference of 2.52 ˚C, 2.2 ˚C, 1.95 ˚C, 2.05 ˚C and 2.54 ˚C was noted for Re of 17812, 20553, 19639, 22014 and 17812, respectively. As the angular positions changed, the temperature difference values remained almost same for all the values because of the same input parameter of heat flux.

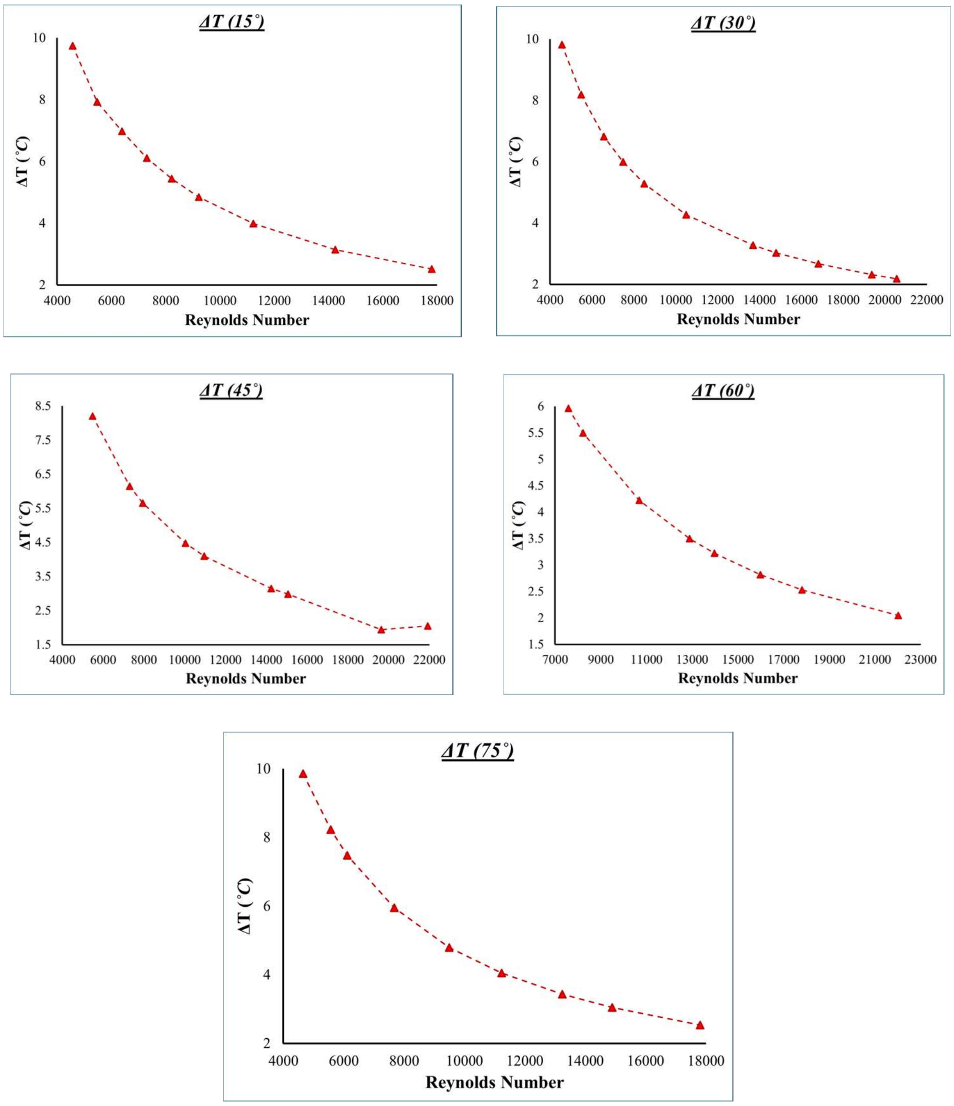
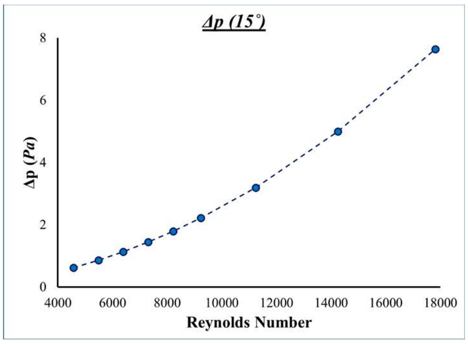
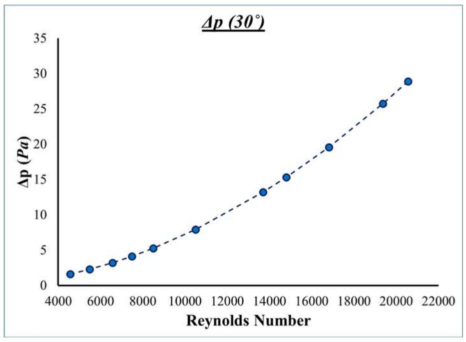


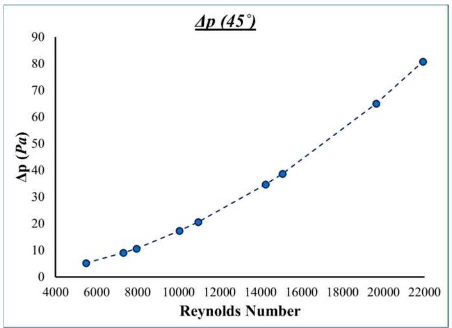
Fig. 5. Temperature difference for varying Re values for different angles

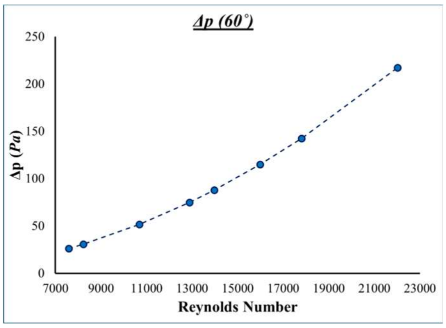
4.3 Effect on Pressure Drop

It is important to note that the improvement in the heat transfer at the cost of pressure drop. More pressure drop requires costlier pumps to drive the flow at the expense of energy. Therefore, it is a need to find a delicate balance between these two parameters. The pressure drop was calculated numerically for all the various Re values. A completely opposite trend was observed as compared to the trend of temperature difference. The pressure drop increased almost exponentially with an increase in Re values [10]. Also, as the angle of the heated circular discs increased, the pressure drop also increased. This result could be utilized to find the trade-off between these two contradicting parameters. For the case of 15˚, 30˚, 45˚, 60˚ and 75˚, maximum pressure drop of 7.64 Pa, 28.93 Pa, 80.74 Pa, 217.07 Pa and 229.961 Pa was noted at Re of 17812, 20553, 19639, 22014 and 17812, respectively. While the minimum pressure drop of 0.623 Pa, 1.61 Pa, 5.22 Pa, 26.11 Pa and 17 Pa was noted for Re of 4567, 4567, 5481, 7582 and 4659, respectively. These observations of temperature difference and pressure drop can be utilized to find the best configuration of the system. To get a particular temperature output from the system, a particular configuration of the heated circular discs can be found by these observations. These will optimize the flow blockage and enhance the system efficiency.









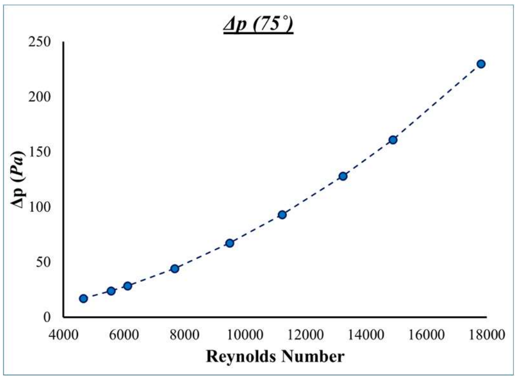
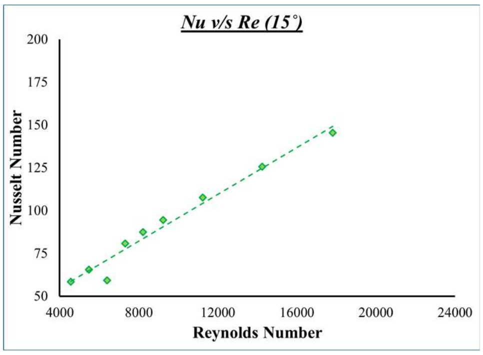
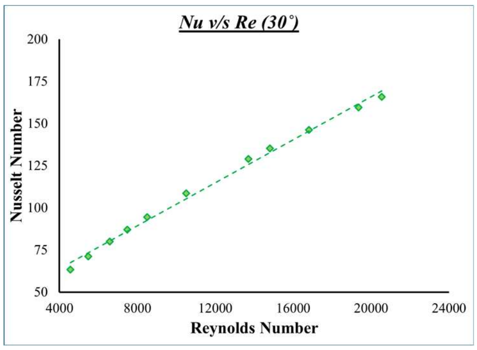


Fig. 6. Pressure drop for varying Re values for different angles

4.4 Thermal Analysis

By collecting all the necessary data, the numerical Nusselt number (Nu) was calculated and plotted against Re values for all the cases as shown in Fig. 7. As shown in the figure, the trend remains the same for all cases, with increase in the Nu number with increased Re number also [11]. Numerical simulations depicted the case of 75˚ as the best case. The turbulence increased as the angle of the discs increased which leads to decreased resident time of the air molecules. This increased resident time decreases the temperature of the discs which ultimately led to increased Nu values in the calculation. Another reason being the bypass flows in different cases of angles. In case of 15˚, the flow slips from the first discs itself which prevents it from taking up heat from the subsequent heater discs. The same thing happened in the case of 75˚. The best-case scenario in terms of turbulence generation would be 60˚ and 45˚. As the flow mixing happened well in both cases with increased turbulence levels.





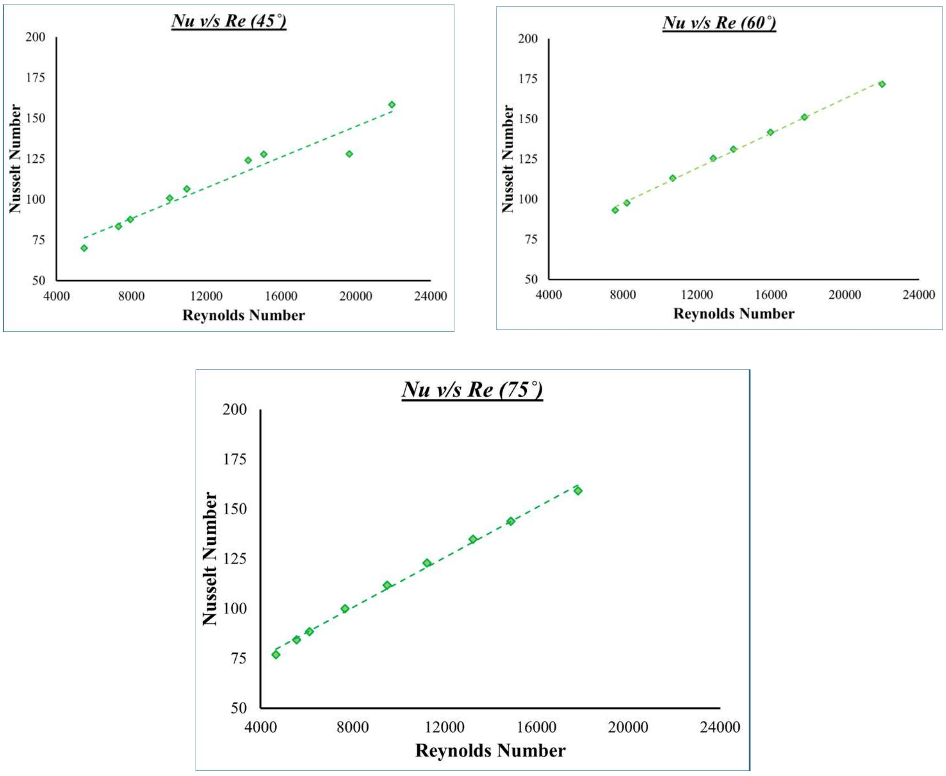


Fig. 7. Nu v/s Re Trends for different angles

4.5 Comparative Analysis

To analyse the performance of the proposed system with respect to the conventional heat transfer techniques, it was compared with the standard correlation available for external or circumferential heating. For fully developed (hydrodynamically and thermally) turbulent flow in a smooth circular tube, the local Nusselt number may be obtained from the well-known Dittus-Boelter equation [12].

$$Nu = 0.023\, Re^{0.8}\, Pr^{0.4}$$

The graphs in Fig. 8 show the comparison of Nu variation with the Re variations with the standard Dittus-Boelter correlation trendlines. It was observed that the proposed technique outperforms the standard technique of external heating. For the case of 15˚, the Nu increase of 187% to 244% was observed compared to the external heating. The same for the case of 30˚, 45˚, 60˚ and 75˚ was observed between 192% to 272%, 134% to 255%, 187% to 265% and 215% to 345%, respectively.

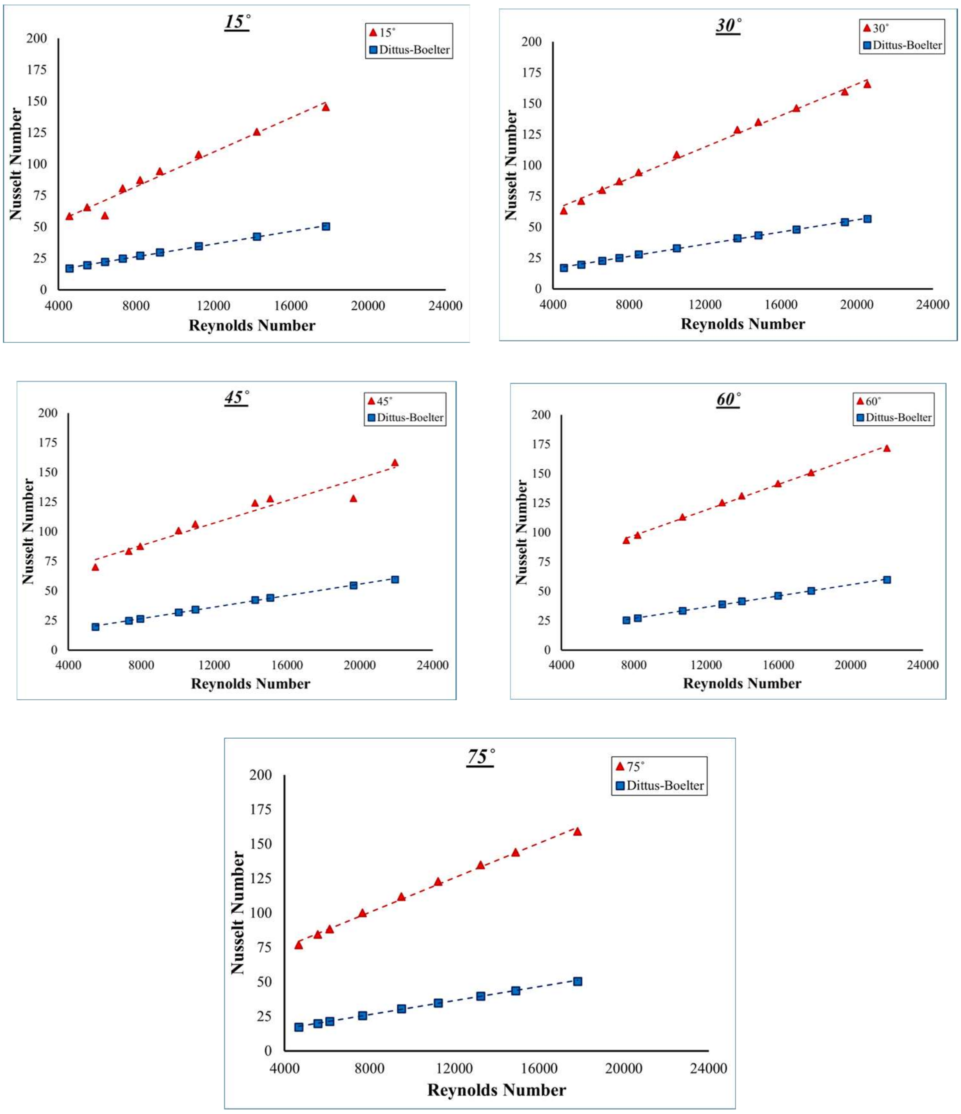


Fig. 8. Comparison of Nu v/s Re Trends for different angles with Dittus-Boelter Correlation

5 Concluding Remarks

The CFD analysis was carried out for different flow configurations at different Re values. A uniform meshing methodology was proposed for all the variations of angles of heated circular discs with respect to the direction of the flow. The grid independent solution was obtained by varying the meshing parameters uniformly. The grid size of 84800 was selected for the performance analysis. The temperature difference values remained almost constant for all the cases considered for the analysis. The pressure drop increased with increase in the angle of heated circular discs. The Nu v/s Re trend can utilize the influence of turbulence on the heat transfer performance of the system. It was concluded that the cases of 45˚ and 60˚ gave considerable heat transfer enhancement for considerable pressure drop. This provides turbulence near the heating surface which increases the thermal performance, and it keeps the fluid dynamics performance almost unaltered. This gives an advantage for the overall heat transfer, where the cost of transferring heat in terms of pressure drop is low. And compared to a basic heat transfer model, that is Dittus-Boelter equation, the obtained heat transfer is significantly high consistently in each case.

References

1. A. Mwesigye, T. Bello-Ochende, J.P. Meyer, Heat transfer and entropy generation in a parabolic trough receiver with wall-detached twisted tape inserts, International Journal of Thermal Sciences 99 (2016) 238–257. https://doi.org/10.1016/j.ijthermalsci.2015.08.015.
2. S. Pourhedayat, S.M. Pesteei, H.E. Ghalinghie, M. Hashemian, M.A. Ashraf, Thermal-exergetic behavior of triangular vortex generators through the cylindrical tubes, International Journal of Heat and Mass Transfer 151 (2020) 119406. https://doi.org/10.1016/j.ijheatmasstransfer.2020.119406.
3. D. Jiang, W. Yang, K.J. Chua, J. Ouyang, J.H. Teng, Analysis of entropy generation distribution in micro-combustors with baffles, International Journal of Hydrogen Energy 39 (2014) 8118–8125. https://doi.org/10.1016/j.ijhydene.2014.03.092.
4. Y. Hong, J. Du, S. Wang, W.-B. Ye, S.-M. Huang, Turbulent thermal-hydraulic and thermodynamic characteristics in a traverse corrugated tube fitted with twin and triple wire coils, International Journal of Heat and Mass Transfer 130 (2019) 483–495. https://doi.org/10.1016/j.ijheatmasstransfer.2018.10.087.
5. W.M. Rohsenow, J.P. Hartnett, Y.I. Cho, eds., Handbook of heat transfer, 3rd ed, McGraw-Hill, New York, 1998.
6. M. Ahadi, A. Abbassi, Entropy generation analysis of laminar forced convection through uniformly heated helical coils considering effects of high length and heat flux and temperature dependence of thermophysical properties, Energy 82 (2015) 322–332. https://doi.org/10.1016/j.energy.2015.01.041.
7. Bamaniya Jayesh Pravinbhai, A.R. Patel, NATIONAL CONFERENCE ON TURBOMACHINES, ENERGY, AND COMBUSTION Performance of the flow of air through a Circular Duct having circular heated disc like obstruction using CFD Analysis, (2021). https://doi.org/10.13140/RG.2.2.12085.60644.
8. J. Bamaniya, M.D. Kevat, A.R. Patel, CFD Analysis of Flow of Air Inside a Circular Duct Around a Series of Vertically Heated Circular Baffled Discs at a Different Angles, in: IIT Ropar,Ropar, Punjab, India, 2022.
9. P. Bamaniya Jayesh, N. Jaimin, K. Mayur D, P. Amit R, Effect of Diameter to Pitch Ratio on Thermal Performance of Air Flow Through A Series of Heated Baffled Discs, in: Proceedings of International Conference RAISE 2023, Faculty Of Technology & Engineering, The Maharaja Sayajirao University Of Baroda, Vadodara, Gujarat, India, 2023.
10. M. Ahishkumar Jashvantlal, P. Lalit B, R. Manish, Numerical Investigation for Thermo-Fluid Analysis on Tube Finned Heat Exchanger using Vortex Generator, in: Institute of Infrastructure Technology Research and Management, Ahmedabad, Gujarat, India, 2019.
11. O. Turgut, E. Kizilirmak, Effects of Reynolds number, baffle angle, and baffle distance on three-dimensional turbulent flow and heat transfer in a circular pipe, Therm Sci 19 (2015) 1633–1648. https://doi.org/10.2298/TSCI121011045T.
12. I. Tosun, EVALUATION OF TRANSFER COEFFICIENTS: ENGINEERING CORRELATIONS, in: Modeling in Transport Phenomena, Elsevier, 2007: pp. 59–115. https://doi.org/10.1016/B978-044453021-9/50005-1.