Computational Investigation of Fluid Flow 90º Bend Pipe Using Finite Volume Approach

Manish Kumar Rathore [1], Dilbag Singh Mondloe[2], Satish Upadhyay[3]

[1]*M.Tech Scholar, Government Engineering College, Jagdalpur, Chhattisgarh, India*
[2]*Assistant professor, Dept. of Mechanical Engineering, GEC Jagdalpur, Chhattisgarh, India*
[3]*Lecturer, Dept. of Mechanical Engineering, GEC Jagdalpur, Chhattisgarh, India*

***

Abstract - *Bend pipe are widely been used in various engineering applications and have significant importance in fluid flow applications. In this paper concentrates on computational analysis of 90º bend pipe using finite volume approach. A two dimensional model of 90º bend has been developed by using ANSYS 14.5 and the hydrodynamic performance characteristics has been examined and compared with the same length of straight pipe using Finite volume tool i,e. ANSYS Fluent. 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*: 90o bend pipe, CFD, Reynolds number, Turbulence, pressure, velocity, skin friction .*

1.INTRODUCTION

In the field of heat transfer and fluid flow implementation of bend pipe or curved surface are the effectively techniques of enhancing heat transfer in various engineering applications with have internal flow applicability.

Due to presence of obstacle in the flow region leads to break boundary layer and creates turbulence this phenomenon is of substantial important phenomena as it may strongly affect pressure drop, wall friction, heat transfer, occurrence of extreme temperatures, and stability of the flow. when fluid flows in a pipe bend a secondary motion of flow is developed due to the presence of centrifugal force which leads to the formation of secondary flow and as a result the fluid particles near the surface are driven outward. This secondary flow is superimposed on its primary axial flow leading to high velocity at the outer core of the pipe bend. As result of these secondary flow within the pipe the rate turbulence intensity increases, which also increase the friction factor and also affect the other hydrodynamic characteristics.

Row 1970 experimentally examined the heat transfer effect in large wavy pipe due to secondary flow which leads to complete interchange of fluid in the wall and the central core line. Their work particularly focuses on single phase flow in various pipe bends like 180 U-bend, 45o/45o S-bend attached to the end of a long straight pipe.

Janyanti et al 1993 performed CFD analysis to examine the Gas-particle motion in 90o and 1800 circular cross-section pipe bends. They found that the secondary flow induced in the gas phas-e due to curvature affects the motion of the particles which causes the smaller particles to come out of the bend without deposition.

Clarke and Finn 2008 numerically investigates the Laminar flow of secondary working fluid such as potassium formate through heat exchanger U-bends and found that heat transfer increases at downstream of the bend.

Dhanasekaran 2012 perform CFD analysis of closed-loop steam cooling in a 180-degree bend tube sof advanced gas turbine. Their aims is to validating a CFD model against experimental results in a 180-degree tube bend and applying the model to predict the mist/steam cooling performance at gas turbine working conditions. The obtained results show that the CFD model can predict the wall temperature within 8% of experimental steam-only flow and 16% of mist/steam flow condition

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.

Pedro M. de Oliveira. 2014. Gas–liquid flows in curved tubes are found in a number of applications, such as heat exchangers and transport pipes. The present work deals with air–water flow in 180tube bends (curvatures of 6.1, 8.7, and 12.2) that connect two 5-m long 26-mm ID horizontal tubes. The bend lies in the vertical position and the two-phase flow can be set as upward or downward. The straight and curved segments of test section were made from borosilicate glass to enable visual access to the two-phase flow.

Chen et al. 2015 experimentally analyzed the effect of length and bending angle on the cooling performance of flat plate heat pipes (FPHP) it has been observed that optimum liquid filling ratio increases as length of pipe increases. While, shorter pipe had a larger effective thermal conductivity, and implying more efficiently. The bending of pipe would not affect the optimum liquid filling ratio of a FPHP.

Hasanpour et al. 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.

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. Different exact and approximate techniques have also been used to solve the different problems in fluid mechanics [19-26].Other significant works with the implementation of keε turbulence model were studied by Cardwell et al. 2011 and Walker 2005. Different approach was proposed by Johansen et al. 2003, Pinson and Wang 1999, Zimparov 2004 and Zhu and Kuznetsov 2005 to study the influence of the rough wall by using the modified mixed length model.Apart from these many research has carried out the research in the field of Magneto hydrodynamic flow (MHD) to explore the effect of magnetic field on the boundary layer flow control [27-32].

1.1 Mathematical Modeling

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

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

x-momentum

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

y-momentum

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

z-momentum

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

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:

$f = \frac{64}{Re}$ (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: $\frac{1}{\sqrt{f}} = 1.74 - 2.0\log_{10}\left[\frac{\varepsilon_p}{R} + \frac{18.7}{Re\sqrt{f}}\right]$

Moody's Chart
R: radius of the channel, duct, pipe
ε_p: degree of roughness (for smooth channel, duct, pipe, $\varepsilon_{p=0}$)
$Re \to \infty$: Completely rough channel, duct, pipe.

1.2 Methodology

The ANSYS 14.5 finite element program was used for analyzing flow in 90° bend 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 90° bend Pipe and straight Pipe.

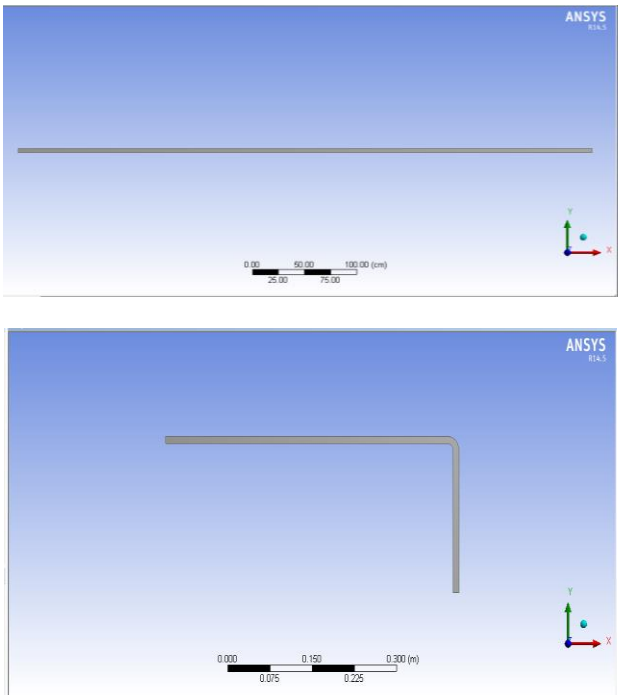

Fig -1: Model Geometry

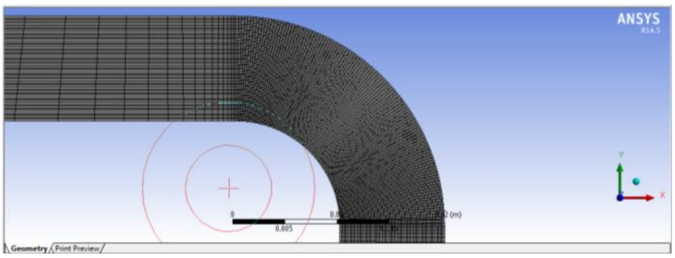

Fig -2: Mesh Model

The modeled 90° bend pipe and straight pipe was discretized into multiple of 105 i.e. 22950 elements with 23332 nodes and the mesh is fine and structured which can be seen from figure 2. The developed geometry for the simulation (FEV analysis) is been taken form Dutta et al. [15] which is taken as base paper for the proposed work.

2. RESULTS AND DISCUSSIONS

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 90° bend Pipe and straight Pipe. As a result of a grid independence study, a grid size of 10^6 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 Dutta et al. [15], sudo et al. [13] , Kim et al. [14] and Tanaka et al. [12] whose works are based on experimental, and FVM results.

On considering base paper as Dutta et al.[35] the measurements of velocities were performed at a Reynolds number of 6×10^4 for the validation and the result are as plotted in fig 3.Figure 6 illustrates the pressure drop across Bend pipe at high Reynolds number. It has been observed that the pressure across the Bend 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.

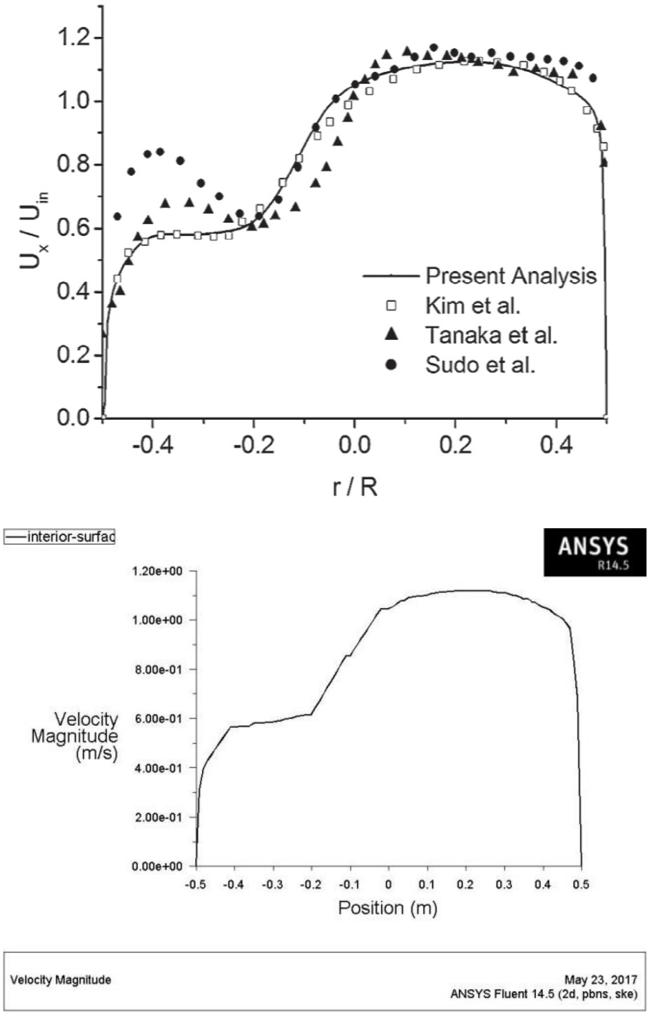

Fig -3: Validation of Velocity magnitude of 90° bend Pipe

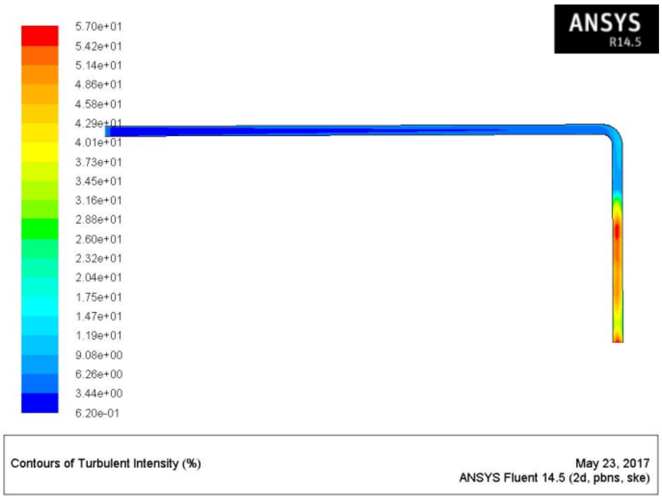

Fig -4: Contour Plot of turbulent intensity Bend pipe

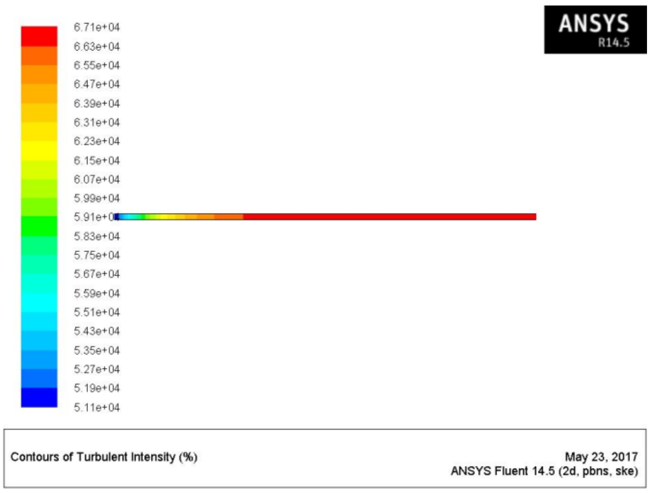

Fig -5: Contour Plot of turbulent intensity Straight Pipe

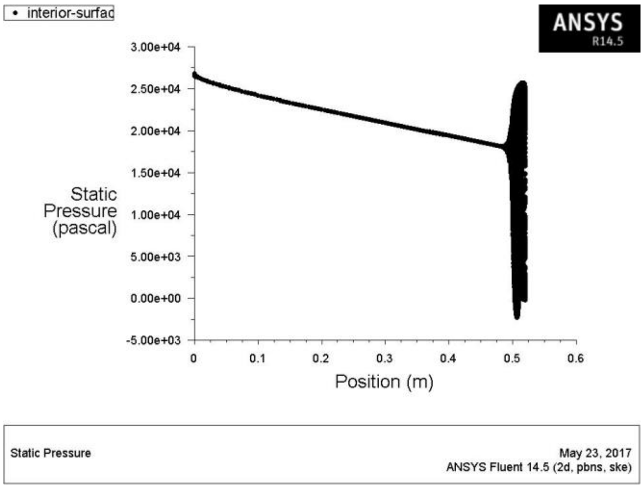

Fig -6: Static pressure distributions across Bend pipe

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 Bend pipe the drop in pressure is linear but have different slope in different direction.

Figure 8 and 10 shows the skin friction coefficient across Bend pipe and straight pipe. It has been seen that in 90° bend pipe, at the inner radius region i.e. (elbow_win) experiences higher rate of friction has compared to outer radius region i.e. (elbow_wout) this is due to higher wall shear stress across inner radius region. While in Straight pipe the nature of skin friction coefficient remain same in top and bottom wall and experiences equal rate of skin friction coefficient.

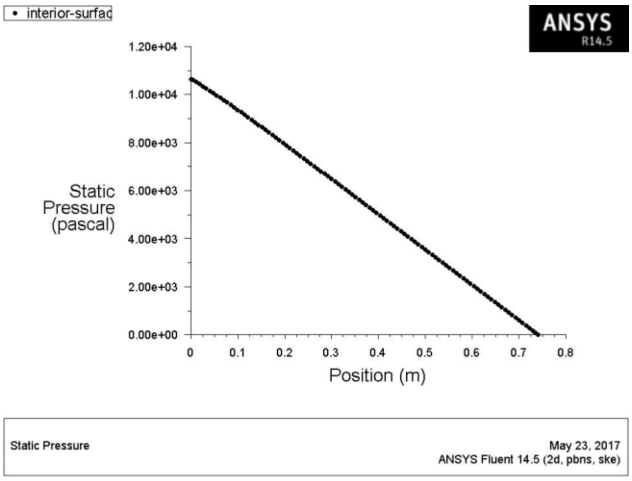

Fig -7: Static pressure distribution across Straight Pipe

The un symmetrical nature of skin friction coefficient at the wall of bend pipe is due to having different turbulence intensity at the inner radius region i.e. (elbow_win) and outer radius region i.e. (elbow_wout).This can be seen from figure 9. While in straight pipe the nature of skin friction coefficient at the wall i.e. top and bottom wall is same. This can be seen from figure 10

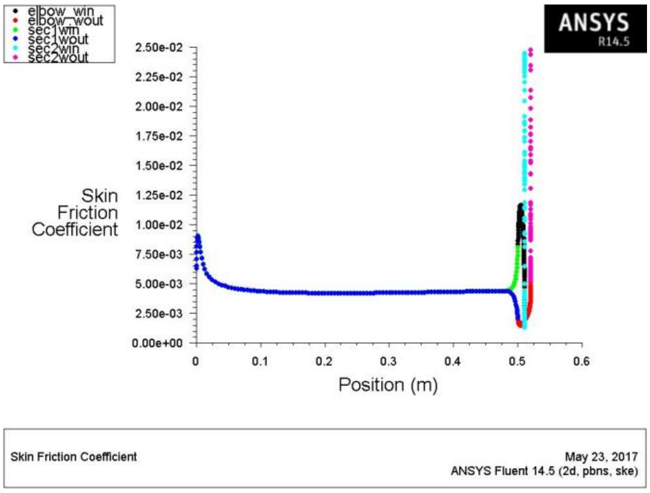

Fig -8: skin friction coefficient across Bend pipe

Figure 11 shows the influence of Reynolds number on Skin Friction coefficient. It has seen that as Reynolds number increases skin friction increases. This is because of occurrence of turbulence in flow which results in increase in friction. The contour plot of turbulence intensity can be seen from figure 4 and 5

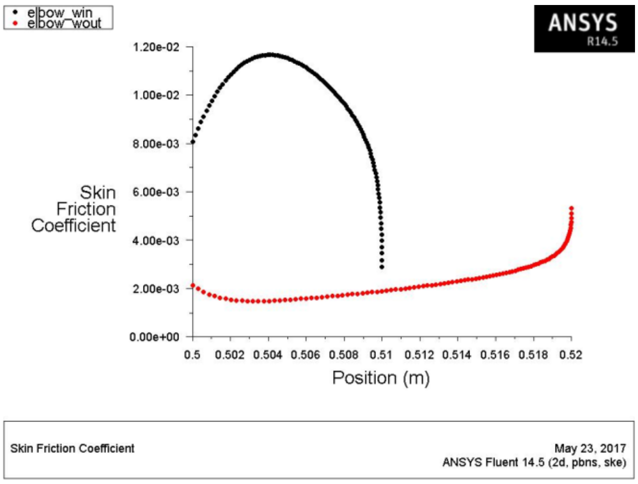

Fig -9: skin friction coefficient across the bend section

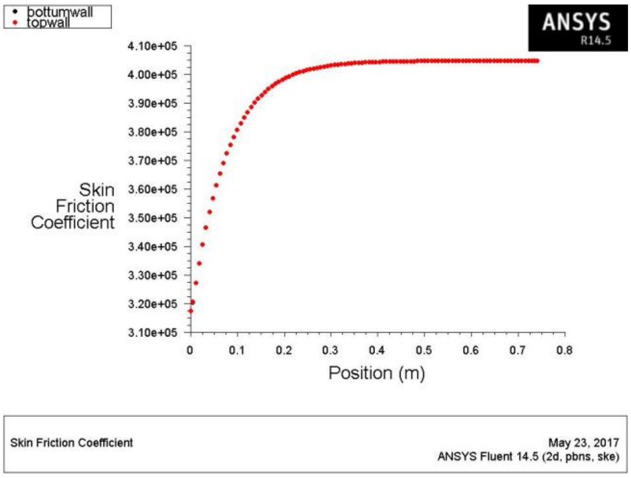

Fig -10: skin friction coefficient across Straight Pipe

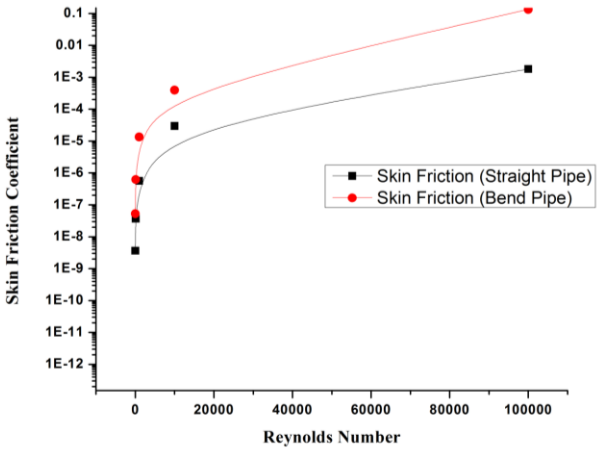

Fig -11: Influence of Reynolds number on Skin friction coefficient

Furthermore the trend of discrepancy is same for both the pipes i.e. 90º bend pipe and straight pipe. But the 90º bend pipe has higher rate of friction.
It has also been seen that on comparing the skin friction coefficient straight pipe has 98.65% less friction has compared to bend pipe at high Reynolds number.

2.2 CONCLUSION

On the basis of developed computational model of 90º bend pipe and straight pipe following conclusions has been drawn which as follows:

- The developed computational model of 90º bend pipe has been compared with the result of experimental and computation work of available literature and the result shows similar trend.
- Increase in Reynolds number, the skin friction factor increases simultaneously.
- The turbulence intensity within the 90º bend pipe is more has compared to straight pipe.
- The pressure drop in straight pipe is continuously linear, while in 90º bend pipe at the bend section sudden increase and decrease has been seen.
- Implementing bend in pipe results in increase in rate of turbulence. Therefore, in heat exchanger where high turbulence is required bend pipe are widely been used.
- The nature of skin friction coefficient across the inner radius region and outer radius region is un symmetrical.
- While in straight pipe the nature of skin friction coefficient is symmetrical at top and bottom wall.
- The straight pipe has 98.65% less friction has compared to bend pipe at high Reynolds number.

REFERENCES

[1] Rowe M. Measurements and computations of flow in pipe bends. J Fluid Mech 1970;43:771-783.

[2] Jayanti S, Wang MJ, Mayinger F. Gas-particle flow through bends. IMechE C461 1993;24:161-166.

[3] Clarke R, Finn DP. Numerical investigation of the influence of heat exchanger U-bends on temperature profile and heat transfer of secondary working fluids. In: 5th European thermal-Sci. Conf., The Netherlands; 2008. p. 1-8.

[4] T.S. Dhanasekaran, Ting Wang, Numerical model validation and prediction of mist/steam cooling in a 180-degree bend tube: International Journal of Heat and Mass Transfer 55 (2012) 3818–3828

[5] 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

[6] Pedro M. de Oliveira, Jader R. Barbosa Jr. Pressure drop and gas holdup in air–water flow in 180_ return bends:

International Journal of Multiphase Flow 61 (2014) 83–93
[7] Jung-Shun Chen, Jung-Hua Chou, The length and bending angle effects on the cooling performance of flat plate heat pipes, International Journal of Heat and Mass Transfer, Volume 90, November 2015, Pages 848-856
[8] 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
[9] 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
[10] Abdelkrim Miloud, 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
[11] 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)
[12] M.A. Tanaka, H. Ohshima, H. Monji, Numerical investigation of flow structure in pipe elbow with large eddy simulation approach, in: ASME 2009 Pressure Vessels and Piping Conference, American Society of Mechanical Engineers, 2009, pp. 449–458
[13] K. Sudo, M. Sumida, H. Hibara, Experimental investigation on turbulent flow in a circular-sectioned 90-degree bend, Exp. Fluids 25 (1) (1998) 42–49.
[14] J. Kim, M. Yadav, S. Kim, Characteristics of secondary flow induced by 90-degree elbow in turbulent pipe flow, Eng. Appl. Comput. Fluid Mech. 8 (2) (2014) 229–239.
[15] 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 ■■ (2016)
[16] Cardwell ND, Vlachos PP, Thole KA. Developing and fully developed turbulent flow in ribbed channels. Exp Fluids 2011;50(5):1357e71
[17] Walker P. CFD modelling of heat exchanger fouling. University ofNew South Wales Sydney Australia; 2005
[18] Johansen S, Skalle P, Sveen J. A generic model for calculation offrictional losses in pipe and annular flows. J Can Pet Technol 2003;42(6).
[19] Pinson MW, Wang T. Effect of two-scale roughness on boundary layer transition over a heated flat plate: part 2dboundary layer structure. J Turbomach 1999;122(2):308e16
[20] Zimparov V. Prediction of friction factors and heat transfer coefficients for turbulent flow in corrugated tubes combined with twisted tape inserts. Part 1: friction factors. Int J Heat Mass Transf 2004;47(3):589e99.
[21] Zhu J, Kuznetsov A. Forced convection in a composite parallel plate channel: modeling the effect of interface roughness and turbulence utilizing a keε model. Int Commun Heat Mass Transf 2005;32(1):10e8.
[22] P. Valinataj-Bahnemiri, A. Ramiar, S.A. Manavi, A. Mozaffari, Heat transfer optimization of two phase modeling of nanofluid in a sinusoidal wavy channel using Artificial Bee Colony technique, Eng. Sci. Technol. Int. J. 18 (4) (2015) 27–737.
[23] M. Azimi, A. Mozaffari, Heat transfer analysis of unsteady graphene oxide nanofluid flow using a fuzzy identifier evolved by genetically encoded mutable smart bee algorithm, Eng. Sci. Technol. Int. J. 18 (1) (2015) 106–123.
[24] Ebtehaj, H. Bonakdari, A.H. Zaji, H. Azimi, F. Khoshbin, GMDH-type neural network approach for modeling the discharge coefficient of rectangular sharp-crested side weirs, Eng. Sci. Technol. Int. J. 18 (4) (2015) 746–757.
[25] F. Selimefendigil, H.F. Öztop, Mixed convection of ferrofluids in a lid driven cavity with two rotating cylinders, Eng. Sci. Technol. Int. J. 18 (3) (2015) 439–451.
[26] M. Shahrbanozadeh, G.A. Barani, S. Shojaee, Simulation of flow through dam foundation by isogeometric method, Eng. Sci. Technol. Int. J. 18 (2) (2015) 185–193.
[27] S. Mukhopadhyay, I.C. Mandal, Magnetohydrodynamic (MHD) mixed convection slip flow and heat transfer over a vertical porous plate, Eng. Sci. Technol. Int. J. 18 (1) (2015) 98–105.
[28] S. Das, R.N. Jana, O.D. Makinde, Mixed convective magneto hydrodynamic flow in a vertical channel filled with nanofluids, Eng. Sci. Technol. Int. J. 18 (2) (2015) 244–255.
[29] A. Mahdy, S.E. Ahmed, Thermosolutal Marangoni boundary layerMagneto-hydrodynamic flow with the Soret and Dufour effects past a vertical flat plate, Eng. Sci. Technol. Int. J. 18 (1) (2015) 24–31.
[30] K. Javaherdeh, M.M. Nejad, M. Moslemi, Natural convection heat and mass transfer in MHD fluid flow past a moving vertical plate with variable surface temperature and concentration in a porous medium, Eng. Sci. Technol. Int. J. 18 (3) (2015) 423–431.
[31] A. Khalid, I. Khan, A. Khan, S. Shafie, Unsteady MHD free convection flow of Casson fluid past over an oscillating vertical plate embedded in a porous medium, Eng. Sci. Technol. Int. J. 18 (3) (2015) 309–317.
[32] M. Hameed, A.A. Khan, R. Ellahi, M. Raza, Study of magnetic and heat transfer on the peristaltic transport of a fractional second grade fluid in a vertical tube, Eng. Sci. Technol. Int. J. 18 (3) (2015) 496–502.