IJSRD - International Journal for Scientific Research & Development| Vol. 5, Issue 04, 2017 | ISSN (online): 2321-0613

CFD Analysis of Fluid Flow through U-Pipe

Biranjeet Halder[1] Abhishek Shukla[2]
[1,2]Department of Mechanical Engineering
[1,2]SSTC (SSEC) Bhilai CG, India

Abstract— This article focuses on the hydrodynamic analysis of U-Pipe using finite volume approach. A two dimensional model of U-Pipe has been developed and the hydrodynamic performance characteristics has been examined and compared with the same length of straight pipe. The flow separation and boundary layer separation has also been examined for wide range of Reynolds number. At the bend section the friction factor has examined and found that the inner and outer surface has un-symmetrical pattern and at turbulent intensity also varies in comparison with straight pipe. The obtained finite volume results have been compared with the published data and the result shows good agreement.

Key words: U-Pipe, CFD, Turbulent Intensity

I. INTRODUCTION

Return bends are found in various applications involving two-phase flow, including heat exchangers, transport pipes and separators. Gas–liquid flows in bends are affected by centrifugal forces that tend to separate both phases. If the bend is oriented vertically, the flow is also subjected to gravitational and buoyant forces. The resulting effect of such forces is a change of the flow configuration as it passes through the bend, and the magnitudes of the forces depend on several aspects, such as direction (i.e. upward and downward), flow rates, physical properties and bend curvature.

II. LITERATURE REVIEW

Lima and Thome 2013 studied, flow pattern observations and pressure drop measurements in a U-bend and contiguous straight tubes are presented. The flow pattern observations were made with R134a at an inlet saturation temperature. The experimental data was compared to the predictions of a new multi-orientation flow pattern based frictional pressure drop method for U-bends of Silva Lima and Thome (2012c) showing that the model predicts 97% of the database within an error window of ±30%.

Georgios A. Florides 2013 developed a numerical model for simulating Single and double U-tube ground heat exchangers. The impact of multi-layer substrates on temperature distribution of ground heat exchanger. The model is also modified to allow the study of a double U-tube GHE in a single borehole and the assessment of its efficiency with regard to its building cost.

Gülşah Çakmak 2013 designed and examined U-tube heat exchanger and waste heat has been stored by using phase change material in the system. Calcium chloride hexa-hydrate with 29 °C melting temperature was used as phase change material and water was used as heat transfer fluid. It has been determined that 588 kJ and 417 kJ heat energy can be stored at two different water inlet temperatures of 65 °C and 45 °C, respectively.

Hasanpour 2016. experimentally studied heat transfer and friction factor in a double pipe heat exchanger which has an inner corrugated tube filled with various categories of twisted tapes from conventional tomodified types which include perforated, V-cut and U-cut types.

The results of themain parameters on heat transfer and pressure drop showthat the Nusselt number and friction factor for all cases of twisted tape corrugated tube are more than the empty corrugated tube.

M. Mozafari 2015. Studied the effect of inclination angle/flow direction on the patterns of a two-phase flow with low mass flux in a U-bend tube is not well documented. In this study, flow patterns and transitions for condensation of refrigerant R-134a inside U-bend tubes and their contiguous straight tubes are visually observed and analyzed.

Zhenbin He.2016 In the present study, heat transfer and flow resistance characteristic in the shell side of a vertical heat exchanger combined helical baffles with elliptic tubes were experimentally and numerically investigated. A helical baffle heat exchanger with circular tubes based on the same equivalent outside diameter of the elliptic tubes was also numerically studied for the performance comparison

Gyun-Ho Gim 2014, studied the surface roughness affecting the pressure drop in a pipe used as the steam generator of a PWR. Based on the CFD (Computational Fluid Dynamics) technique using a commercial code named ANSYS-FLUENT, a straight pipe was modeled to obtain the Darcy frictional coefficient, changed with a range of various surface roughness ratios as well as Reynolds numbers. The result is validated by the comparison with a Moody chart to set the appropriate size of grids at the wall for the correct consideration of surface roughness.

Prasun Dutta 2016 explore an effort to find the flow separation characteristics under high Reynolds number in pipe bends. Single phase turbulent flow through pipe bends is investigated using k-ε turbulence model. After the validation of present model against existing experimental results, a detailed study has been performed to study the influence of Reynolds number on flow separation and reattachment.

Bhusan et al. 2017 numerically examine the flow and heat transfer characteristics in 180 bend pipe with having flow of water-fly ash slurry. In their work they considered RNG k-ε turbulence model. The pressure drop and heat transfer has been examined for multiphase flow using finite volume approach.

Li et al. 2017 performed a plastic collapse load analysis for un-cracked and circumferential through-wall cracked pipe bends under torsion moment by three dimensional FE methods considering geometric nonlinearity. Results show that pipe parameters bend radius-to-radius and crack length have l

III. MATHEMATICAL MODELLING

For the fluid flow through pipe, duct and channel the conventional governing equations are the Navier–Stokes equations can be written in the most useful form for the development of the finite volume method:

$$\rho \frac{Du}{Dt} = -\frac{\partial p}{\partial x} + div(\mu\, grad u) + S_{Mx} \qquad (1)$$

$$\rho \frac{Dv}{Dt} = -\frac{\partial p}{\partial y} + div(\mu\, grad v) + S_{My} \qquad (2)$$

$$\rho \frac{Dw}{Dt} = -\frac{\partial p}{\partial z} + div(\mu\, grad w) + S_{Mz} \qquad (3)$$

Governing equations of the flow of a compressible Newtonian fluid

A. *Continuity*

$$\frac{\partial \rho}{\partial x} + div(\rho u) = 0$$

B. *x-momentum*

$$\frac{\partial(\rho u)}{\partial x} + div(\rho u u) = -\frac{\partial p}{\partial x} + div(\mu\, grad u) + S_{Mx} \qquad (4)$$

C. *y-momentum*

$$\frac{\partial(\rho v)}{\partial y} + div(\rho v u) = -\frac{\partial p}{\partial y} + div(\mu\, grad v) + S_{My} \qquad (5)$$

D. *z-momentum*

$$\frac{\partial(\rho w)}{\partial z} + div(\rho w u) = -\frac{\partial p}{\partial z} + div(\mu\, grad w) + S_{Mz} \qquad (6)$$

E. *Energy*

$$\frac{\partial(\rho i)}{\partial t} + div(\rho i u) = -p\, div u + div(k\, grad T) + \Phi + S_i \qquad (7)$$

Using various correlation FEV results are been compared analytically

$$h_f = f \frac{LV^2}{D_h 2g}$$

Where,

f is the friction factor for fully developed laminar flow

L: length of the channel, duct, pipe

V: mean velocity of the flow

d: diameter of the pipe

f is the friction factor for fully developed laminar flow:

$$f = \frac{64}{Re} \text{ (For Re<2000) } Re = \frac{\rho u_{avg} d}{\mu}$$

C_f is the skin friction coefficient or Fanning's friction factor.

For Hagen-Poiseuille flow: $C_f = \tau_{wall} l \frac{1}{2} \rho u_{avg}^2 = \frac{16}{Re}$

For turbulent flow:

$$\frac{1}{\sqrt{f}} = 1.74 - 2.0 \log_{10}\left[\frac{\varepsilon_p}{R} + \frac{18.7}{Re\sqrt{f}}\right] \text{ Moody's Chart}$$

R: radius of the channel, duct, pipe

ε_p: degree of roughness (for smooth channel, duct, pipe, $\varepsilon_{p=0}$)

$Re \rightarrow \infty$: Completely rough channel, duct, pipe.

IV. METHODOLOGY

The ANSYS 14.5 finite element program was used for analyzing flow in U-Pipe and straight Pipe. For this purpose, the key points were first created and then line and spline segments were formed. The lines were combined to create an area. Finally, this area was extruded a We modeled the U-Pipe and straight Pipe.

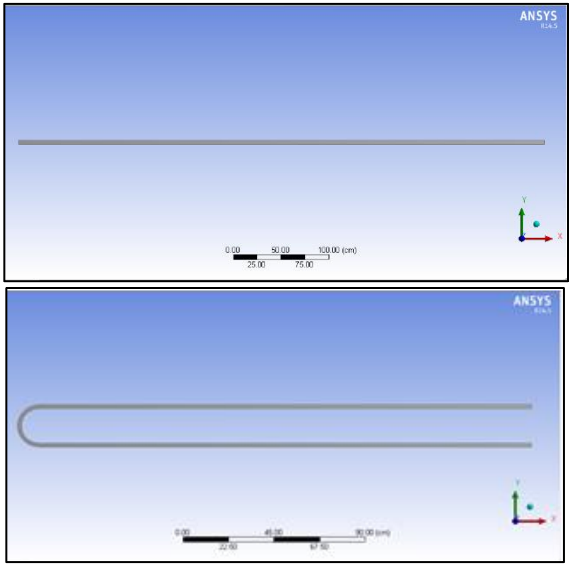


Fig. 1: Model Geometry

A 20-node three-dimensional structural solid element was selected to model the U-Pipe and straight Pipe. The U-Pipe and straight Pipe was discretized into 6529 elements with 5990 nodes. The wavy channel boundary conditions can also be provided in mesh section through naming the portion of modeled Pipe i.e Inlet, Outlet, Top wall, Bottom Wall, inner radius and outer radius.

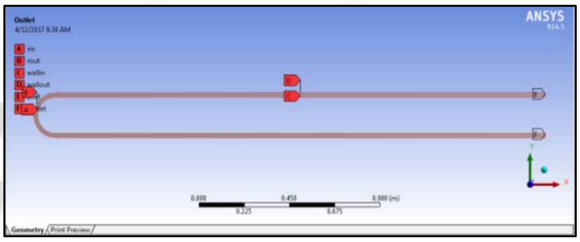
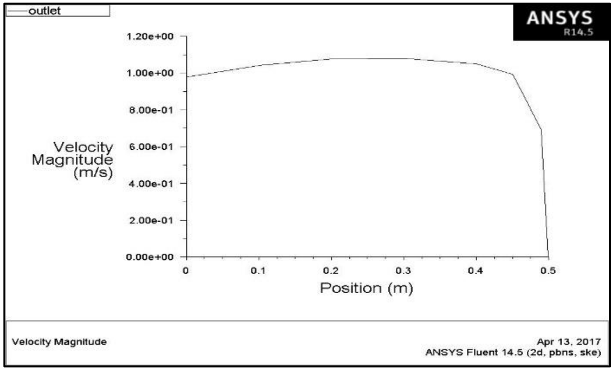


Fig. 2: Boundary Condition

V. RESULT AND DISCUSSION

The governing equations of the problem were solved, numerically, using a Element method, and finite Volume method (FVM) used in order to calculate the Hydrodynamic characteristics of a U-Pipe and straight Pipe. As a result of a grid independence study, a grid size of 106 was found to model accurately the Hydrodynamic performance characteristics are described in the corresponding results.

The accuracy of the computational model was verified by comparing results from the present study with those obtained by bdelkrim [11] and Azzola [12] whose works are based on experimental, and FVM results.



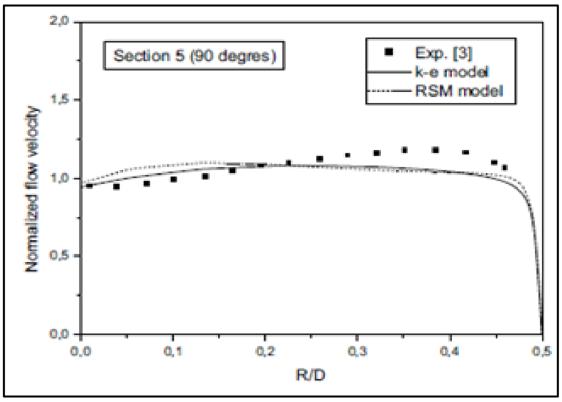


Fig. 3: Validation of Velocity magnitude of U-pipe

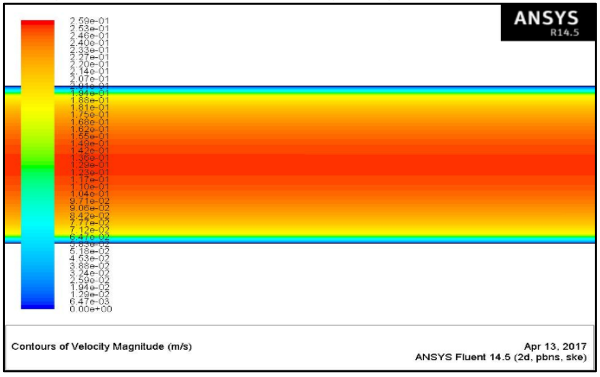


Fig. 4: Contour Plot of Velocity Magnitude of straight Pipe

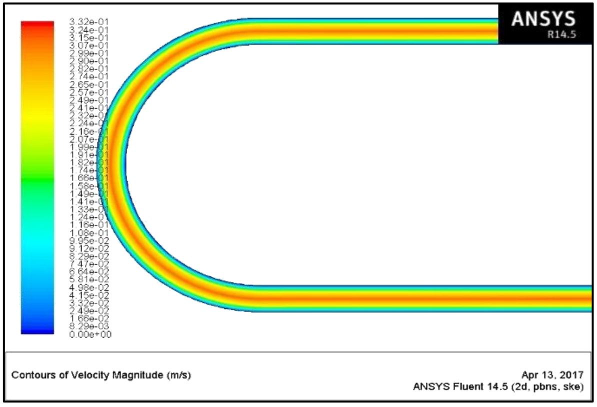


Fig. 5: Contour Plot of Velocity Magnitude of U- Pipe

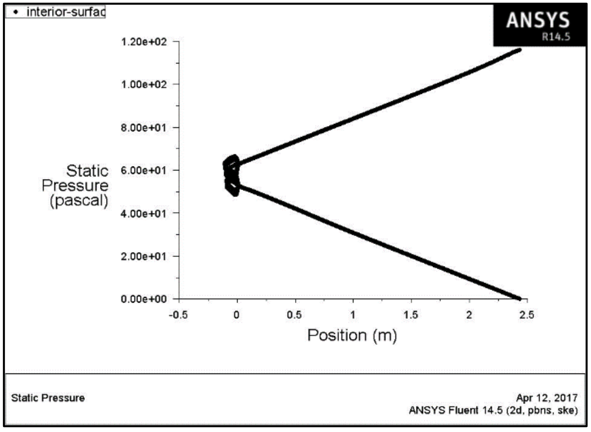


Fig. 6: Static pressure distributions across U-Pipe

Figure 6 illustrates the pressure drop across U-pipe at high Reynolds number. It has been observed that the pressure across the U-pipe continuously goes on decreasing from inlet to outlet. It is interesting to know that across the pipe bending region the variation in pressure has been seen in term of increase and decrease.

Figure 7 shows the pressure drop across straight pipe at high Reynolds number. It has been observed that the pressure significantly decreases throughout the pipe length. It is interesting to know that in straight channel this drop in pressure is linear throughout the pipe length, which can be evident from figure 7 while in U-pipe the drop in pressure is linear but have different slope in different direction.

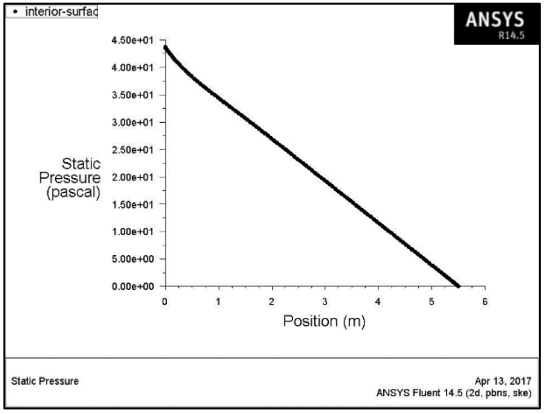


Fig. 7: Static pressure distribution across Straight Pipe

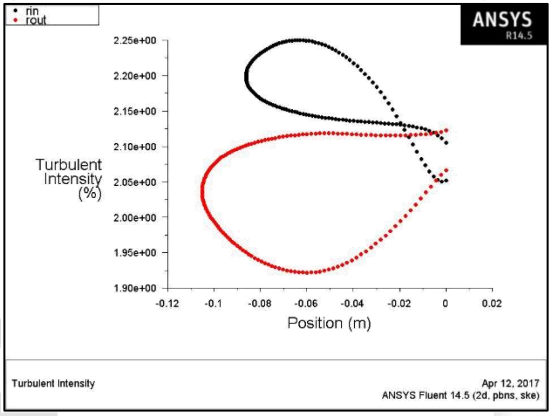


Fig. 8: Turbulent intensity across U-Pipe

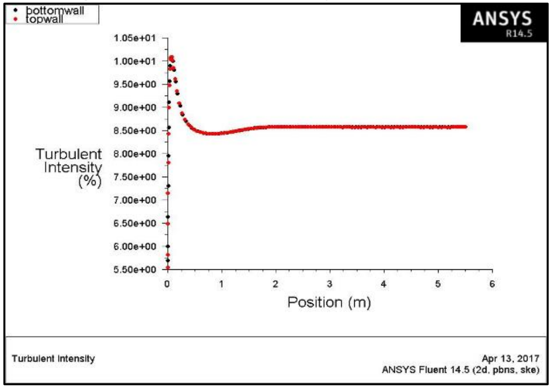


Fig. 9: Turbulent intensity across U-Pipe

Figure 8 and 9 shows the turbulent intensity across U-Pipe and straight pipe. It has been seen that in U pipe, at the inner radius region experiences higher rate of turbulence has compared to outer radius region this is due to higher rate of flow separation in inner radius region due to friction. While in Straight pipe the nature of turbulent intensity remains same in top and bottom wall and experiences equal rate of turbulence.

VI. CONCLUSION

The pressure drop at the bend section, is quite complex as compared to straight pipe.

On comparing the performance in terms of heat transfer between straight pipe and bend pipe, bend pipe yields better performance.

The rate of turbulence intensity increases as the angle of pipe bend varies.

The flow separation and vortex formation at the bend section plays major role in heat and momentum transfer in bend pipe.

The inner region experience higher rate of friction and shear stress as compared to the outer region.

The wall shear stress at the bend section, i.e. at inner region and at outer region is quite different.

REFERENCES

[1] Ricardo J. da Silva Lima , John R. Thome , Two-phase flow pressure drops in U-tubes: Towards more accurate measurement methods and prediction models: international journal of refrigeration 3 6 (2013) 492 -503
[2] Georgios A. Florides , Paul Christodoulides, Panayiotis Pouloupatis, Single and double U-tube ground heat exchangers in multiple-layer substrates; Applied Energy 102 (2013) 364–373
[3] Gülşah Çakmak, Experimental investigation of thermal storage in U-tube heat exchanger: International Communications in Heat and Mass Transfer 44 (2013) 83–86.
[4] A. Hasanpour, M. Farhadi , K. Sedighi, Experimental heat transfer and pressure drop study on typical, perforated, V-cut and U-cut twisted tapes in a helically corrugated heat exchanger: International Communications in Heat and Mass Transfer 71 (2016) 126–136
[5] M. Mozafari , M.A. Akhavan-Behabadi , H. Qobadi-Arfaee , P. Hanafizadeh , M. Fakoor-Pakdaman, Experimental study on condensation flow patterns inside inclined U-bend tubes; Experimental Thermal and Fluid Science 68 (2015) 276–287
[6] Zhenbin He, Xiaoming Fang, Zhengguo Zhang, Xuenong Gao, Numerical investigation on performance comparison of non-Newtonian fluid flow in vertical heat exchangers combined helical baffle with elliptic and circular tubes. Applied Thermal Engineering 100 (2016) 84–97
[7] Gyun-ho gim, se-myong chang, sinyoung lee, and gangwon jang, fluid-structure interaction in a u-tube with surface roughness and pressure drop: nuclear engineering and technology, vol.46 no.5 october 2014
[8] Prasun Dutta , Sumit Kumar Saha, Nityananda Nandi, Nairit Pal. Numerical study on flow separation in 90° pipe bend under high Reynolds number by k-ε modelling. Engineering Science and Technology, an International Journal 19 (2016) 904–910
[9] Jian Li, Chang-Yu Zhou, Le Chang, Xin-Ting Miao, Xiao-Hua He, Limit loads for un-cracked and circumferential through-wall cracked pipe bends under torsion moment considering geometric nonlinearity, Thin-Walled Structures, Volume 116, July 2017, Pages 37-52
[10] Bibhuti Bhusan Nayak, Dipankar Chatterjee, Amar Nath Mullick, Numerical prediction of flow and heat transfer characteristics of water-fly ash slurry in a 180° return pipe bend, International Journal of Thermal Sciences, Volume 113, March 2017, Pages 100-115
[11] Abdelkrim Miloud1, Mohammed Aounallah1,a, Mustapha Belkadi1, Lahouari Adjlout, Omar Imine and Bachir Imine, Turbulent flow computation in a circular U-Bend, EPJ Web of Conferences, 67, 2014, 02075
[12] J. Azzola and J. A. C Humphrey, Developing Turbulent Flow in a 180° Curved Pipe and its Downstream Tangent, Report LBL-17681, Materials and Molecular Research Division, Lawrence Berkeley Laboratory, University of California (1984).