NUMERICAL INVESTIGATION OF THERMAL HYDRAULIC PERFORMANCE AND PRESSURE DROP CHARACTERISTICS FOR INTERNALLY GROOVED TUBES AT VARIOUS DEPTHS

Chandan B.B[1], Santosh K.S[2], Manjunath G A[3]
[1]M.Tech Student, Thermal Power Engineering, Rajeev Institute of Technology, Hassan, India.
[2]Assistant Professor, Mechanical Engineering, Rajeev Institute of Technology, Hassan, India.
[3]Assistant Professor, Mechanical Department, SJBIT, Bangalore

Abstract: Pipes are widely used in various engineering applications such heat exchangers and oil transport. Plain pipes are not so efficient in the heat exchanging process. Heat transfer enhancement technique such as corrugated pipe is commonly used in the heat exchanging applications. In general these pipes consist of periodically distributed grooves at the inner side of the pipe wall. Along with the heat transfer enhancement corrugation causes more pressure losses than the plain pipe. Aspect ratio and shape of the grove plays a major role in the heat transfer rate enhancement as well as the pressure loss. The purpose of the present study is to investigate numerically the heat transfer rate and pressure loss in a different grove shape pipes with different grove aspect ratio at different Reynolds number to find the better grove shape and aspect ratio.

I. INTRODUCTION

Heat transfer enhancement is important in the development of high performance thermal systems. Many industrial processes involve the transfer of heat energy and most employ old technology. If improved process performance is desired, redesign should be considered using enhanced surfaces. Enhanced heat transfer performance is the result of a combination of structured surface variations that are a result from this detailed surface study. Enhanced performance characteristics include: increased fluid turbulence, secondary fluid flow pattern enhancement, disruption of the thermal boundary layer and increased process surface area. These enhancement factors lead to an increase in the heat transfer coefficient; the ability to produce units with a smaller unit footprint; systems that are more economic to operate; and units with a prolonged life. This development provides a very important and exciting advancement in the design of processes that utilize heat transfer tubes and process surfaces Use of Heat transfer enhancement techniques lead to increase in heat transfer coefficient but at the cost of increase in pressure drop. So, while designing a heat exchanger using any of these techniques, analysis of heat transfer rate and pressure drop has to be done. Apart from this, issues like long term performance & detailed economic analysis of heat exchanger has to be studied. To achieve high heat transfer rate in an existing or new heat exchanger while taking care of the increased pumping power, several techniques have been proposed in recent years and are discussed in the following sections Small scale roughness or surface modification promotes turbulence in the flow field near the wall region by disturbing the viscous laminar sub layer. This disturbance causes higher momentum and heart transfer. This small scale roughness has little effect in laminar flows, but is very effective in turbulent single phase flows. Nowadays instead of natural roughness, artificial and structured roughness is used in most applications. Structured roughness can be integral to the surface gra et al. (2011) numerically investigated the heat transfer and pressure drop characteristics of smooth, corrugated and helically finned tubes using CFD analysis. Analysis was carried out for Reynolds Numbers ranging from 12,000 to 57,000. Water was used as a fluid and a constant temperature boundary condition was applied to the tube wall. Effects of the helix angle and ridge height on: the temperature difference between the fluid and the tube wall; magnitude of the pressure drop; and the convective heat transfer coefficient were investigated as a function of temperature, mass flux and Reynolds Number. Kumar et al. (2012) presents a summary of several CFD investigations detailing the effect of roughness geometries on the heat transfer and friction factor for heating ducts used in solar applications In general the objective of this study is to optimize the surface pattern on an enhanced heat transfer surface in order to maximize heat transfer while minimizing pumping power, for the conditions considered, the new enhanced surface was then compared to an unenhanced surface using CFD analysis

II. MATHEMATICAL MODEL

Realizable k-ε Model; The Realizable k-ε Model differs from the standard k-ε model in two important ways, The realizable k-ε model contains an alternative formulation for the turbulent viscosity A modified transport equation for the dissipation rate ε, has been derived from an exact equation for the transport of the mean-square vorticity fluctuation. The term 'realizable' means that the model satisfies certain mathematical constraints on the Reynolds stresses, consistent with the physics of turbulent flows. Neither the standard k-ε model nor the RNG k-ε model is realizable. To understand the mathematics behind the realizable k-ε model, consider combining the Boussinesq relationship and the eddy viscosity definition to obtain the following expression for the normal Reynolds stress in an incompressible strained mean flow

$$\overline{u^2} = \frac{2}{3}k - 2v_t\frac{\partial U}{\partial x} \quad \text{(1)}$$

We know that for $v_t = \mu_t/\rho$,, one obtains the result that the normal stress, $\overline{u^2}$ which by definition is a positive quantity, becomes negative, that is, "non-realizable", when the strain is large enough to satisfy

$$\frac{k}{\varepsilon}\frac{\partial U}{\partial x} > \frac{1}{3C_\mu} = 3.7 \quad \text{(2)}$$

Similarly, it can also be shown that the Schwarz inequality for shear stresses $\overline{u_\alpha}\,\overline{u_\beta}^2 \leq \overline{u_\alpha{}^2 u_\beta{}^2}$

No summation over α and β can be violated when the mean strain rate is large. The most straightforward way to ensure the realizability (positivity of normal stresses and Schwarz inequality for shear stresses) is to make C_μ variable by sensitizing it to the mean flow (mean deformation) and the turbulence k-ε. The notion of variable C_μ is suggested by many modellers including Reynolds, and is well substantiated by experimental evidence. For example, C_μ is found to be around 0.09 in the logarithmic layer of equilibrium boundary layers, and 0.05 in a strong homogeneous shear flow. Both the realizable and RNG k-ε models have shown substantial improvements over the standard k-ε model where the flow features include strong streamline curvature, vortices, and rotation. Since the model is still relatively new, it is not clear in exactly which instances the realizable k-ε model consistently outperforms the RNG model. However, initial studies have shown that the realizable model provides the best performance of all the k-ε model versions for several validations of separated flows and flows with complex secondary flow features. One of the weaknesses of the standard k-ε model or other traditional k-ε models lies with the modelled equation for the dissipation rate 'ε'. The well-known round-jet anomaly (named based on the finding that the spreading rate in planar jets is predicted reasonably well, but prediction of the spreading rate for axisymmetric jets is unexpectedly poor) is considered to be mainly due to the modelled dissipation equation. One limitation of the realizable k-ε model is that it produces non-physical turbulent viscosities in situations when the computational domain contains both rotating and stationary fluid zones (for example, multiple reference frames, rotating sliding meshes). This is due to the fact that the realizable k-ε model includes the effects of mean rotation in the definition of the turbulent viscosity, this extra rotation effect has been tested on single moving reference frame systems and showed superior behaviour over the standard k-ε model. However, due to the nature of this modification, its application to multiple reference frame systems should be taken with some caution.

III. SOLUTION METHODOLOGY

Geometry and grid generation: in the present work, three different configurations are assumed for the corrugated pipe,(rectangular, semicircular, trapezoidal) involving different grooves depth and grove length is maintained constant, both geometry and grid is generated using same Ansys workbench to avoid unpredictable errors, Corrugated pipes are cylindrical ducts with periodically distributed grooves at the wall, and can be solved as a single corrugated domain subject to a periodic fully developed flow, as described by Patankar et al. (1977).

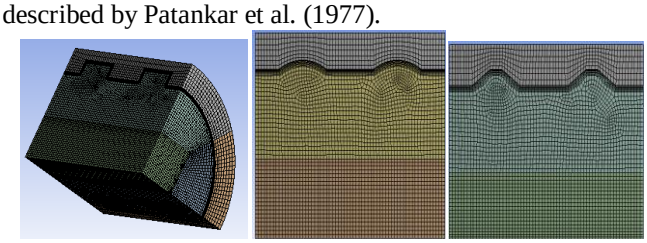

Figure 1. Grid generation of grooves (a) Rectangular 3mm depth (b) semicircular 1mm depth (c) trapezoidal 2mm depth

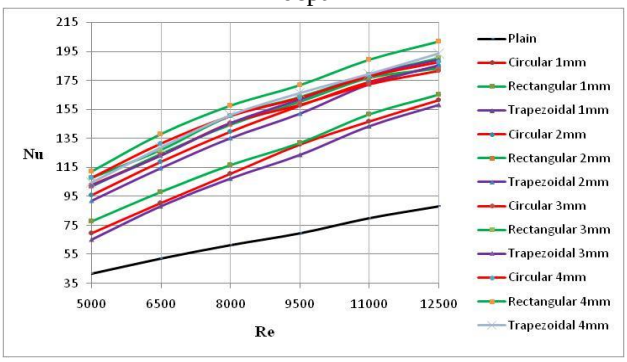

Figure 2. Variation of Reynolds number with Nusselt number for all grooves and all depths and plain pipe

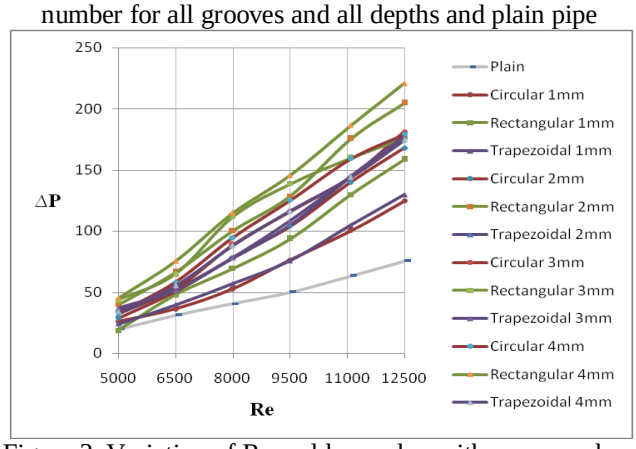

Figure 3. Variation of Reynolds number with pressure drop for all grooves and all depths and plain pipe

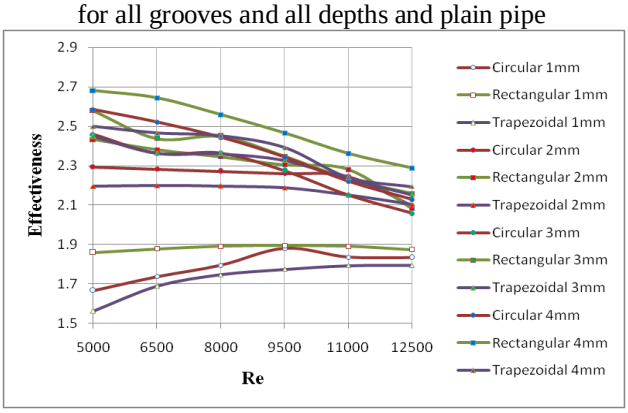

Figure 4. Variation of Reynolds number with Effectiveness for all grooves and all depths

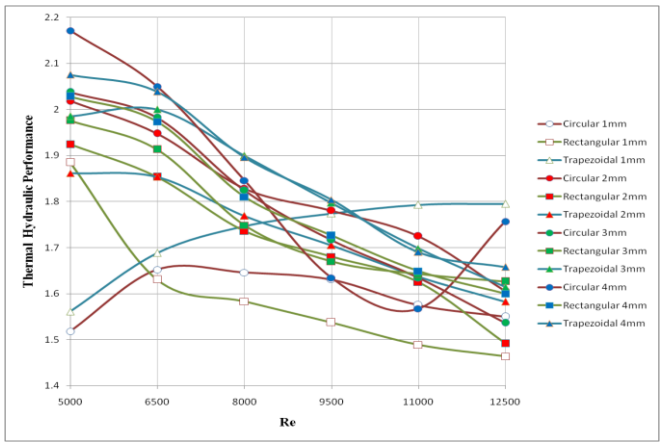

Figure 5. Variation of Reynolds number with Thermal Hydraulic Performance for all grooves and all depths

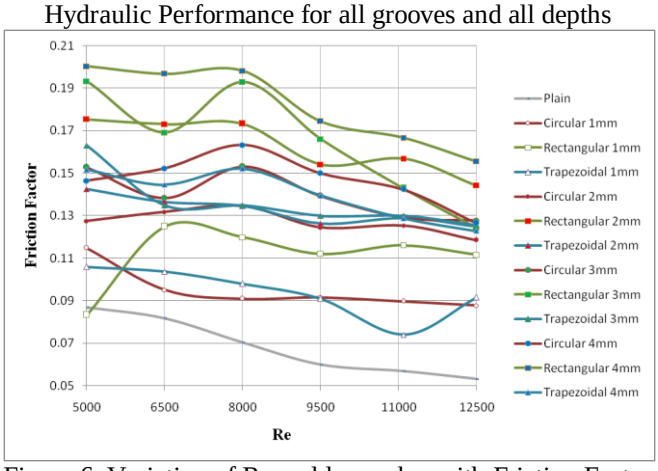

Figure 6. Variation of Reynolds number with Friction Factor for all grooves and all depths

Figure 1 shows a schematic representation of a grooved pipes, grooved pipes are cylindrical ducts with periodically distributed grooves at the wall, and can be solved as a single corrugated domain subject to a periodically fully developed flow, Computational mesh was created by using special method of meshing called sweep method, and inflation is done for required walls to catch the viscous and pressure drop fluctuations at the wall, all groves are structural grids, several cell size were tested synthetically by considering simulation time needed for specific number of iteration and the accuracy of the solution. Because the local Reynolds number near any wall becomes very small and owing to viscous influence, to overcome this issue, a special a near wall modelling approach applied to possess the accuracy of the standard boundary layer approach for fine near -wall mesh, so the first cell is placed in the turbulent boundary layer, though the laminar sub layer is valid.

IV. SOLUTION METHODOLOGY

Fully developed turbulent flow was imposed at the inlet of the all grooved channels, flow motion and energy equations solved numerically by using CFD (computational fluid dynamics) consider different parameters such as Reynolds number, groove depth and groove length, the channel surface consider under constant wall heat flux, the overall performance of the evaluated in terms of pressure drop and average Nusselt number which calculated numerically. The solution domain is considered 3D, enclosed by outlet inlet and wall boundaries. No-slip conditions are assumed for momentum equations on walls. The inlet velocity values have been derived from given Reynolds numbers. To save the computational time; the periodicity code was used to define hydrodynamics fully developed flow at inlet, the outlet boundary condition is called "pressure outlet", which implies a static (gauge) pressure at the outlet boundary, which means the pressure will be extrapolated from the flow in the interior.

V. NUMERICAL SIMULATIONS

The numerical results obtained for the three different grooves proposed for a depth of 4mm, 3mm, 2mm, 1mm and the plain pipe as well as relevant physical interpretations related to the observed turbulent flow pattern. The fluid properties assumed are density $\rho = 998.2$ kg/m3 and cinematic viscosity $\nu = 1.003\times10^{-6}$ m2 /s, and the Reynolds numbers 5000, 6500, 8000, 9500, 11000 and 12500 are simulated

VI. RESULTS AND DISCUSSION

The change in temperature and pressure drop across the test tube at different inlet mass flow rates for the plain and various grooved geometry tubes were obtained from the numerical modelling. However, in general the pressure drop, had increased with any of the grooved tubes were used. In addition, the results show that for all fluid velocities the highest temperature obtained when the trapezoidal grooved tube was used although it caused more pressure drop. The whole results show that increased pressure drop and increased temperature ensures the increased heat transfer rate In addition, the effect of groove depth and Reynolds number are more significant on enhancement of the Nusselt number, high values of average Nusselt number in small hydraulic diameter grooved pipe are related to strong interaction between the flow inside the channel and over the corrugation crest, accompanied by the secondary flows [5 and 6], as (1) increasing advection and turbulence of fluid from the centre of the channel to the near- wall region; (2) they are responsible for the breakup and separation of the boundary layer and its new formation and reattachment [3]; [4] they decrease the probability of appearance of stagnation areas [8]. The variation of Nusselt number with Reynolds Number for the tubes is shown in Fig. 2 and it illustrates that as Reynolds number increases Nusselt number is also increases. Generally, the test in each plot has nearly the same Prandtl number. The increase in Nusselt number indicates an enhancement in heat transfer co-efficient due to increase of convection heat transfer. The overall results show that when the tube with internal grooves, the Nusselt numbers is higher than those obtained for the plain tube. Furthermore, the results show that the calculated Nusselt number for rectangular grooved tube was higher than that obtained for others at all examined Reynolds numbers

Effect of grooved configuration: Effect of groove depth (1mm,2mm,3mm,4mm) on heat transfer performance for

three different corrugated surface configuration are shown in Figures 2,3,4,5 and 6. The predicated Values of average surface Nusselt number was plotted with various Reynolds number, these figures shown that the values of average surface Nusselt number are increase with decreasing the groove depth and increase with increase the values of Reynolds number. Variation of pressure drop per unit length ($\Delta P/L$) with Reynolds number for different groove depths and groove geometries are shown in Fig. 3 In general, characteristics of the flow through corrugated surfaces channel is quite complex with comparison with smooth surface channel [4]. Higher pressure drop in corrugated channel is generated because corrugated surfaces (1) produce rotational flow, (2) exerted drag force on the flow field which leads to turbulence augmentation. Based on the discussion above, as expected, the pressure drop in corrugated channel are higher compared with ones for smooth surface channel as shown in Fig. 3 It is clear that the pressure drop tends to increase with increase in Reynolds number and decrease with increasing the groove depth By using grooved surface with different geometries, it causes enhanced heat transfer because it promise better fluid mixing and introducing turbulence but all of that with pressure drop will be the penalty, and based on the previous studies, it is common that the pressure drop penalty often higher than the values of heat transfer enhancement[30]. The calculation results for the thermal enhancement ratio for different corrugated surface with different groove geometries and groove depths illustrated in Fig. 5 It is clear from these figures that, the thermal enhancement ratio increase linearly with increasing Reynolds number also it is obvious that trapezoidal grooved surface has the better thermal performance compared with rectangular and semi-circular grooved surface, because in case of trapezoidal grooves fluid can be easily sweepable along the crest. As shown in Figures, it is clear that, the effect of groove geometries on the temperature and flow development in the grooved channel, that there are growth of swirl flow near to corrugated wall which promote the mixing of the main stream with the hot fluid near to the wall, the swirl and turbulence intensity change with changing corrugated geometries, recirculation region and more separation bubble regions formed for trapezoidal and rectangular grooves rather than semi-circular grooves, which means that the influence of the wall on the main stream becomes greater; thus generated more swirl flow in the wavy wall due to transfer vortices of the bulk flow field in the wavy wall trough, these result induced higher temperature gradient near the wavy wall. Therefore, the net heat transfer rate from the wavy wall to the fluid as shown in Fig (19.B).

VII. CONCLUSION

In this study different geometry grooved tube like circular, square and trapezoidal were introduced for this analysis. Numerical simulation results and graphs shows that in terms of heat transfer enhancement all the groves are working better than the plain pipe. But total pressure loss was increased drastically due to disturbance caused by these grooves. To find the optimal grove further numerical studies were carried out by changing the aspect ratio of the grooves. These study shows that higher Nusselt number and pressure loss were obtained for the 4mm depth rectangular slot grooved tube compared with all other grooved tubes in the studied range of Reynolds number. Aspect ratio study with different grove shapes also show that reduction in total pressure loss is significant only at 1mm depth with drastic reduction in the Nusselt number. Remaining 2mm and 3mm depths are causing lesser pressure loss reduction compared with trapezoidal and circular groves 4mm depth slot. Under the studied range of Reynolds number 4mm depth trapezoidal groove is better than all other cases. Because it is causing less reduction in Nusselt number and significant reduction in total pressure loss compared with 4mm depth rectangular slot.

REFERENCES

[1] Naphon P. Study on the heat-transfer characteristics and pressure drop in channels with arc-shaped wavy plates. Journal of Engineering Physics and Thermo physics November 2010; 3:1061-69.
[2] Sui Y, Teo CJ, Lee PS, Chew YT, Shu C. Fluid flow and heat transfer in wavy microchannels. Int J Heat Mass Transfer 2010; 53:2760-72.
[3] Naphon P. Effect of wavy plate geometry configurations on the temperature and flow distributions. IntCommun Heat Mass Transfer 2009; 36:942-6.
[4] Elshafei EAM, Awad M, El-Negiry E, Ali AG. Heat transfer and pressure loss in narrow channels with corrugated walls. Thermal Issues in Emerging Technologies, 2008 ThETA '08 Second International Conference on 2008:279-90.
[5] Heggs PJ, Sandham P, Hallam RA, Walton C. Local transfer coefficients in corrugated plate heat exchangers channels, Trans IChemE, Part A. ChemEng Res Des 1997;75(A7):641–5.
[6] Dovic D, Palm B, Svaic S. Generalized correlations for predicting heat transfer and pressure drop in plate heat exchanger channels of arbitrary geometry. Int J Heat Mass Trans 2009;52(19–20):4553–63.
[7] Shah RK, Sekuli´c DP. Fundamentals of heat exchanger design. New York: John Wiley & Sons; 2003. pp. 9–39.
[8] Li Q, Flamant G, Yuan X, Neveu P, Luo L. Compact heat exchangers: A review and future applications for a new generation of high temperature solar receivers. Renewable and Sustainable Energy Reviews 2011; 15:4855-75.

