Review on CFD Investigation of Bend Angle in Pipe Flow

A.Jothivel,
Department of Mechanical Engineering,
Sri Krishna College Of Technology,
Coimbatore, India.

Mr.L.Sankar, M.E.
Department of Mechanical Engineering,
Sri Krishna College Of Technology,
Coimbatore, India.

Abstract:- **In present paper, pipes with different bend angles are designed and simulated using computational fluid dynamic software for water with different temperatures. Pipe will be considered as real wall and its temperature distribution is measured to corresponding temperature boundary condition at inlet. Ultimately, influence of bend angle over heat transfer enhancement in pipes will explored computationally and experimentally.**

Keywords: Pipe bend, angle, temperature, CFD

I. INTRODUCTION

Heat transfer enhancement techniques are developed vigorously in heat transfer applications. Induced turbulence in heat exchanger at shell side is one of the heat transfer enhancement technique. If we generate turbulence and cross flow in flow path of fluid by baffles, heat transfer rate is increased. In many literatures, effects of bend over flow properties of pipe flow are investigated. There is very few such kind of literature over pipe bend effect in heat transfer rate because bend causes turbulence and reduces energy of fluid. Turbulence is undesirable in case of flow properties like pressure and velocity but favourable to heat transfer enhancement. Therefore, it is required to investigate effects pipe bend angle over heat transfer rate is important.

In this present work, pipes with different bend angles are designed and simulated using computational fluid dynamic software for water with different temperatures. Pipe will be considered as real wall and its temperature distribution is measured to corresponding temperature boundary condition at inlet. Ultimately, influence of bend angle over heat transfer enhancement in pipes will explored computationally and experimentally.

II. LITERATURE REVIEW

Ahmed F. Khudheyer et. al (2014) presents numerical analysis of fin-tube plate heat exchanger by using CFD technique. The objective of this study was to make CFD simulations using open source software, and validate the results against experimental data. The system to study was a fin-and-tube heat exchanger. The purpose of the work was to investigate the possibilities of eventually using CFD calculations for design of heat exchangers instead of expensive experimental testing and prototype production. To analyse the flow and heat transfer characteristics of the heat exchanger, a model of a two-row fin-and-tube heat exchanger was created using open source Salomé software to create the geometry and mesh. The resulting mesh (after a grid independence test was carried out) was used for running a variety of simulations using a laminar flow model and two turbulence models for comparison of results. Ten different inlet flow velocities ranging from 0.3 m/s to 6.2 m/s and corresponding to Reynolds numbers ranging from 330 to 7000 were simulated in the three different flow models (laminar, k-epsilon turbulence model, and SST k-omega turbulence model). A sampling dictionary was written into the CFD model to record pressure and temperature measurements at the inlet and outlet of the heat exchanger model. Using the simulation results and some specific non-dimensional numbers, calculations related to heat flow and pressure loss can be carried out to determine the Fanning friction factor and Colburn j-factor for comparison with the literature values used for the validation [1].

Arafat A. Bhuiyan et. al (2012) presented numerical study of 3d thermal and hydraulic characteristics of wavy fin and tube heat exchanger. Numerical visualisations are used to study the thermal and hydraulic performance of four row wavy staggered fin and tube heat exchanger. In this paper, the effects of tube arrangements, different geometrical parameters and inlet flow angles are investigated in terms of heat transfer and pressure drop and efficiency for the wavy fin-and-tube heat exchanger for turbulent flow regime using k-ω turbulence model with 5% turbulence intensity. The tube arrangement and the geometrical parameters such as pitch, wavy angle and inlet flow angle have strongly affected the flow structure. Comparatively higher heat transfer and pressure drop is found in staggered arrangement than in lined for both laminar and turbulent case. By increasing Ll and Lt, f and j both decreases as the flow becomes free and less compact. But efficiency goes high. The fin spacing very strongly influences the heat transfer and pressure drop. If it is too small, the effects are less; if it is too large, the effect is comparatively higher. Higher pressure drop and heat transfer are observed for more inclination in a given flow length, but efficiency decrease with the increase in the wavy angle. With the increase in positive flow angle, the effects in the heat transfer and pressure drop is decreased. Again, with the increase in negative flow angle, the effect in those criteria is same. But the decrease for the negative flow angle is higher compared to the positive flow angle [2].

A. K. Biswas et. al (2012) presents flow investigation in a constant area curved duct. The present study dealt with the wall static pressure of a rectangular curved duct with angle of turn 90° at all four faces with various locations. The normalized wall static pressure distributions at the top, bottom, inside and outside surfaces of the C-duct are drawn in the form of contours at three different air velocities, Uav = 20m/s, 40m/s and 60m/s respectively. These iso-bar contours indicate towards the presence of secondary flow in the duct in the form of a pair of contra-rotating vortices for all the three air velocities. Keywords: Curved Duct, Secondary Motion, Wall pressure, Wind Tunnel [3].

Jignesh M. Chaudhari et. al (2014) presented a complete set of numerical parameter studies on automobile radiator has been presented in detail in this paper. The calculation has been carried out by well verified and validated rating. The corresponding mathematical formulation has been briefly described within the paper. A first part of the parametric studies has been focused on the influence of working condition on fluid (mass flow rate, input parameters). In Cross flow heat exchanger experiment setup is a useful tool for analyzing different parameters of finned tube heat exchanger and heat loss related testing, it will provide information on the variables heat effects on water temperature. The model will help characterize heat exchanger performance and provide a basis for assessing current temperature controlling condition. Classification and Design details helps to understand the working and industrial application of heat exchangers. As shown in graph we conclude that the Overall heat transfer rate of finned tube heat exchanger is greater than without finned tube heat exchanger. Now as we increases the air velocity heat transfer rate of finned tube heat exchanger is increases because the Reynolds number is increases the Nusselt number is also increases because Nusselt number is directly proportional to the heat transfer coefficient. So, heat transfer rate is increases [4].

Jogi Nikhil G. et. al (2012) presents heat transfer analysis of corrugated plate heat exchanger of different plate geometry. Corrugated plate heat exchangers have larger heat transfer surface area and increased turbulence level due to the corrugations. In this study, experimental heat transfer data will obtain for single phase flow (water-to-water) configurations in a corrugated plate heat exchanger for symmetric 45°/45°, 45°/75° chevron angle plates. The effect of variation of chevron angles with other geometric parameter on the heat transfer coefficient will be study. Reynolds number ranging from 500 to 2500 and Prandtl number ranging from 3.5 to 6.5 will be taken for given experiment. Based on the experimental data, a correlation will estimate for Nusselt number as a function of Reynolds number, Prandtl number and chevron angle [5].

I.P. Kandylas et. al (1999) presented engine exhaust system design based on heat transfer computation. The heat transfer conditions in automotive exhaust piping are only recently being studied in depth because of their important role in the design and optimization phases of exhaust after-treatment systems. The complex geometry of the exhaust line and the special flow conditions complicate the problem of accurately estimating several important heat transfer parameters. This paper initially summarizes the current status of knowledge regarding heat transfer phenomena in automotive exhaust systems. Experimental data from steady state and transient heat transfer measurements in automotive exhaust systems are presented and analyzed by means of a comprehensive transient computer model covering all exhaust piping conjurations (single wall, double wall with air gap or insulation) already presented elsewhere. Examples are presented, illustrating the application of the model in the comparative assessment of different exhaust conjurations. In conjunction with existing models which simulate the operation of three-way catalytic converters and of other exhaust gas after-treatment devices, the model is already integrated in a CAE package for the support of exhaust system design optimization [6].

G. Kumaresan et. al (2012) presents a review on heat transfer enhancement studies of heat pipes using Nano fluids. The review reports the use of conventional fluids and different Nano fluids with varying mass/volume fractions in heat pipes. Nanoparticles like Ag, Au, Cu, CuO and Al2O3 were dispersed in various base fluids. Replacing the conventional fluid by nano-fluid reduces the dry out problems and enhances the heat transfer capacity. Improvement in thermal efficiency and reduction in thermal resistance is witnessed with increasing mass/volume fraction of nanoparticles in base fluids. Orientation of the heat pipe also affects its thermal performance. The optimum performance is obtained at a tilt angle of around 60° for wick heat pipes and vertical position for thermo syphon heat pipes. Size of nanoparticles and its concentration has a strong influence on the temperature distribution. Effect of heat pipe geometry and its impact on heat transfer characteristics could be explored in future. Uses of hybrid Nano fluids in heat pipes have also been deliberated [7].

Kumbhar et. al (2010) presented a twisted tape insert mixes the bulk flow well and therefore performs better in laminar flow, because in laminar flow the thermal resistant is not limited to a thin region. The result also shows twisted tape insert is more effective, if no pressure drop penalty is considered. Twisted tape in turbulent flow is effective up to a certain Reynolds number range. It is also concluded that twisted tape insert is not effective in turbulent flow, because it blocks the flow and therefore pressure drop increases. Hence the thermo hydraulic performance of a twisted tape is not good in turbulent flow. These conclusions are very useful for the application of heat transfer enhancement in heat exchanger networks [8].

Paisarn Naphon et. al (2006) presents a review of flow and heat transfer characteristics in curved tubes. In addition, only some papers have presented the effects of the combined active and passive method on the enhancement of heat transfer rate and pressure drop. The study points out that although numerous studies have been conducted on the characteristics of heat transfer and flow in curved tubes, study on some types of curved tubes is limited, especially on spirally coiled tubes. The performance of heat exchangers can be improved to perform a certain heat-transfer duty by heat transfer enhancement techniques. In general, these techniques can be divided into two groups: active and passive techniques. The active techniques require external forces, e.g. electric field, acoustic or surface vibration, etc. The passive techniques require fluid additives or special surface geometries. Curved tubes have been used as one of the passive heat transfer enhancement techniques and are the most widely used tubes in several heat transfer applications. This article provides a literature review on heat transfer and flow characteristics of single-phase and two-phase flow in curved tubes. Three main categories of curved tubes; helically coiled tubes, spirally coiled tubes, and other coiled tubes, are described. A review of published relevant correlations of single-phase heat transfer coefficients and single-phase, two-phase friction factors are presented [9].

K.S.Parmar et. al (2010) said that heat exchangers are popular used in industrial and engineering applications. The design procedure of heat exchangers is quite complicated, as it needs exact analysis of heat transfer rate, efficiency and pressure drop apart from issues such as long-term performance and the economic aspect of the equipment. Whenever inserts technologies are used for the heat transfer enhancement, along with the improvement in the heat transfer rate, the pressure drop also increases, which induces the higher pumping cost. Therefore any augmentation device or methods utilized into the heat exchanger should be optimized between the benefits of heat transfer coefficient and the higher pumping cost owing to the increased frictional losses. So, if we provided fins rectangular or circular type on internal periphery of the tube type heat exchanger than the heat transfer rate will go to be increased [10].

Quamrul H. Mazumder (2012) presented effect of liquid and gas velocities on magnitude and location of maximum erosion in u-bend. Solid particle erosion is a micromechanical process that is influenced by flow geometry, material of the impacting surface, impact angle, particle size and shape, particle velocity, flow condition and fluid properties. Among the various factors, particle size and velocity have been considered to be the most important parameters that cause erosion. Particle size and velocity are influenced by surrounding flow velocities and carrying fluid properties. Higher erosion rates have been observed in gas-solid flow in geometries where the flow direction changes rapidly, such as elbows, tees, valves, etc, due to local turbulence and unsteady flow behaviours. This paper presents the results of a Computational fluid dynamic (CFD) simulation of dilute gas-solid flow through a U-Bend and the dynamics behaviour of entrained solid particles in the flow. The effect of liquid and gas velocities on location of erosion were investigated for 50, 100, 150, 200, 250 and 300 microns sand particles. Three different fluid velocities of 15, 30.48 and 45 m/s were used in the CFD analysis. The magnitude and location of erosion presented in the paper can be used to determine the areas susceptible to maximum erosive wear in elbows and U-bends, along with corresponding rate of metal loss in these areas [11].

Shinde Digvijay D et. al (2009) presented heat transfer analysis of a cone shaped helical coil heat exchanger. In this work, the experimental evaluation of cone shaped helical coil heat exchanger is carried out. The overall conclusions related to the comparative analysis between the cones shaped coil & simple helical coil are presented. It is found that the inner Nusselt number, Convective heat transfer coefficient and overall heat transfer coefficient increases when the coil side fluid flow rate increases. From comparative experimental analysis for the conical coil & simple helical coil it is found that the inner Nusselt number, convective heat transfer coefficient & overall heat transfer coefficient are higher in case of conical coil than that of simple helical coil. From comparative experimental analysis, it is found that the effectiveness of heat exchanger is on higher side in conical coil than that of simple helical coil. It was found that the heat transfer rates are 1.18 to 1.38 times more for the cone shaped helical coil than that of simple helical coil [12].

N. D. Shirgire et. al (2013) presents that comparative study is carried out between helical coil heat exchanger and straight tube heat exchanger, The effectiveness of heat exchanger greatly affected by hot water mass flow rate and cold water flow rate. When cold water mass flow rate is constant and hot water mass flow rate increased the effectiveness decreases. Increase in cold water mass flow rate for constant hot water mass flow rate resulted in increase in effectiveness. For both helical coil and straight tube heat exchangers with parallel and counter flow configuration this result obtained. Helical coil counter flow is most effective in all these conditions and straight tube parallel flow heat exchanger is least effective. Overall heat transfer coefficient on other hand increases with increase in hot water mass flow rate and cold water mass flow rate. The highest overall heat transfer coefficient is noted for cold water mass flow rate 100 LPH and hot water mass flow rate 100 LPH, in helical coil counter flow. Use of a helical coil heat exchanger was seen to increase the heat transfer coefficient compared to a similarly dimensioned straight tube heat exchanger [13].

Shivalingaswamy B.P et. al (2014) presented computational heat transfer flow modelling is one of the great challenges in the classical sciences. As with most problems in engineering, the interest in the heat transfer augmentation is increasing due to its extreme importance in various industrial applications. CFD modelling for the heat transfer augmentation in a circular tube fitted with and without rod circular inserts in turbulent flow conditions has been explained in this paper using ANSYS Fluent version 14.0. This paper presents the effect of the circular-ring tabulator (CRT) on the heat transfer and fluid friction characteristics in a heat exchanger tube. The experiments were conducted by insertion of CRTs with various geometries, including three different diameter ratios (DR=d/D=0.5, 0.6 and 0.7) and two different pitch ratios (PR=p/D=4, 8). During the CFD simulation air at 27 °C was passed through the test tube which was controlled under uniform wall heat flux condition. The Reynolds number was varied from 4000 to 20,000. According to the experimental results, heat transfer rates in the tube fitted with CRTs are augmented around 57% to 195% compared to that in the plain tube, depending upon operating conditions. In addition, the results also reveal the CRT with the smallest pitch and diameter ratios offers the highest heat transfer rate in accompany with the largest pressure loss [14].

Suhas V. Patil et. al (2011) presented heat transfer augmentation in a circular tube and square duct fitted with swirl flow generators. A twisted tape insert mixes the bulk flow well and therefore performs better in laminar flow, because in laminar flow the thermal resistant is not limited to a thin region. The result also shows twisted tape insert is more effective, if no pressure drop penalty is considered. Twisted tape in turbulent flow is effective up to a certain Reynolds number range. It is also concluded that twisted tape insert is not effective in turbulent flow, because it blocks the flow and therefore pressure drop increases. Hence the thermo-hydraulic performance of a twisted tape is not good in turbulent flow. These conclusions are very useful for the application of heat transfer enhancement in heat exchanger networks. This helical screw tape can help to promote higher heat transfer exchange rate than the use of twisted-tape because of shorter pitch length which leads to stronger swirling flow and longer residence time in the tube. Because of lower pressure drop and ease of manufacturing, the twisted-tape is, in general, more popular than the helical screw-tape having a higher heat transfer rate at the same mass flow rate. However, at low values of Reynolds number the pressure drops for using both tapes are not much different. Heat transfer of square tubes was found considerably higher than the circular tube. This is mainly because of square duct has high surface to volume ratio. The combined use of full-length twisted-tape and transverse ribs enhances the thermo-hydraulic performance of the square and rectangular ducts compared to the use of only twisted-tape or only transverse ribs for laminar flow. The short-length twisted tape in square and rectangular ducts performs worse than the full-length twisted tape. However, regularly spaced twisted-tapes perform significantly better than the full-length twisted tapes [15].

XueHong Wu et. al (2011) presented numerical study on the effect of tube rows on the heat transfer characteristic of dimpled fin. The paper presents the effects of tube rows on the flow structure and heat transfer performance. The investigated results show that the fluid across the dimple concave can form a flow structure like a ponytail streamlines, which can enhance the heat transfer performance. the average Nusselt number ,Q/p and COP decrease with the increase of the number of tube rows. At the same number of tube rows, they decrease as the velocity increases. The value of Q/p and COP are the largest at two-row tube. Compared with the three, four, five rows tube, the average Nusselt number of two rows tube increases 8.39%, 20.4%, 20.53% at Re=3415, respectively. The JF factor of the two-row tube is 17%, 27%, 29% bigger than the value of 3, 4, 5-row tube, respectively, so the comprehensive performance of the two rows tube is bette [16].

III. PROBLEM DESCRIPTION

Requirement of heat transfer enhancement in heat transfer applications is increased to bring efficient process or products where ever necessary. Turbulence is induced in shell side of S & T exchanger by placing baffles to get more heat transfer rate. Some of the literatures speak about turbulence in U-bend of tubes in S & T exchanger and heat transfer enhancement. It is required to know relationship between pipes bend angle and heat transfer rate (indirectly turbulence) for future implementation in S & T exchanger tubes.

IV. PROPOSED SOLUTION

In present work, computational fluid dynamics is used to explore the relationship mentioned above. Straight pipe is the reference. From that, pipes are designed with different bend angles (25°- 125°). Water is working fluid. Inlet temperatures vary from (20-75°C). Inlet velocities are 5m/s and 10m/s. Outlet temperature is found using CFD. Based on turbulence at pipe bend, heat transfer varies to pipe material, correspondingly outlet temperature varies. From this, we can explore relationship between bend angle and heat transfer rate.

V. PREPARATION OF MODEL

Dimensions of 1 inch pipe:

- ✓ Outer diameter – 1.315 inch = 33.401mm
- ✓ Inner diameter – 1.049 inch = 26.64 mm
- ✓ Thickness - 0.133 inch = 3.38 mm

GI pipe is taken for study with different bend angles (0°- 125°). Water is working fluid. Inlet temperatures vary from (20-75°C). Inlet velocities are 5m/s and 10m/s.

Q = U A DT

U – Overall heat transfer co-efficient

A – Provisional area for heat transfer

DT – Temperature difference of two fluids

Free Convection Air- Forced liquid (flowing) water: $U = 5\ W/m^2\ K$

$A = D_0 * L$

D_0 – Outside diameter

L – Length (Each pipe taken to 1m)

A = 0.033401 m^2

DT = Water temperature - Ambient temperature (assumed)

Ambient temperature is varied place to place and time to time. Here, it is fixed as 25°C. Therefore, it is commonly taken as temperature difference. For straight pipe theoretical results listed below:

Table 1 Heat transfer rate

Sl.No	Temperature Difference (°C)	Heat transfer (W)
1	5	0.835
2	15	2.51
3	35	5.85
4	50	8.4

Fig. 1 Straight pipe or 0° bend

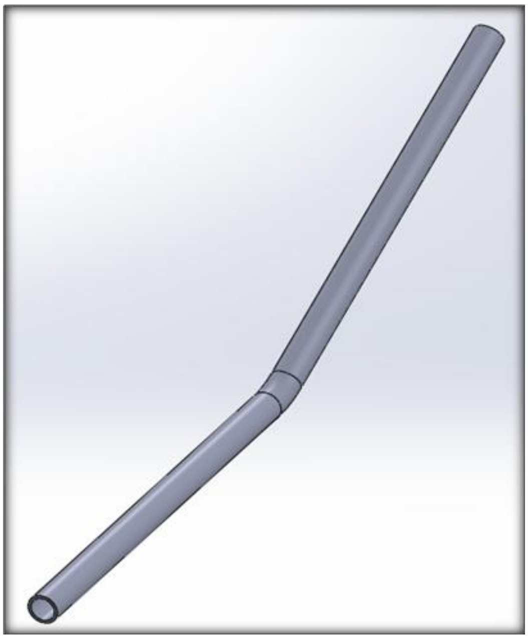

Fig. 2 25° bend

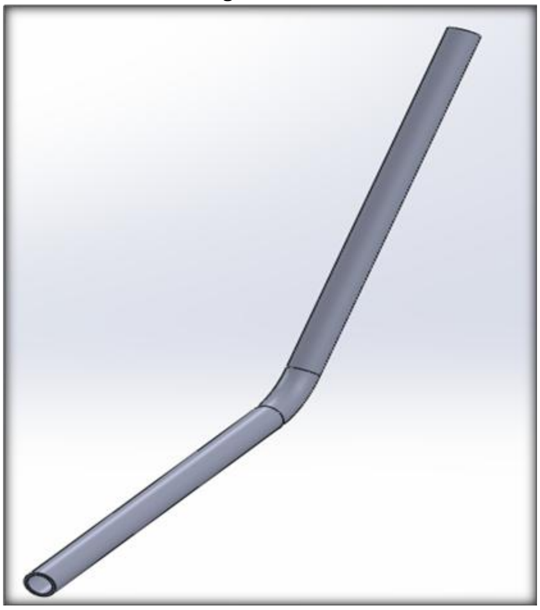

Fig. 3 45° bend

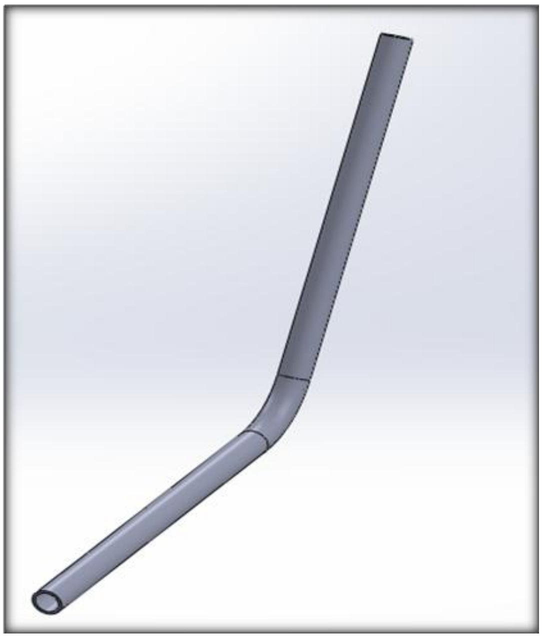

Fig. 4 60° bend

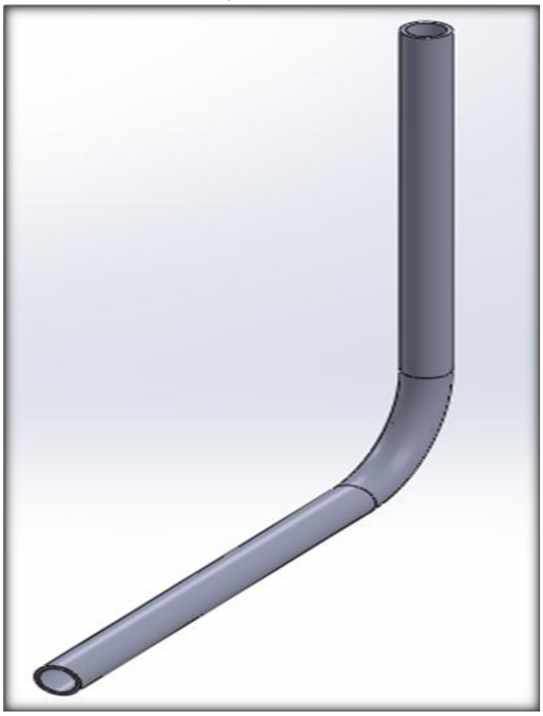

Fig. 5 90° bend

Fig. 6 100° bend

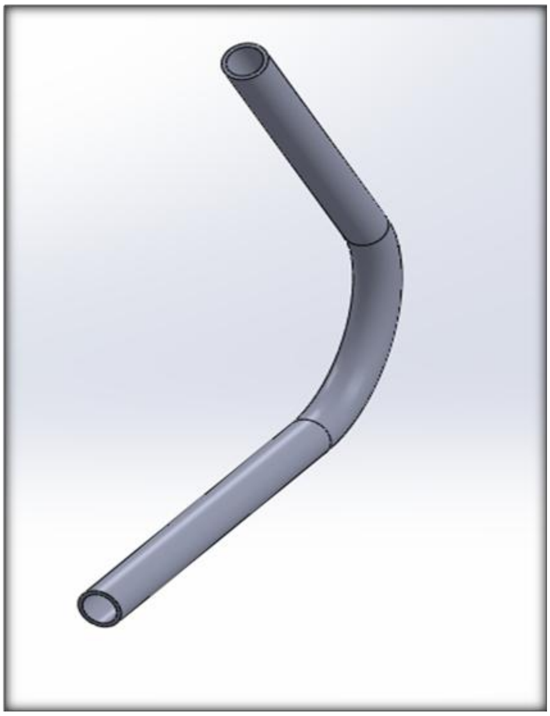

Fig.7 125° bend

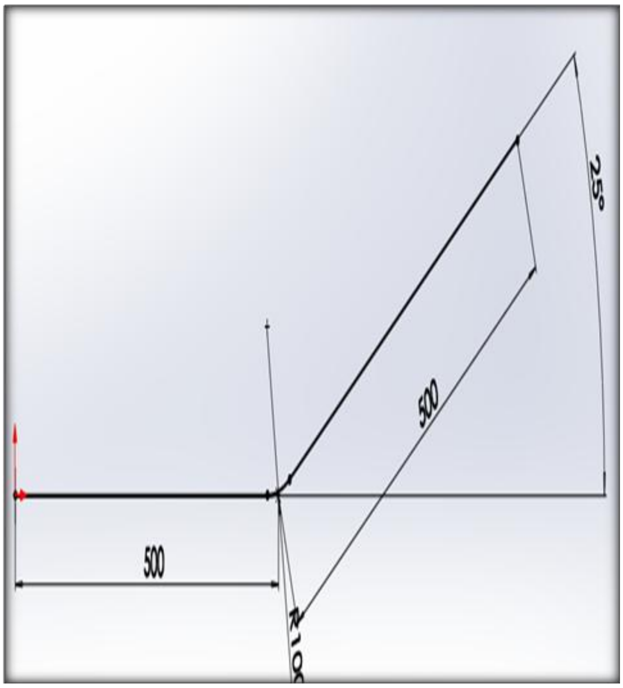

Fig. 8 Bend radius and angle measurement

VI. CONCLUSION AND FUTURE WORK

Requirement of heat transfer enhancement in heat transfer applications is increased to bring efficient process or products where ever necessary. Turbulence is induced in shell side of S & T exchanger by placing baffles to get more heat transfer rate. Some of the literatures speak about turbulence in U-bend of tubes in S & T exchanger and heat transfer enhancement. It is required to know relationship between pipes bend angle and heat transfer rate (indirectly turbulence) for future implementation in S & T exchanger tubes. In present work, computational fluid dynamics is used to explore the relationship mentioned above. Straight pipe is the reference. From that, pipes are designed with different bend angles (25°- 125°). Water is working fluid. Inlet temperatures vary from (20-75°C). Inlet velocities are 5m/s and 10m/s. Outlet temperature is found using CFD. Based on turbulence at pipe bend, heat transfer varies to pipe material, correspondingly outlet temperature varies. From this, we can explore relationship between bend angle and heat transfer rate.

REFERENCES

1. Ahmed F. Khudheyer and Mahmoud Sh. Mahmoud, “ NUMERICAL ANALYSIS OF FIN-TUBE PLATE HEAT EXCHANGER BY USING CFD TECHNIQUE”, ARPN Journal of Engineering and Applied Sciences, VOL. 6, NO. 7, JULY 2011 ISSN 1819-6608
2. Arafat A. Bhuiyana , A. K. M. Sadrul Islam and M. Ruhul Amin, “Numerical Study Of 3d Thermal And Hydraulic Characteristics Of Wavy Fin And Tube Heat Exchanger”, Frontiers in Heat and Mass Transfer (FHMT), 3, 033006 (2012) DOI: 10.5098/hmt.v3.3.3006.
3. Biswas A. K. and Prasanta K Sinha, A N Mullick and B. Majumdar, “ Flow Investigation in a Constant Area Curved Duct”, International Journal of Engineering Research and Applications (IJERA) ISSN: 2248-9622, Vol. 2, Issue6, November- December 2012, pp.1232-1236.
4. Jignesh M. Chaudhari , Dattatraya Subhedar and Nikul Patel, “Experimental Investigation of Finned Tube Heat Exchanger”, International Journal of Innovative Research in Advanced Engineering (IJIRAE) ISSN: 2349-2163, Volume 1 Issue 5 (June 2014).
5. Jogi Nikhil G and Assist. Prof. Lawankar Shailendra M, “ Heat Transfer Analysis of Corrugated Plate Heat Exchanger of Different Plate Geometry: A Review”, International Journal of Emerging Technology and Advanced Engineering, ISSN 2250-2459, Volume 2, Issue 10, Oct 2012.
6. Kandylas I.P. and A.M. Stamatelos, “Engine exhaust system design based on heat transfer Computation”, Energy Conversion & Management 40 (1999) 1057±1072, Elsevier.
7. Kumaresan G. and S. Venkatachalapathy, “A REVIEW ON HEAT TRANSFER ENHANCEMENT STUDIES OF HEAT PIPES USING NANOFLUIDS”, Frontiers in Heat Pipes (FHP), 3, 043001 (2012) DOI: 10.5098/fhp.v3.4.3001.
8. Kumbhar D.G and , Dr. Sane N.K, “Heat Transfer Enhancement in a Circular Tube Twisted with Swirl Generator: A Review”, Proc. of the 3rd International Conference on Advances In Mechanical Engineering, January 4-6, 2010 s.v. National Institute of Technology, Surat - 395 007, Gujarat, India.
9. Paisarn Naphon and Somchai Wongwises, “A review of flow and heat transfer characteristics in curved tubes”, Renewable and Sustainable Energy Reviews 10 (2006) 463–490, Elsevier.
10. Parmar K.S., M.J.Patel and U. R. Soni, “ Enhance the Heat Transfer Rate of Finned Tube Heat Exchanger: A Review”, International Journal on Recent and Innovation Trends in Computing and Communication ISSN: 2321-8169 Volume: 2 Issue: 3 441 – 446.
11. Quamrul H. Mazumder, “ Effect of Liquid and Gas Velocities on Magnitude and Location of Maximum Erosion in U-Bend”, Open Journal of Fluid Dynamics, 2012, 2, 29-34.
12. Shinde Digvijay D. and Dange H. M, “Heat Transfer Analysis of a Cone Shaped Helical Coil Heat Exchanger”, International Journal of Innovations in Engineering and Technology (IJIET).
13. Shirgire N. D. and P. Vishwanath Kumar, “ Review on Comparative Study between Helical Coil and Straight Tube Heat Exchanger”, IOSR Journal of Mechanical and Civil Engineering, e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 8, Issue 2 (Jul. - Aug. 2013), PP 55-59.
14. Shivalingaswamy B.P and Narahari G A, “ Numerical Investigation of Convective Heat Transfer and Pressure Loss in a Round tube Fitted with Circular-Ring Turbulators”, International Journal of Scientific and Research Publications, Volume 4, Issue 4, April 2014 1 ISSN 2250-3153.
15. Suhas V. Patil and P. V.Vijay Babu, “Heat Transfer Augmentation in a Circular tube and Square duct Fitted with Swirl Flow Generators: A Review”, International Journal of Chemical Engineering and Applications, Vol. 2 , No. 5 , October 2011.
16. Xue Hong Wu, LiHua Feng, DanDan Liu, Hao Meng and YanLi Lu, “ Numerical study on the effect of tube rows on the heat transfer characteristic of dimpled fin”.