IOSR Journal of Mathematics (IOSR-JM)
e-ISSN: 2278-5728, p-ISSN: 2319-765X. Volume 15, Issue 4 Ser. I (Jul – Aug 2019), PP 48-57
www.iosrjournals.org

Numerical Study of Water Flow through U-Bend Pipe

Sonkar Mondol[1], Md. Ismail Hossain[2] and Md. Tajul Islam[3]
([1] Department of Mathematics, Begum Rokeya University, Rangpur-5400, Bangladesh)
([2] Department of Mathematics, Begum Rokeya University, Rangpur-5400, Bangladesh)
([3] Department of Mathematics, Begum Rokeya University, Rangpur-5400, Bangladesh)
Corresponding Author: Md. Ismail Hossain

Abstract: *Pipe bends are an integral part of any pipeline network system as these provide flexibility in routing. Also the fluid flow inside the pipe bend has applications in various industrial sectors, from power plants and food industries to oil and gas companies and petrochemical procedures. In this study, a U-bend pipe is taken with constant diameter and length for geometry. Steady, laminar and fully developed flow with no-slip wall condition has been solved applying control volume technique using ANSYS FLUENT. Water vapor is used as working fluid with the assumption that the fluid is continuum. The continuity equation and Navier-Stokes equations are solved. The results obtained from our simulations agree with the results published in literature. The aim of the simulations is to investigate the behavior of the pressure and the velocity distributions in U- bend pipe. Our results show a change of the pressure and the velocity distributions in U-bend pipe for different Reynolds numbers.*

Keywords: *No-slip conditions, Reynolds number, Steady, Laminar, incompressible, U- bend pipe.*

Date of Submission: 27-06-2019 Date of acceptance: 13-07-2019

I. Introduction

In many industrial processes, pipe bends are used to convey supplies and a comprehensive understanding of fluid dynamics in pipe bends is essential for good pipe design, pipeline arrangements and frequently used fitting in piping system. Due to limitations of installation space, pipe bends are frequently used to change the direction of a pipe. In this way, the fluid can be transported to the desired delivery positions. Also fluid flow through in the pipe bend is one of the primary characteristics of water conveying structures, related structure in dams and irrigation system. Because of secondary circulation in the flow, the pattern of the flow in the pipe bend is complicated and these complications make it important issue in hydraulic engineering.

Also curved tubes and bends are widely employed in heat exchangers and flow transmitting devices. The curved channels can be in the form of helical, spiral, or U-tube return bend. For typical evaporator and condenser in refrigerators and air-conditioners that incorporated two phase flow inside the consecutive U-type return pipe bends is very common. As expected, the U-type return pipe bends will cause higher pressure drop than those of straight tubes.

The velocity and pressure distribution in a U-bend pipe (180 degree pipe bend) have been investigated by Bovendeerd et al. [1] via finite element method. They used a laminar parabolic profile as the inflow condition. Also they provided a coherent description of the flow field throughout the pipe bend, presenting the intensity of the secondary motions and axial velocity profiles for different section along the bend.

Nakayama et al. [2] performed their experimental studies on 180 degree ducts and the result of measuring in separated zone and distribution of Reynolds stress are discussed by them. They presented some equations for length and thickness of separation zone with regard to their numerical studies and observed bends allow centrifugal forces and other complex phenomena to occur and influence each other.

These phenomena are secondary flows, the formation and dispersion of ropes of fluid particles before and after bends, erosion at bend outer walls, an increase in wall–particle and particle–particle interactions, an increase in heat and mass transfer, as well as pressure drop. They are characterized by a pipe segment showing a degree of curvature, in which a fluid enters and exits by a linear segment of pipe .

Anwer et al. [3] investigated fluid flow and laminar characteristics of the U-bend pipe (180 degree pipe bend). They also investigated the flow development after the downstream tangent and they found that the flow reattaches 18 hydraulic diameters away from the bend exit.

Azzola et al. [4] investigated the fluid flow through in a U- bend pipe (180 degree pip bend). They observed that the fluid flow behavior for circular cross-sectioned U-bend is quite different than the squire cross-sectioned U-bend. They revealed that the secondary flow patterns start at 45 degree bend angle, which is not the case for square U-bend. Also Azzola et al. [4] checked the accuracy of their numerical result by comparing numerical results are reasonably good agreement with the experimental results or data.

Change et al. [5] used a square sectioned U- bend in their researches and they increased the upstream section length to ensure the fully developed flow before the bend inlet and measured the Reynolds stresses from the bend inlet to the U-bend pipe that is 180 degree bend angle at periodic locations. They showed that the increase of the Reynolds stresses in the region between the inlet and the 90 degree plane destabilizes the flow close to the concave wall. The measurements in between 90 degree and 180 degree plans showed intriguing changes in radial direction for the Reynolds stresses.

Sudo et al. [6] focused on the laminar flow in 180 degree bend section of a square duct by using hot-wire anemometer. They rotate an inclined hot-wire probe, inside the pipe bend and they measured the different velocity components and the Reynolds stresses. They found that the pressure difference between the right wall and left wall of the pipe bend causes a secondary flow in the cross section and as a result a centrifugal force acts on the fluid. This centrifugal force accelerates the fluid near the left wall decelerates near the right wall.

Also Ducret [7], focused on the fluid flow generated sound in U- bends (180degree pipe bend) in his study. He conducted experiments with different bend geometries. One of the bend geometries that he focused on is 180 degree bends. For these bend, the angle separating the upstream and downstream sections is 90 degree bends. He conducted experiments for two different radius of curvature that ratios of curvature R to pipe diameter D to be of 1.6 and 2.5. In his analysis, fluid flow induced sound power level of the straight pipe is subtracted from the sound power level measured for each bend to obtain the right sound power level increase for bends. He compared the results of two different bend curvatures with straight pipe results. According to his study, sound power levels are higher for the pipe with bends than straight pipes.

In this study the pattern of flow in U-bend with angle 180 degree with regard to inlet flow, has been investigated numerically and velocity profile and pressure distribution has been assessed. It is worth mentioning that the flow mode is steady and the range of Reynolds number is 827 to 1654.

II. Model Development

Problem Statement: We considered water vapor flow through the U-bend pipe in Cartesian coordinates systems. Fig.1 shows the geometry and coordinates of the U- bend pipe.

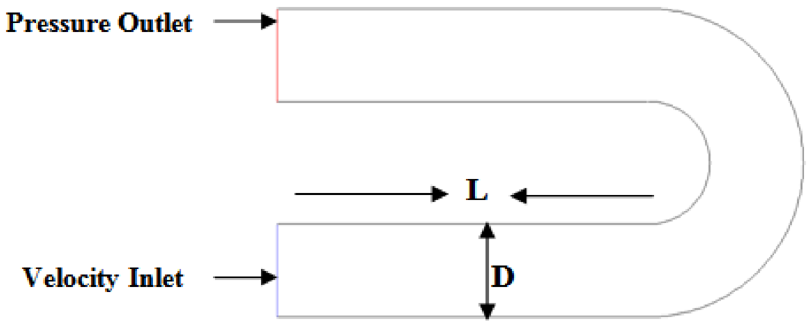


Figure.1: The schematic display of the fluid domain for the test case.

Where L=350 mm (0.35 m) is the length and D=2mm (0.002 m) is the constant diameter of the U-bend pipe. A number of investigations are performed with no slip boundary conditions on the walls. This model geometry constructed by using Gambit 2.2.30 component of the Fluent package as shown in figure 1. The meshing is scheme is composed of elements and type. The elements selected were quadrilateral defines the shape of the element used to mesh and the type parameter defines the meshing algorithm which in our case was pave. After the geometry is successfully meshed, the zone types are specified. A mesh file is exported for use in Fluent 6.3.26.

Governing Equations: The 2D water vapor flow is assumed to be steady and laminar. The two basic laws of conservation of mass or continuity equation and Navier-Stokes equations are solved for incompressible flows for Newtonian fluid. The incompressible forms of the governing equations are expressed in following forms. The equation of conservation of mass or continuity equation can be expressed as:

The continuity equation : $$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \qquad (1)$$

The Navier-Stokes equations:

$$\rho(u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y})=-\frac{\partial p}{\partial x}+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right)+\rho g_x \qquad (2)$$

$$\rho(u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y})=-\frac{\partial p}{\partial y}+\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right)+\rho g_y \tag{3}$$

Where ρ is the density of the fluid and u , v are the x and y velocity components of the flow field and p is the pressure and $\rho\, g_x$ and ρg_y are the gravitational body forces (per unit volume) acting on the fluid.

Boundary Conditions: Boundary conditions were specified at the inlet, outlet and no-slip boundary condition is implied on the walls. As inlet boundary condition the initial velocity (v=10, 15, 20 m/s) of the pipe were specified for the Reynolds number Re= 827, 1241, 1654 respectively. The static pressure at the outlet was set 0 Pa and the other settings that have been made are complied in table 1.

Table no. 1: Material specific settings.

Material	Water-vapor
Density(kg/m^3))	0.5542
Specific heat at constant pressure (J/kg-k)	2014
Conductivity K(w/m-k)	0.0261
Fluid viscosity (kg/m-s)	1.34×10^{-5}

Numerical Method: The control volume method is used to discretize the governing equations. FLUENT solves the governing equations for continuity and Navier-Stokes equations in order to obtain the velocity filed .To do this control volume based technique is used .The pressure based solver was used due to modeling of laminar flow. The continuity and Navier-Stokes equations are solved with first order up-wind scheme to interpolated the corresponding cell centre variables to the faces of the cells. For the pressure-velocity coupling. The SIMPLE (Semi – Implicit Method for Pressure-Linked Equations) algorithm was used for introducing velocity and pressure into continuity and Navier-Stokes equations. The computation are considered to be converged when the residues for continuity and momentum are less than 10^{-6}.

Grid dependency Test: To evaluate the grid size effect, grid dependency tests are carried out. Three different sizes of grid 4×1416, 6×2124 and 8×2832 are tested for U- bend pipe and laminar flow with no slip boundary conditions and the results are listed in the following table. The flowing table-2 shows that the difference of average velocity at outlet for grid sizes 4×1416, 6×2124 and 8×2832 respectively. For convenience, we used grid size 6×2832.

Table no. 2: Grid dependency test of the average velocity under three sizes of grids.

Grid size	Average velocity
10×1416	9.999996
20×2124	10.00005
40×2832	9.999951

Validity and Verification: To valid our model, we simulated 2D steady, laminar flow through U-bend pipe with the boundary conditions as was done by Jason Oakley [8]. We compared velocity distribution with that of Jason Oakley [8] along the horizontal line at X=0.05 to 0.1255 m. The results are displayed in figure.2. The square symbol represents the Jason Oakley [8] results and the solid line represents the results from current work. The results show good agreement which validate our model.

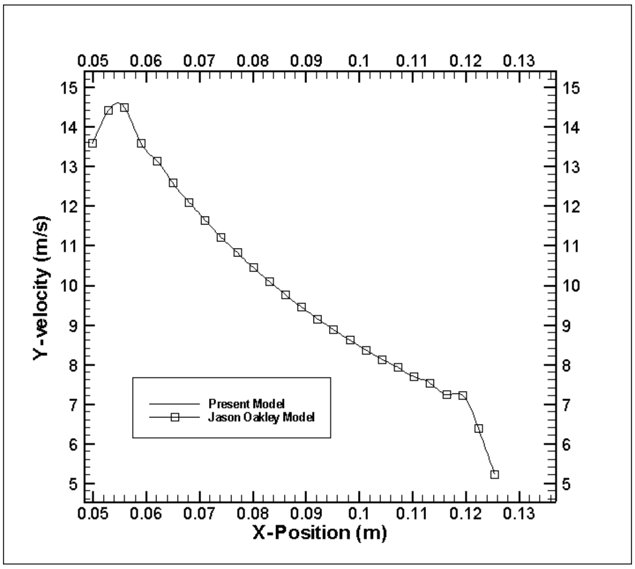


Figure. 2: The velocity distribution along the horizontal line at x=0.05 to 0.1255 m.

III. Result and Discussion

The comparison of the velocity distributions among three cases is shown by figure.3. The velocity distribution is taken along downstream direction for the three cases ranges from x= -0.35 to 0 m, which is positioned in the center line of the horizontal pipe shown in the figure.1. The following figure.3 shows that the velocity gradually increases for three cases (Re=827, 1241, 1654) from the inlet but near the bend (in the position at x=0 m) velocity decreases rapidly because the flow interruption with the right wall or there are stagnation points and at that position the fluid flow changes its direction through the upward section of the bend pipe. Also the mean velocities for Re=827, 1241, 1654 are 13.87991, 20.58417, 27.13529 m/s respectively. Here we see that the mean velocity for Re=1654 is large than the other two cases since Re is proportional to the velocity.

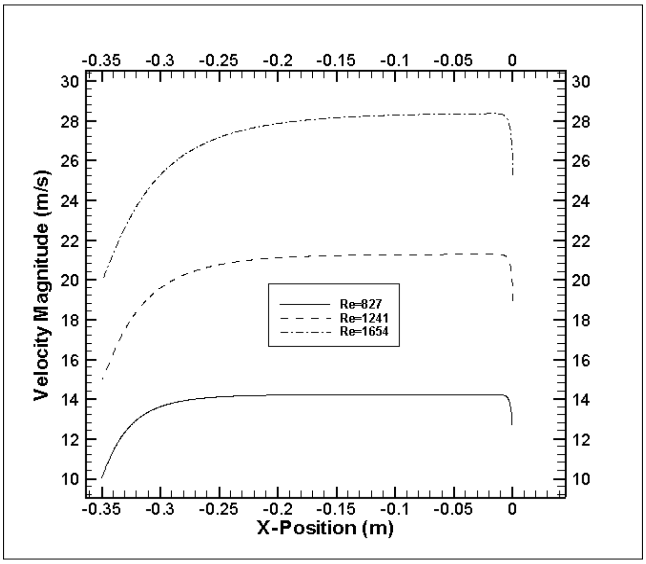


Figure.3: Velocity distributions in the lower portion of the bend pipe from x= -0.35 to 0 m.

The along downstream direction for the three cases on the center line of the upper portion of the pipe from x=0 to -0.35 m. comparison of the velocity distributions among three cases is shown by figure.4. The following figure.4 shows that the velocities for three cases (Re=827, 1241, 1654) near the bend (in the position at x=0 m) rapidly decreases because the flow interruption with the upper wall or there are stagnation points and that position the fluid flow changes its direction through the upward section of the bend pipe but the velocity increases gradually after the bend pipe because there is no stagnation points or flow interruptions so that the fluid flow through the pipe easily. Also the mean velocities for Re=827, 1241, 1654 are 13.8669, 20.53103, 27.02085 m/s respectively. Here we see that the mean velocity for Re=1654 is large than the other two cases since Re is proportional to the velocity.

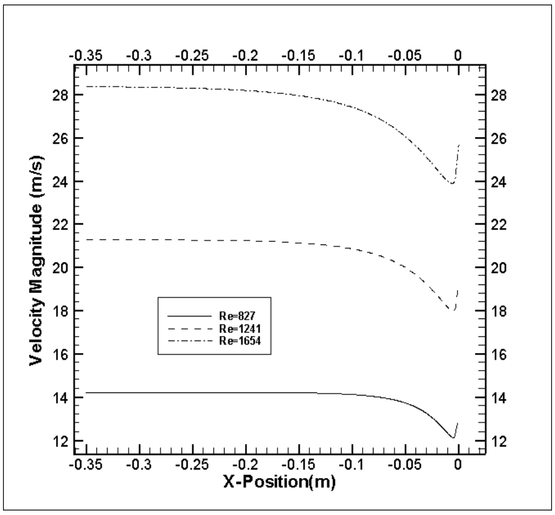
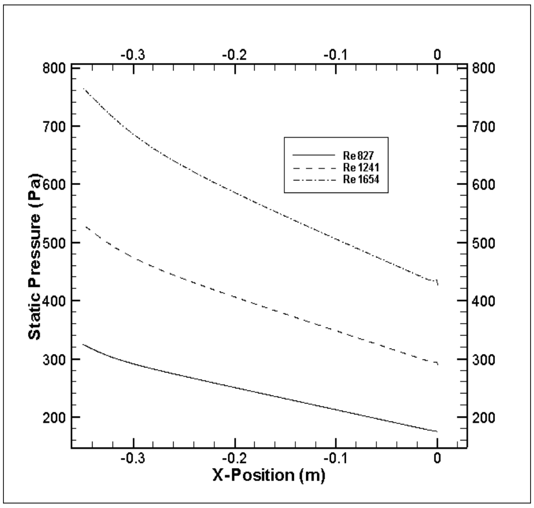


Figure.4: Velocity distributions in the upper portion of the bend pipe from x= 0 to - 0.35 m.



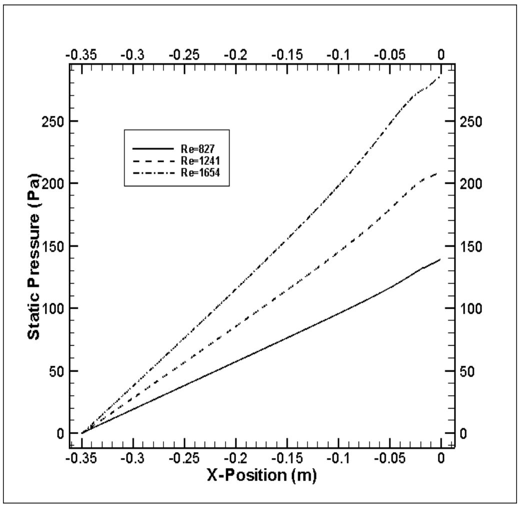
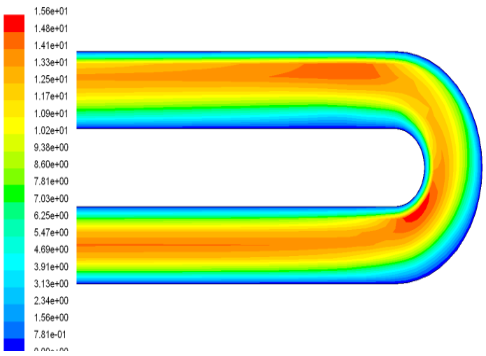


Figure.5: Pressure distributions in the lower portion bend pipe from x= -0.35 to 0 m.

Figire.6: Pressure distributions in the upper the portion of the bend pipe from x= 0 to - 0.35 m.

The comparison of the pressure distributions among three cases are shown by figure.5 and figure.6. The pressure distribution is taken along downstream direction for the three cases in the lower portion of the of bend pipe from x= -0.35 to 0 m and in the upper portion of the of bend pipe from x= 0 to - 0.35 m, which are positioned in the center of the horizontal pipe shown in the figure.1. The above figure.5 and figure.6 show that the highest static pressure are produced in the inlet and after the bend respectively for three cases, which (Pressure distributions) are linear for incompressible flow. The static pressure (average) for three cases are 242.0746, 396.2147, 573.744 and 67.37096, 102.4944, 139.2383 Pa respectively.



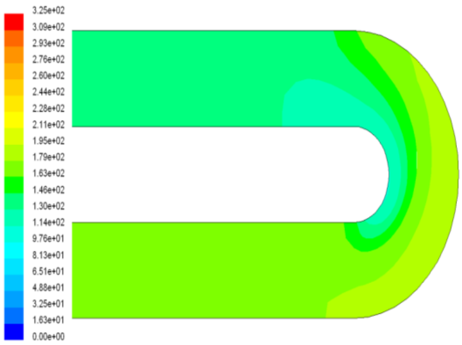


Figure.7: Velocity contours for Re=827

Figure.8: Contours of Static Pressure (Pa) for Re=827

Here we represent the velocity contours close to the bend for three different cases of Reynolds number trying to depict the flow distribution variations. At figure-7 the velocity distribution in the bend pipe is presented at the Reynolds number 827 while at the figure-8 the pressure distribution in the bend pipe. The velocity contour shows a color scheme ranging from 3.13 to 6.25 and 10.2 to 14.1 m/s, in accordance to the inlet and outlet velocity while the pressure reaching from 16.3 to 114 and 211 to 309 Pa respectively. According to the relationship pressure- velocity the distribution of the results seems reasonable and acceptable.

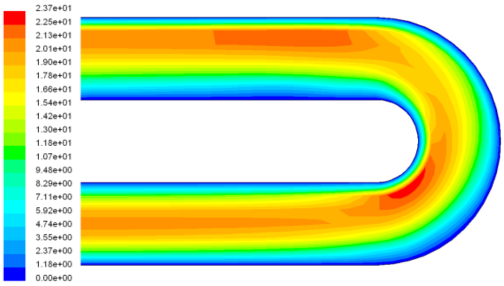


Figure.9: Velocity contours for Re=1241

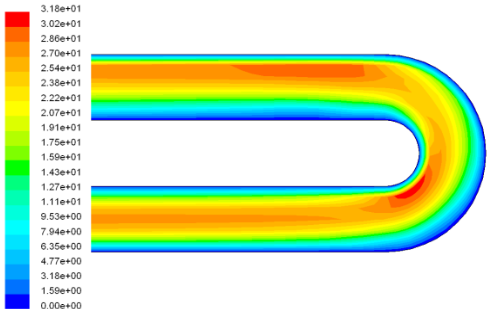


Figure.10: Velocity contours for Re=1654

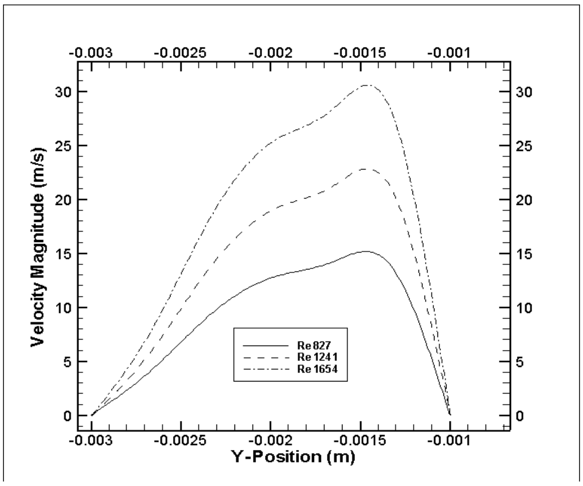


Figure.11: Velocity profiles in the portion of the pipe before the bend starts.

At figure-9 and figure-10 comparison between the velocities distribution along the bend pipe are presented for two different Reynolds numbers, 1241 and 1654. The contours show that with the increase of the Re number and in turn the velocity, the fluid flows with a higher velocity the pipes causing it to deviate after the bend. This is present in the contour showing the results for Re 12410. At Re 1654 the flow develops a region at which recirculation may occur. These are the point where the flow is separated by the wall curved and maximum shear stress is expected. From the above result it can be predicted that in such locations, corrosion is more possible to happen.

Figure.11 shows the velocity profiles in the portion of the pipe before the bend starts for three different values of Reynolds numbers. The influence of the Reynolds number is significant, not only to the velocity maximum values, which is reasonable. From the above figure-11, we see that the velocity profiles are skew parabolic and the velocities increase near the upper wall since there is no flow interruption with the upper wall and low friction while near the lower wall, velocities are low for flow interruption with the right wall and at that position the fluid flow changes its direction through the upward section of the bend pipe. The mean velocities for Re=827, 1241, 1654 are 10.11732, 15.18658 and 20.24596 respectively.

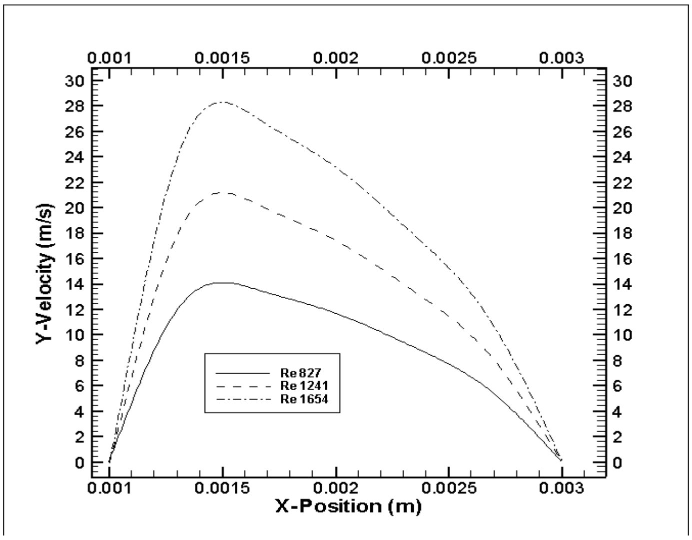


Figure.12: Velocity profiles in the bend portion of the pipe.

Figure.12 velocity profiles in the bend portion of the pipe for three different values of Reynolds numbers. From the above figure-12, we see that the velocity profiles are skew parabolic and the velocities increase near the left wall for no flow interruption with the left wall so that the fluid flow through the pipe easily with large velocity and low viscosity but after x=0.0015 m, velocities decrease gradually near the right wall for flow interruption with the right wall and at that position the fluid flow changes its direction through the upward section of the bend pipe. The mean velocities for Re=827, 1241, 1654 are 10.20608, 15.32872 and 20.45246 respectively.

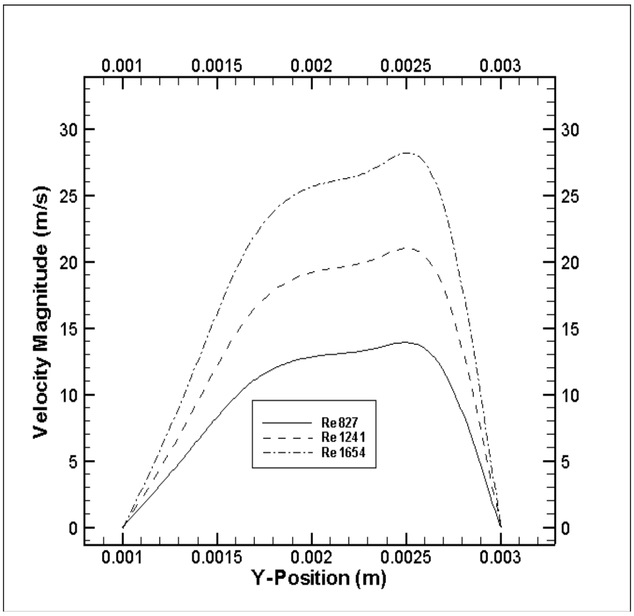


Figure.13: Velocity profiles in the portion of the pipe after the bend ends.

Figure.13 shows the velocity profiles in the portion of the pipe after the bend ends for three different values of Reynolds numbers. The influence of the Reynolds number is significant, not only to the velocity maximum, which is reasonable. From the above figure-13, we see that the velocity profiles are skew parabolic and the velocities increase near the upper wall since there is no flow interruption with the upper wall but after x=0.0025 m, velocities decrease for flow interruption with the lower wall and that position the fluid flow changes its direction through the pipe. The mean velocities for Re=827, 1241, 1654 are 10.108625, 15.50335 and 20.73449 respectively. Here we see that the mean velocity for Re=1654 is large than the other two cases since Re is proportional to the velocity.

Pressure Drop within the Bend:

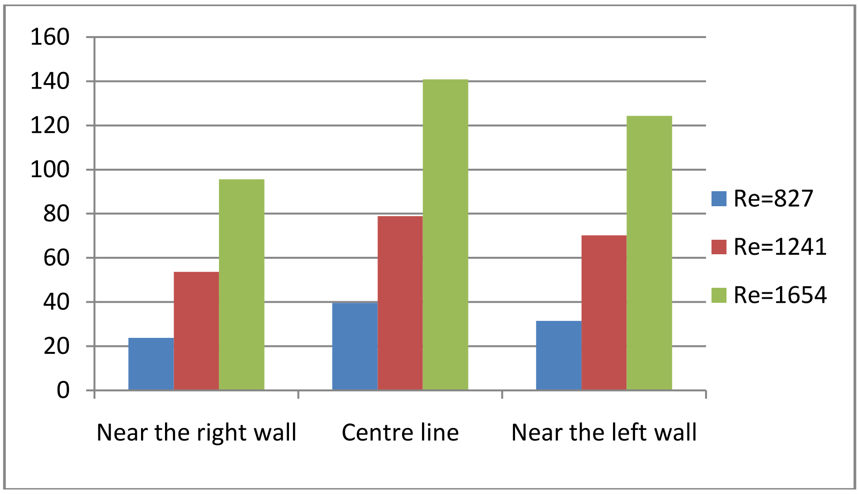


Figure.14: Pressure drop comparison along the bend for three cases.

Figure-14 represents the comparison of pressure drop within the bend region for three cases. The pressure drop is taken for three different positions along the bend at near the right wall, centre line and near the left wall for three cases (Re= 827, 1241, 1654). From above figure-14 we see that the large pressure drop in the centre line along the bend for three cases than near the right and left wall since the large velocity occurs at the centre line of the bend and the large velocity produces large pressure drop in the bend pipe. Also we see that the large pressure drop occurs for the Re=1654 than the other two cases since Re is proportional to the velocity.

IV. Conclusion

Also the numerical analysis and estimation was conducted for fluid flow through a U-bend pipe. The analysis has been conducted on three different Re values (827, 1241and 1654) which in turn impacted the inlet velocity of the fluid (10, 15 and 20 m/s). We observed that within the lower portion of the pipe along the downstream direction velocity gradually increases from the inlet but velocity decreases rapidly near the bend because of flow interruption with the right wall. Along the upper portion of the pipe velocities for three cases (Re=827, 1241, 1654) near the bend is lower and increases gradually along the downstream direction. Also the cross wise velocity profiles are skew-parabolic before the bend starts, in the bend portion and after the bend. The irregularities of velocity increases as the Re value is increased and the large pressure drop as well as large velocity occurred as well on the centre line compared to any other portion of the pipe for three cases. Also the pressure distributions are linear within the upper and lower portion of the pipe. The present findings may be useful to understand the flow characteristics inside the bend pipes at any pumping stations and water refineries.

References

[1]. P. H. M. Bovengeerd, A. A. V. Steenhoven, F. N. D. Vosse and G. Vossers, " Steady entry flow in curved pipe," Journal of Fluid Mechanics, vol.177, pp.233-246, 1978..

[2]. H. Nakayama, M. Hirota, H. Fujita, T. Yamada, and Y. Koide, " Flow characteristics in rectangular ducts with a sharp 180 degree turn,". Transactions of the Japan Society of Mechanical Engineers, vol. 69(681), pp 1171-1179, 2003.

[3]. M. Anwer, R. M. C. So, and Y. G. Lai, "Perturbation by and Recovery from Bend Curvature of fully Developed Laminar and Turbulent Pipe Flow," J. Phys. Fluids A, vol.1, no.8, pp. 1387-1397, 1989.

[4]. J. Azzola, J. A. C. Humphrey, H. Iacovides, and B. E. Launder, "Developing Laminar Flow in U-Bend of Circular Cross Section: Measurement and Computation," Journal of Fluids Engineering, vol. 108. pp. 214-221, 1986.

[5]. H. Iacovides, B. Launder, P. A. Loizou and H. H. Zhao, "Boundary- Layer Development Around a Square-Sectioned U-Bend : Measurements and Computation," ASME J. Fluids Eng., vol. 113, pp. 409-415, 1990.

[6]. K. Sudo, M. Sumida and H. Hibara, "Experimental Investigation on Laminar Flow in a a Square-Sectioned 180-Degree Bend," Exp. Fluids, vol. 30, no.3, pp. 243-252, 2001.

[7]. F. Ducret, "Studies of Sound Generation and Propagation in Flow Ducts," The Royal Institute of Technology, 2006.

[8]. Jason Oakley, "Pipe U-Bend FLUENT Example," J. Spring , 2007.

[9]. M. A. Cade, W. C. P. B. Lima, S. R. Farias Neto and A. G. B. Lima, "A Numerical Study of Natural Gas Laminar Flow in Elliptic Cylindrical Pipes," *Brazilian Journal of Petroleum and Gas*, vol. 4, no. 1, pp. 19-33, 2010.

[10]. M. Yazdi and A. Bardi, "Estimation of Friction Factor in Pipe Flow Using Artificial Neural Networks," *Canadian Journal on Automation, Control & Intelligent Systems*, vol. 2, no. 4, pp. 52-56, 2011.

[11]. A. Nouri-Borujerdi and M. Layeghi, "A Numerical Analysis of Vapor Flow in Concentric Annular Heat Pipes," *Transactions of the ASME, Journal of Fluids Engineering*, vol. 126, pp. 442-448, 2004.

[12]. A. Mussa, P. Asinari and L.S. Luo, "Lattice Boltzmann Simulations of 2D Laminar Flows past Two Tandem Cylinders," Elsevier Science, New York, 2008.

[13]. M. S. Ghidaoui and A. A. Kolyshkin, "A Quasi-Steady Approach to the Instability of Time-Dependent Flows in Pipes," *Journal of Fluid Mechanics*, vol. 465, pp. 301-330, 2002.

[14]. M. Bahrami, M. M. Yovanovich and J. R. Culham, "Pressure Drop of Fully Developed, Laminar Flow in Micro-Channels of Arbitrary Cross-Section," *Proceedings of ICMM* 2005 3*rd International Conference on Micro-channels and Mini channels*, Toronto, 13-15 June 2005.

[15]. Z. G. Makukula, P. Sibanda and S. S. Motsa, "A Novel Numerical Technique for Two-Dimensional Laminar Flow between Two Moving Porous Walls," *Hindawi Publishing Corporation Mathematical Problems in Engineering*, vol. 2010, 2010, Article ID: 528956.

[16]. M. S. Favero and K. R. Berquist, "Use of Laminar Air-Flow Equipment in Microbiology," *Applied Microbiology Copyright @* 1968 *American Society for Microbiology*, vol. 16, no. 1, pp. 182-183, 1968.

[17]. D. G. Fox, "A Study of the Application of Laminar Flow. Ventilation to Operating Rooms," *Public Health Mono-graph No*. 78, US Government Printing Office, Washington, 1969.

[18]. H. D. Chen, L. Wen, S. Y. Zheng, H. Gao, B. Xiong, H. N. Bian, Z. A. Liu and L. J. Wel, "Application of Laminar Air Flow Techniques in Burn Treatment," Department of Burn Surgery, People's Hospital of Guangdong Province, Guangzhou, 2005.

[19]. N. M. Crawford, G. Cunningham and P. L. Spendding, "Prediction of pressure drop for laminar flow in 90 degree bends," Proceedings of the institution of mechanical engineers, vol. 218, no. 3, pp. 153-155, 2003.

[20]. P. L. Spendding, E. Benared and G. M. McNally, "Fluid Flow through 90 degree Bends," Developments in Chemical Engineering and Mineral Processing, vol. 12, no. 1-2, pp. 107-128, 2004.

[21]. P. Louda , K. Kozel, J. Prihoda, J. Benes and T. Kopacek , " Numerical solution of incompressible flow through branched channels," vol. 46, pp. 318-326, 2011.

[22]. H. Phan, L.Wen, H. Zhang," Numerical simulation and analysis of gas liquid flow in a T-junction micro channel," Advances in Mechanical Engineering, (2012), ID 231675.

[23]. H. Shamloo, B. Pirzadeh, "Investigation of characteristics of separation zones in T-junction channels," WSEAS transactions on Mathematics, vol. 7, pp. 303-312, 2012.

[24]. J. Marn, P. Ternik, "Laminar flow of a shear thickening fluid in a 90o pipe bend", Fluid Dynamics Research, vol. 38, pp. 295-312, 2006.

[25]. H. Ito "Pressure losses in smooth pipe bends," J. Basic Engineering, vol. 82, pp. 131–143, 1960.

[26]. M. M. Enayet, M. M. Gibson, A. M. K. P. Taylor, "Laser Doppler measurements of laminar and turbulent flow in a pipe bend," Int. J. Heat Fluid, vol. 3, pp. 213–219, 1982.

[27]. Van de, F. N. Vosse, A. A. Steenhoven, A. Segal A, " A finite element analysis of the steady laminar entrance flow in a 90 degree curved tube," Int. J. Number Math, vol. 9, pp. 275–287, 1989.

[28]. P. L. Spedding, E. Bernard and G. M. McNally, "Fluid flow through 90 Degree Bends," Asia-Pac J. Chem Eng, vol. 12, pp. 107–128, 2004.

[29]. J. Marn and P. Ternik, " Laminar flow of a shear-thickening fluid in a 90 degree pipe bend," Fluid Dynamic Research, vol.38, pp. 295–312, 2006.

[30]. M. R. Hajmohammadi, H. EskandarI, M.Saffar-Avval, et al. "A new configuration of bend tubes for compound optimization of heat and fluid flow," Energy 2013; 62:418–424.

[31]. V. M. Soundalgekar and V. G. Divekar, "Laminar Flow through pipe bend with Small Suction," Publication de l'Institut Mathematique, Nouvelle Serie, Vol. 16, No. 30, pp. 147-157, 1993.

[32]. M. Toda, N. Kamori, S. Saito, S. Maeda, "Hydraulic conveying of solids through pipe bends," J. Chem. Eng. Jpn. Vol. 5, pp. 4–13, 1972.

[33]. W. R. Dean, " Note on the motion of fluid in a curved pipe," The London, Edinburgh, and Dublin Philosophical Magazine and Journal of Science, Series 7, PP. 208-223, 1927.

[34]. A. Alizadehdakhel, M. Rahimi and J. Sanjari, "CFD and artificial neural network modeling of two-phase flow pressure drop," *International Communications in Heat and Mass Transfer*, vol.8, no.36, pp**.** 850-856, 2009.

[35]. D. Bhandari and Dr. S. Singh "Analysis of fully developed laminar flow in a pipe bend using computational fluid dynamics," International Journal of Engineering Research and Technology, vol. 1, no. 5, pp. 1-8, 2012.

[36]. M. Sahu, Kishanjit Kumar Khatua, Kanhu Charan Patra and T. Naik, "Developed laminar flow in pipe using computational fluid dynamics," International R &D Conference on Development and Management of Water and Energy Resources, Bhubaneswar, India, 4-6 February 2009.

[37]. P. Dutta, N. Nandi, "Study on pressure drop characteristics of single phase laminar flow in pipe bend for Reynolds number," ARPN J. Eng. Appl. Sci., vol.4, pp.2221–2226, 2004.

[38]. Christina G. Georgantopoulou, Ahmed Khaled Mohamed Khan, Nikolaos S. Vasilikos and George A. Georgantopoulos "Newtonian Flow Modeling through 90 Degree Pipe Bends," vol.11, no.16, 2016.

[39]. V. M. Soundalgekar and V. G. Divekar, "Laminar Flow through a Uniform Circular Pipe with Small Suction," *Publication de l'Institut Mathematique, Nouvelle Serie*, vol. 16, no. 30, pp. 147-157, 1993.

[40]. T. Zhao and P. Cheng, "A Numerical Solution of Laminar Forced Convection in a Heated Pipe Subjected to a Reciprocating Flow," *International Journal Heat Mass Transfer*, vol. 38, no. 16, pp. 3011-3022, 1995.

[41]. S. K. Karode, "Laminar Flow in Channels with Porous Walls, Revisited," *Journal of Membrane Science*, vol. 191, no. 1, pp. 237-241, 2001.

[42]. M. Rowe, " Measurements and Computations of Flow in Pipe Bends, " J. Fluid Mech., vol. 43, no. 4, pp. 771-783, 1970.

[43]. A. Nouri and A. Nabovati, "Numerical Study of Viscous Dissipation in Circular Microchannels," 8*th International and* 12*th Annual Mechanical Engineering Conference*, Tarbiat Modarres University, Tehran, 2008.

[44]. S. M. Change, J. A. C. Humphrey, and A. Modavi, "Laminar Flow in Strongly Curved U-bend and Downstream Tangent of Square Cross-Sections," Physicochem. Hydrondyn. , vol. 4, no 3, pp. 243-269, 1983.

[45]. W, J. Federspie and I. Valenti, "Laminar Flow in Micro Fabricated Channels with Partial Semi-Circular Profiles," *Open Journal of Applied Sciences*, vol. 2, No. 1, pp. 28-34, 2012.

[46]. C. Munch and O. Metais, "Large eddy simulations in curved square ducts: variation of the curvature radius," J. Turbul., vol. 8, no 29. pp. 1-18, 2007.

[47]. R. Yang and S. F. Chang, "A Numerical Study of Fully Developed Laminar Flow and Heat Transfer in a Curved Pipe with Arbitrary Curvature Ratio," *International Journal of Heat and Fluid Flow*, vol. 14, no. 2, pp. 138-145, 1993.

[48]. K. Suga, "Predicting laminar flow in 2-D curved ducts by near wall second moment closures," ASME J. Fluids Eng., vol. 46, no. 11, pp. 161-173, 2003.

[49]. P. Batten, T. J. Craft, M. A. Leschziner and H. Loyau, "Reynolds-Stress Transport Modeling for Compressible Aerodynamics Application," AAIA J., vol. 37, no. 7, pp. 785-79, 1999.

[50]. I. Sarbu and A. Iosif, "Numerical Analysis of the Laminar Forced Heat Convection between Two Coaxial Cylinders," *International Journal of Energy*, vol. 3, no. 4, 2009.
[51]. N. Yucel and N. Dinler, "Numerical Study on Laminar and Turbulent Flow through a Pipe with Fins Attached," *An International Journal of Computation and Methodology*, vol. 49, no. 2, pp. 195-214, 2006.

Md. Ismail Hossain. "Numerical Study of Water Flow through U-Bend Pipe." IOSR Journal of Mathematics (IOSR-JM) 15.4 (2019): 48-57