Published by :
http://www.ijert.org

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 7 Issue 08, August-2018

Hydrodynamic Flow Analysis across bent sections of Pipe using CFD Software

Rahul Pandey[1], Ankit Kumar Shukla[2], Vinay Sati[3], Shivasheesh Kaushik[4], Dr. Anirudh Gupta[5]
[1&2]Scholar, B.Tech, Department of Mechanical Engineering,
Amrapali Institute of Technology And Science, Haldwani, Uttarakhand, India.
[3&4]Assistant Professor, Department of Mechanical Engineering, AITS Haldwani, Uttarakhand, India.
[5]Associate Professor, Department of Mechanical Engineering, BTKIT Dwarahat, Uttarakhand, India

***Abstract* - The energy losses occur in a flowing fluid whenever there is a change in the path of flow. In this paper an approach has been made to reduce the energy losses occurring at bends of a pipe containing a flowing fluid. Velocity and Pressure are the parameters on which have been our areas of primary interest. All the calculations and simulations have been done on the 2-D axis-symmetric sketches of pipe models using ANSYS R16.0. The fluid being considered is water.**

Keywords: K-Epsilon, Naiver-Stokes equation, FLUENT Flow, fillet

.

I. INTRODUCTION

Fluid has a tendency to flow in layers, as the layer distance increase from the surface of flow the higher becomes the velocity gradient ($\frac{\partial v}{\partial x}$). Conversely with the decrease in layer distance from the surface leads to a rise in pressure gradient ($\frac{\partial p}{\partial x}$). Also just at the surface of the body where the layer is in contact with it, due to its low velocity and relatively high pressure gradient a situation is created as such that the layer moves at a relatively slow speed as compared to other layers present, known as the no-slip condition. The energy which is useful for transformation lies in the free stream region and when the fluid flows over a bend which changes its direction of flow then the width of the boundary layer starts increasing leading to a lesser free stream region with a reduced energy. Such losses can be rectified by providing smooth curved surfaces which bend the path of flow with a lesser energy loss.

Let $\mho_{net}$ be the resultant velocity of fluid in 2-D then,

$$\mho_{net} = \mho_{drag} \cos\Theta + \mho_{lift} \sin\Theta \qquad \text{---(1.1)}$$

In ideal condition the drag and lift component both will be equal and hence the energy loss is minimum in such case. In this paper we have removed the negative pressure zones and thus rectified the problem of back flow/reverse flow in the pipe which thus in leads to lesser variation between input and output parametric quantities. Also the drag effect is reduced using the fillet which is proved in this paper.

II. MATHEMATICAL MODELLING

There are two models used for this analysis, the first one being a pipe bent at 90 degrees sharply and the other model is a pipe bent at 90 degree using 2 fillets. Both the pipes are of same dimensions as described and the direction of flow is also the same.

We will be using named references further for these pipes, the first pipe described be referred to as T-90 and the second one described is F-90. For the geometric modelling both have been constructed taking zero tolerance.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 7 Issue 08, August-2018

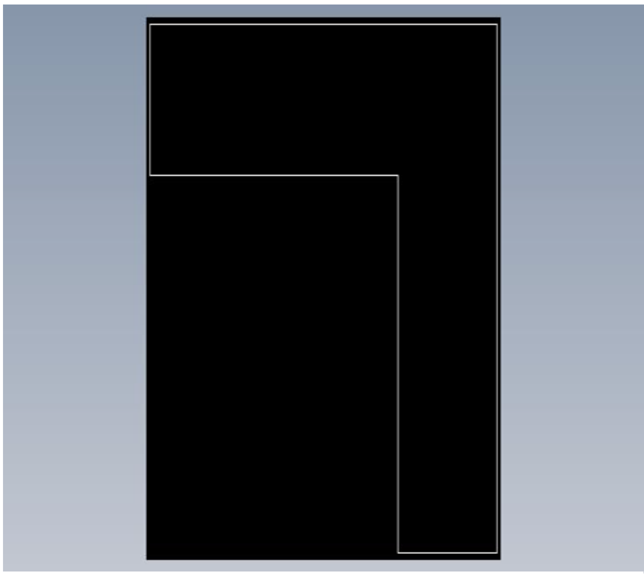

Figure- T-90

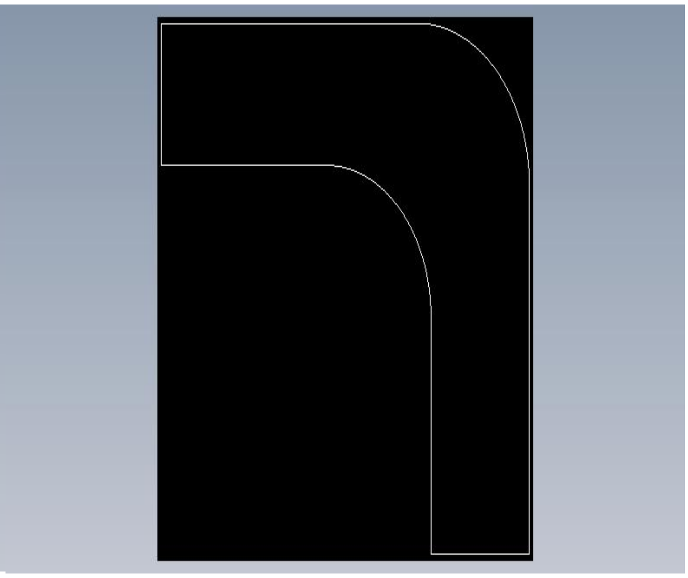

Figure- F-90

The specifications of T-90 are as follows: -

1. Diameter= 0.80 m
2. Base length= 2.00 m
3. Boundary length= 11.2 m
4. Surface Area= 3.84 m^2

Similarly, the specifications of F-90 are as follows: -

1. Diameter= 0.80 m
2. Base length= 1.35 m
3. Boundary length = 11.03 m
4. Arc lengths= 1.23,1.30 m

As evident from the above values, both have the same discharge capacity although having varying geometries. Since all the forces due to turbulence are neglected we have done the simulation by applying Naiver-Stokes equation and K-Epsilon model.

The fluid from region 1 to region 2 as shown on the flow diagram, considering that no mass has been stored thus rate of mass inflow=rate of mass outflow. Applying momentum integral and continuity equations at these regions-

Continuity Equation

$$\frac{\delta(\rho \text{ʉ})}{\delta x} + \frac{1}{r}\frac{\delta(\rho r\vartheta)}{\delta r} \qquad \text{---(1.2)}$$

Momentum Equations:

Axial component (z-component)

$$\rho\vartheta\left[\frac{\delta \text{ʉ}}{\delta r} + \text{ʉ}\frac{\delta \text{ʉ}}{\delta x}\right] = \frac{\delta p}{\delta x} + \frac{\delta}{\delta x}\left(\mu\frac{\delta \text{ʉ}}{\delta x}\right) + \frac{1}{r}\frac{\delta}{\delta r}\left(\mu\frac{\delta \text{ʉ}}{\delta r}\right) + \frac{\delta}{\delta x}\left(\mu\frac{\delta \text{ʉ}}{\delta x}\right) + \frac{1}{r}\frac{\delta}{\delta r}\left(\mu\frac{\delta \text{ʉ}}{\delta r}\right) \qquad \text{---(1.3)}$$

Radial component(r-component)

$$\rho\left[\vartheta\frac{\delta y}{\delta r} + \text{ʉ}\frac{\delta y}{\delta x}\right] = -\frac{\delta p}{\delta z} + \frac{\delta}{\delta x}\left(\mu\frac{\delta y}{\delta x}\right) + \frac{1}{r}\frac{\delta}{\delta r}\left(r\mu\frac{\delta r}{\delta r}\right) + \frac{\delta}{\delta x}\left(\mu\frac{\delta \text{ʉ}}{\delta r}\right) + \frac{1}{r}\frac{\delta}{\delta r}\left(r\mu\frac{\delta\vartheta}{\delta r}\right) - 2\mu\frac{\vartheta}{r2} + \rho\frac{w2}{r} \qquad \text{---(1.4)}$$

Tangential component (Ө-component)

$$\rho\left[\vartheta\frac{\delta p}{\delta r} + \text{ʉ}\frac{\delta p}{\delta x}\right] = \frac{\delta}{\delta x}\left[\mu\frac{\delta p}{\delta x}\right] + \frac{1}{r}\frac{\delta}{\delta r}\left[r\mu\frac{\delta p}{\delta r}\right] - \frac{2}{r}\frac{\delta}{\delta r}[\mu\phi] \qquad \text{---(1.5)}$$

Here ʉ,ϑ,w are the velocity components along z, r, θ directions respectively and the variables ϕ=rw and $\mu=\mu_{eff.}$

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 7 Issue 08, August-2018

III. INPUT PARAMETERS

Analysis of the flow conditions was done using ANSYS Fluent 16.0 CFD Software. These equations are solved by converting complex partial equation into simple algebraic equation. Both the geometries T & F 90 are 2-D rigid solids comprising of 1 region 3 named selections as inlet, outlet, and path. These 2-D geometries were used for solving momentum and energy equations. The initial phase velocity of the flow was defined at the inlet section of the pipe upstream. Acceleration due to gravity has been taken as 9.81 m^2/sec. The standard wall functions were applied to the κ-ε turbulence models for solving the problems. The initial velocity given to both the T & F 90 geometries were 1 m/sec.

IV. RESULTS AND DISCUSSION

Keeping the inlet velocity constant at 1 m/sec in both cases we get the following data:

1. Inlet velocities

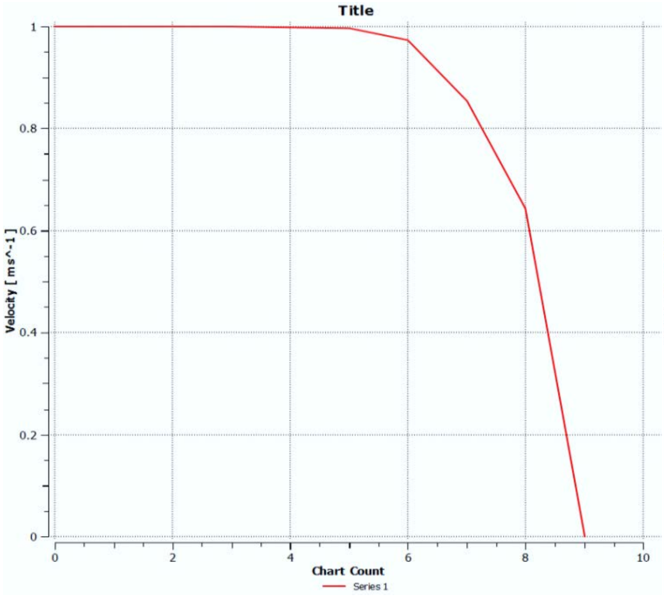

Figure- inlet velocity at T-90

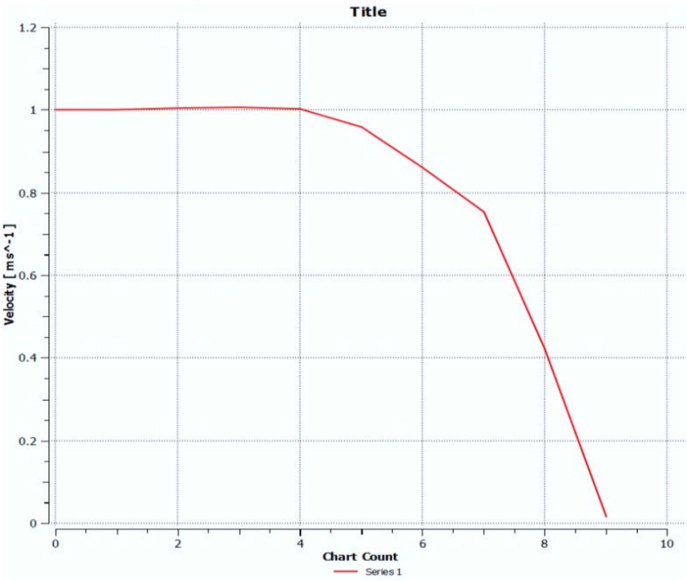

Figure- Inlet velocity at F-90

2. Outlet velocities

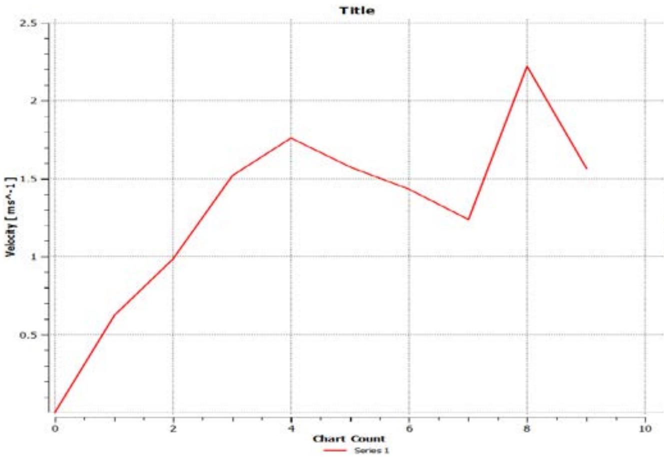

Figure- Outlet Velocity at T-90

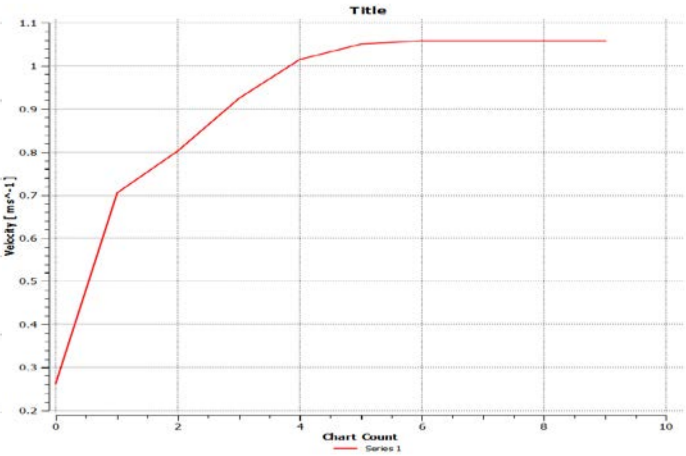

Figure- Outlet velocity at F-90

3. Pressure at inlet

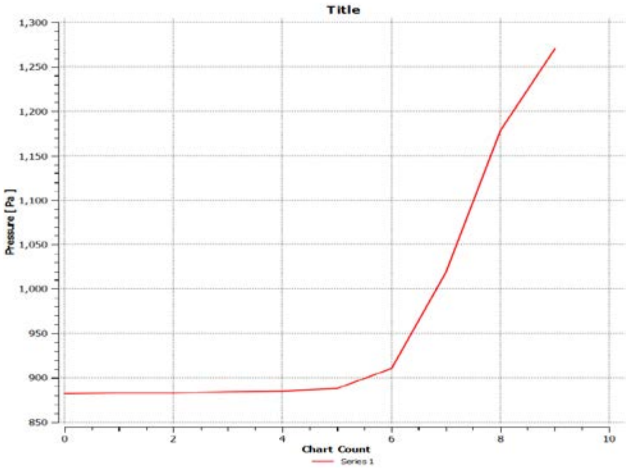

Figure- Inlet pressure at T-90

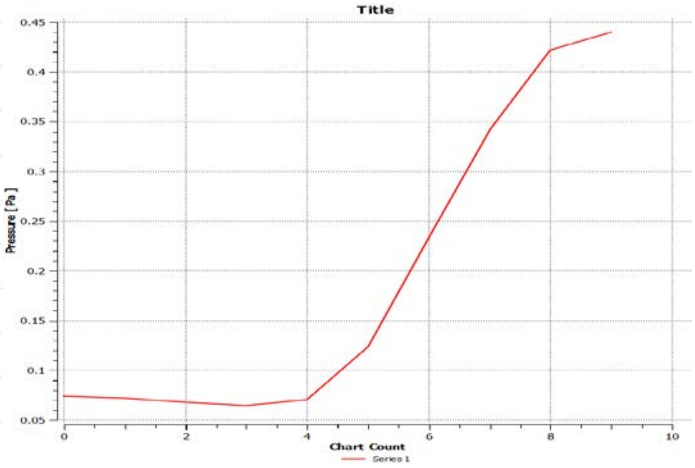

Figure- Inlet pressure at F-90

4. Pressure at outlet

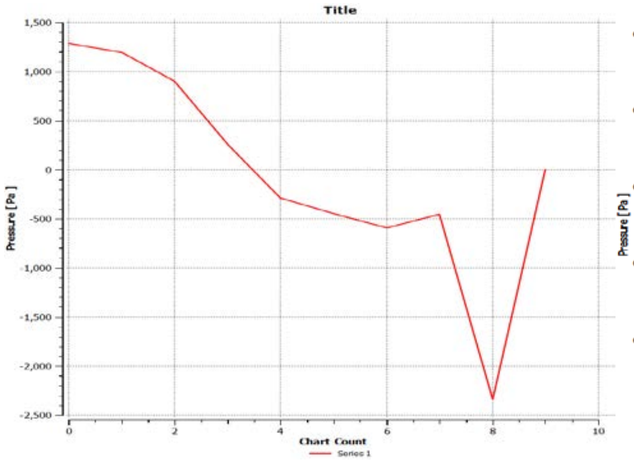

Figure- Outlet pressure at T-90

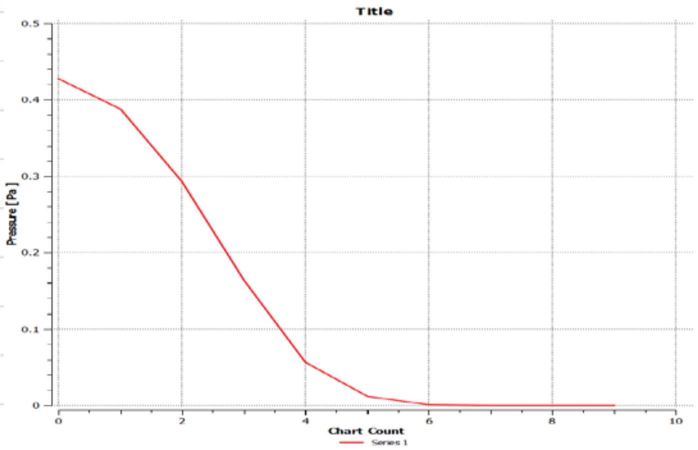

Figure- Outlet pressure at F-90

5. Pressure Contours

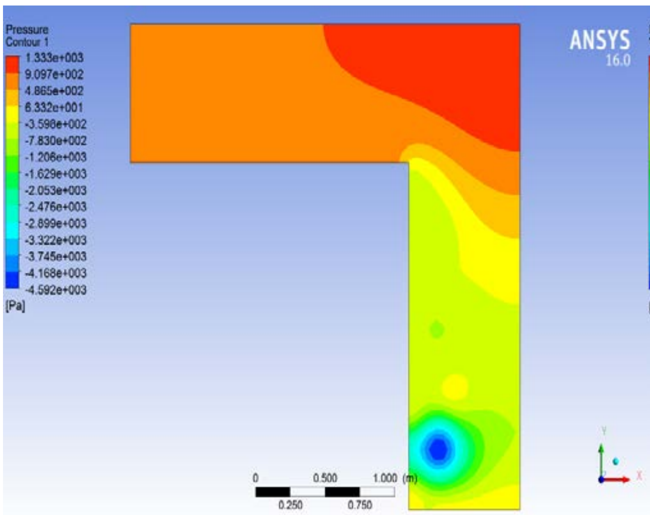

Figure- Pressure Contour for T-90

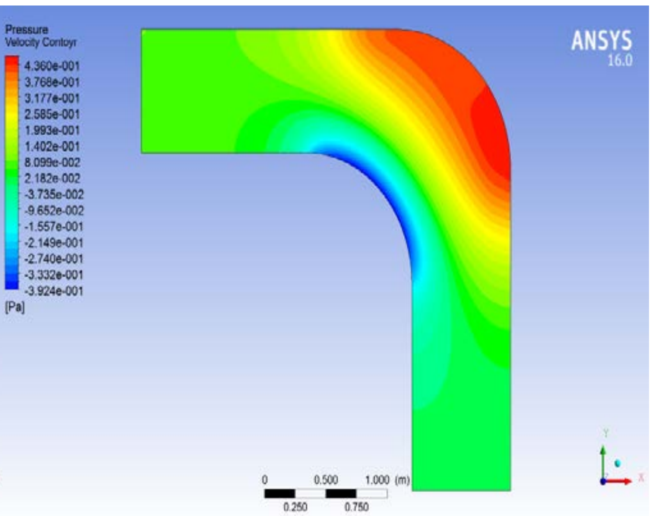

Figure- Pressure Contour for F-90

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 7 Issue 08, August-2018

Discussion

Providing a fillet in the bend course we observe the cavitation is reduced significantly as compared to direct bend in the pipe course. As cavitation is removed also the case of back flow at the bend region is removed and the continuity equation remains valid by a greater degree as compared to that in the 90-degree bend. The energy losses as observed are also reduced significantly by providing the fillets. The flow tends to greater degree of continuity when fillet is employed as evident from the above mentioned results.

V. CONCLUSION

We must employ fillets while providing bends in the course of the pipeline layout to reduce the energy losses and hence forth increase our I/O ratio, from the above results we can interpret that providing a smooth curved path results in greater efficiency as compared to sharp edged bends. These results were obtained using the ANSYS Fluent 16.0 CFD Software. In practice there will be a variation in the above calculated parameters by a certain factor, as these above calculations were done for an ideal case of flow, but the conclusion shall remain the same that curved path is better than sharp edged path if high velocity is required at the output of the section.

VI. REFERENCES

[1] S K Som, Suman Chakraborty- “Introduction to fluid mechanics and fluid machines” 3e, McGraw Hill International- ISBN 978-0-07-132919-4

[2] Yunus A. Cengel, John M. Cimbala- “Fluid Mechanics Fundamentals and Applications” 3e, McGraw Hill International-ISBN 978-93-392-0465-5

[3] Fox R. W., A. T. McDonald, and P. J. Pritchard, “Introduction to Fluid Mechanics”, 5e, John Wiley & Sons, Inc., 2004.

[4] Laufer, J., “The structure of turbulence in fully developed pipe flow”, NACA Report, NACA-TN-2954 (1953).

[5] Vinay Sati et al, “Hydrodynamic and Thermal Analysis in Pipe Flow using ANSYS Software”, International Journal of Research and Technology, ISSN: 2278-0181, March 2015.