Research Article *ISSN: 2319-507X*
Bhadange PJ, IJPRET, 2013; Volume 1(8): 224-233 *IJPRET*

INTERNATIONAL JOURNAL OF PURE AND APPLIED RESEARCH IN ENGINEERING AND TECHNOLOGY

A PATH FOR HORIZING YOUR INNOVATIVE WORK

CFD ANALYSIS OF FLUID FLOW THROUGH PIPE FITTING

P.J.BHADANGE[1], DR. S. B. THAKRE[2]

1. P.G. Student Mechanical Engineering (CAD-CAM), Prof. Ram Meghe Institute of Technology and Research, Badnera, Amravati
2. Professor, Department of Mechanical Engineering, Prof. Ram Meghe Institute of Technology and Research, Badnera, Amravati.

Accepted Date:
27/02/2013

Publish Date:
01/04/2013

Keywords

CFD Analysis

Pipelines

Corresponding Author

Mr. P. J. Bhadange

Abstract

Pipes fitting are used for piping in industry for fluid flow such as liquid, gasses. Present work will be carried out to minimize the losses in piping which are encountered due to pipe fittings by suggesting exact pipe fittings for particular flow range in order to minimize energy losses and provided cost effective solution. Pipelines are normally designed to deliver fluid at the required head and flow rate in a cost effective manner. Increase in conduit diameter leads to increase in annual capital costs, and decrease in operating costs. Selection of an optimum conduit diameter for a particular fluid flow will therefore be a vital economic decision. This paper presents a use of computational fluid dynamics (CFD) for determination of optimum pipe diameter for a number of idealized turbulent flow. Relationships were formulated connecting theories of turbulent fluid flow with pipeline costing.

Introduction:

Pressure losses in pipe bends have been measured by a number of investigators and widely vary results have been reported. This is probably due not merely to their use of differing inlet velocity distributions and also to the conditions at the outlet of the pipe.

Proper consideration in material selection and pipe sizing will not be developed without proper pipeline design calculation methods. There might be different calculation methods for each industry based on their necessity, but generally it's all derived based on commonly used piping design theory. For pipeline calculations, some commonly used theories are Bernoulli's Theorem and Darcy's Formula, which are also used for piping calculation in this guideline. Bernoulli's Theorem is used for general energy equation and pressure measurement. Meanwhile, Darcy's Formula is used for friction loss in fluid flow general equation and also as basic principle for compressible flow in pipe equation. There are interconnect points need to be considered as well. They are usually includes in piping calculation. Hence this guideline covers calculation in those piping-related equipment's such as elbow and bend. Since it is important to get piping design calculation concept, herewith some applications and sample calculations, which is generally used in industrial, is attached in this guideline to make an engineer easier to understand and calculate an actual design of pipeline. The first group of experiments reported here was designed to study the important effect on the buildup of pressure losses of the several of the secondary circulation at the bend outlet. A three pipe was used first to determine the losses due to bend, elbow with different diameter. Three such systems were used to investigate the difference in the buildup of pressure losses in both the bend itself and in the downstream transition region. The results are compared with losses calculated for the GI pipe. All the tests were made at approximately the same Reynolds number with approximately the same inlet velocity profiles, and in each case the downstream transition length was sufficient for a nearly symmetrical velocity distribution to be attained at the pipe outlet.

Literature Review –

Principal -Change in flow velocity due to change in the geometry of a pipe system (i.e., change in cross-section, bends, and other pipe fittings) sets up eddies in the flow resulting in Energy losses.

Resistance to flow in a pipe -

When a fluid flows through a pipe, the internal roughness of the pipe wall can create local eddy currents within the fluid adding a resistance to flow of the fluid. The velocity profile in a pipe will show that the fluid elements in the center of the pipe will move at a higher speed than those closer to the wall. . Therefore friction will occur between layers within the fluid. This movement of fluid elements relative to each other is associated with pressure drop, called frictional losses. Pipes with smooth walls such as glass, copper, brass and polyethylene have only a small effect on the frictional resistance. Pipes with less smooth walls such as concrete, cast iron and steel will create larger eddy currents which will sometimes have a significant effect on the frictional resistance. Rougher the inner wall of the pipe more will be the pressure loss due to friction.

As the average velocity increases, pressure losses increase. Velocity is directly related to flow rate.

Velocity=Volumetric flow rate /Cross sectional area of the pipe

An increase or decrease in flow rate will result in a corresponding increase or decrease in velocity. Smaller pipe causes a greater proportion of the liquid to be in contact with the pipe, which creates friction. Pipe size also affects velocity. Given a constant flow rate, decreasing pipe size increases the velocity, which increases friction. The friction losses are cumulative as the fluid travels through the length of pipe. The greater the distance, the greater the friction losses will be. Fluids with a high viscosity will flow more slowly and will generally not support eddy currents and therefore the internal roughness of the pipe will have no effect on the frictional resistance.

Flow in Bent Circular Pipes

In a given system consisting of a circular pipe of diameter d,bent on a circular arc of radius R, with a length of straight pipe of thesame diameter downstream of the bend, three flow regions may be distinguished.Firstly there is the inlet transition region in which the inertia forces are more important than those due to viscosity. The main effect here is the generation of a component of vorticity ξ in the direction of flow. Using an in viscid fluid theory Hawthorne (1955) shows how, when the inlet velocity profile is linear, to a first approximation the effect of the secondary flow is to rotate the entire streamline pattern about the axis of the pipe. Having previously made this assumption Hawthorne (1951) showed that the nature of the secondary flow is oscillatory, the angular displacement a being given approximately by the equation:

$\frac{d}{R}\frac{d^2\alpha}{d\phi^2} = \cos\alpha$ ....where ϕ is the bend deflection.

The energy balance between two points in a pipe can be described by the Bernoulli equation, given by

$$\frac{p_1}{\gamma} + z_1 + \frac{V_1^2}{2g} = \frac{p_2}{\gamma} + z_2 + \frac{V_2^2}{2g} + h_L$$

Where– P_1 is static pressure (in Pa) at point 1, g is specific weight of the fluid (in N/m^3), Z_1 is the elevation (in meters) of point 1, V_1 is the fluid velocity (in m/s) at point 1, g is the gravitational constant (in m/s^2), and h_L is head loss (in meters).

Determination of Pressure Losses

Consider a system in which the bend deflection is ϕ and the downstream transition length is l. The overall losses in the system may be calculated from the difference between the values of total pressure at bend inlet and pipe outlet. A mass averaged stagnation pressure H may be defined by the integral

$$H = \frac{4}{\pi q d^3}\int_0^{d/2}\int_0^{2\pi} hqrdrd\theta$$

where r and 0 are polar co-ordinates in the cross-section of the pipe , h is the stagnation pressure at the point (r,Θ), q is the velocity at the point (r, Θ),q is the mean velocity.At the bend inlet, assuming that the straight entry segment is long enough to ensure fully developed turbulent flow there, the velocity profile issymmetrical about the pipe axis and therefore h is independent of Θ.

$$H_T = \frac{8}{qd^2}\int^{d/2} hqrdr$$

At the pipe outlet a surface traverse may be made which will giveh = h(r) ϴ. A numerical value for H_O may therefore be determined.The mean overall loss is then

$$\frac{H_T - H_O}{\frac{1}{2}\rho q^2}$$

With the straight part of the pipe is associated a frictioncoefficient λs given by:

$$\lambda_S = \frac{\Delta p}{\frac{1}{2}\rho q^2}\cdot\frac{d}{x}$$

WhereΔp is the pressure difference measured over a length x. Then, the loss in a length of straight pipe equivalent to the length of the givensystem = λs. (ϕR/d + l/d).

Two loss coefficients may then be defined:

An excess loss coefficient

$$\Delta\xi_o = \frac{Total.\Pr essure.loss}{Mean.Dynamic.Head}$$

$$= \frac{Total.pressure.loss.in.equivalent.length.of.stright.pipe}{Mean.dynamic.head}$$

And a loss ratio co-efficient

$$\xi_r = \frac{Total.pressure.loss.}{Total.pressure.loss.in.equivalent.length.of.stright.pipe}$$

$$\Delta\xi = \frac{H_T - H_O}{\frac{1}{2}\rho q^2} - \frac{\lambda_s}{d}(\phi R + \ell)$$

$$\Delta\xi = \frac{\frac{H_T - H_O}{\frac{1}{2}\rho q^2}}{\frac{\lambda_s}{d}(\phi R + \ell)}$$

Pressure losses may also be estimated over parts of the system separately.In a region of fully developed curved flow tine inlet and

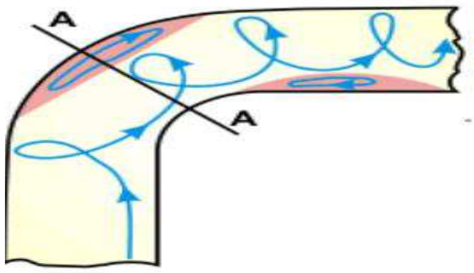


Fig. No. 1 Pressure loss region

outlet velocityprofiles are similar; hence the total pressure loss is constant across thecross-section of the pipe. If H_{cI} and H_{cO} are the values of H atinlet and outlet to this region respectively and if h_{cI} and h_{cO} are thevalues of h at the corresponding cross - sections for the same values ofr and Ø.

$$H_{CI} - H_{CO} = \frac{4}{\pi qd^3}\left\{\int_0^{d/2}\int_0^{2\pi} qh_{CI} rdrd\theta - \int_0^{2\pi} qh_{CO} rdrd\theta\right\}$$

$$H_{CI} - H_{CO} = \frac{4(h_{CI} - h_{CO})}{\pi qd^2}\int_0^{d/2}\int_0^{2\pi} qrdrd\theta$$

$$= h_{CI} - h_{CO}$$

Downstream of the bend, the static pressure becomes constant over a cross-section and the actual total pressure loss. Here will be equal to the static pressure loss. This value may be differing from the mean loss defined above since Depends on the velocity profile and this is changing through the region. For the bend transition region as a whole a mean loss may be estimated equal to the overall loss minus the losses in the other regions. A more detailed determination of the buildup of pressure losses in this region shown in figure1. However requires the making of total head traverses in several plane sat each cross-section and this technique must also be applied for information about the losses in the immediate vicinity of the bend outlet. An alternative method when only comparative results are needed is to define a teach cross-section a mean static pressure calculated from readings taken at a number of points on the pipe wall.

Let Θ be measured from the radius directed towards the bendcentre and let p = p (Θ.z) be the static pressure on the wall, where z isthe distance in diameters from the bend inlet. Then associated with thecross-section z = constant a mean pressure may be defined by the equation:-

$$p(z) = \frac{1}{\pi d}\int_0^{2\pi} p.(z.\theta)\frac{d}{2}d\theta$$

Also useful for comparative purposes, conducting analysis of bend with different flow velocities and observe the actual static pressure difference $p(z_1) - p(z_2)$ for constant Θ.

CFD Analysis –

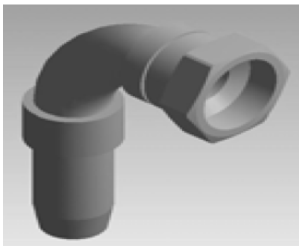
The tool used is computational fluid dynamics (CFD) to predict the flow rate, pressure and velocity at each point in pipe bend. The k-ε model is used is the most used model in the industries and it's enough strong and accurate for our calculations. This tool enables us to do several simulations rapidly.

In this analysis the single phase models and multiphase models are used for solving the respective category problems. This model will calculate one transport equation for the momentum and one for continuity for each phase, and then energy equations are solved to analyze the behavior of the system. The theory for this model is taken from the fluent software.

Specification of Problem

Consider a steady state fluid flowing through a bend of pipe of constant cross section. A 3D model of bend is as shown in Fig. 2. The diameter and length of bend in pipe 25 mm and 50 mm respectively. The inlet velocity is u (m/s),which is constant over the inlet cross-section. The fluid exhausts into the ambient atmosphere which is at a pressure of 1 atm. For creating a geometry Gambit 6.0 software is used.

Geometry in Gambit -

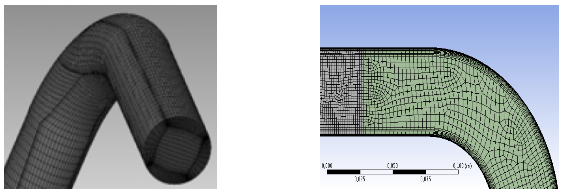


The Computational domain of circular micro channel is represented in two dimensional (3D) forms by a arc and displayed in Fig. 2. Fig.No.2– 3D model of Bend

The geometry consists of a wall, a centerline, and an inlet and outlet boundaries. The radius, R and the length, L of the pipe are in the figure no.2.

Meshing –

Hexahedral Structured meshing method done in gambit was used for meshing the geometry. 100×10 nodes were created. The 3D geometry of bend pipe with hexahedral structured mesh is shown in Fig. 3.(a) and (b).



(a) (b)

Fig.No.3 – Hexahedral Mesh in pipe bend.

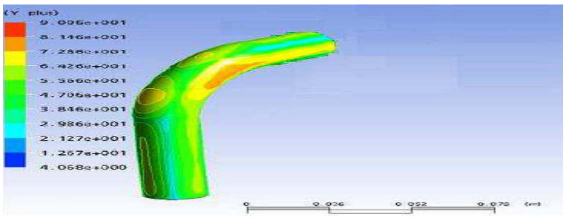
Boundary Conditions -

A steady state fluid flowing through a bend of pipe. A no slip boundary condition was assigned for the wall surfaces, friction factor is given to 0.0180 and at inlet section mass flow rate, velocity and pressure value are given as a boundary condition.

Fig.No.3 – CFD of Bend

Method of Solutions

The CFD method follows the use of commercial software Fluent to solve the problem. The specified solver in Fluent uses



a pressure correction based iterative SIMPLE algorithm with 1st order upwind

Scheme for discretising the convective transport terms. The convergence criteria for all the dependent variables are specified

Factors	Value
Pressure	10N/m^2
Velocity	4 m/s
Density	1.0
Momentum	0.7
Energy	1.0

as 0.001. The default values of under-relaxation factor as shown in Table 1 are used in the analysis.

Result and Discussion

The above results show that there is indeed a marked difference in the buildup of pressure losses according to the conditions at the bend outlet. For the bend pipe, where there is a region of fully developed curved flow, there is little or no excess loss in the downstream transition region; Although the excess loss is so small the stilling length, and diameter. When the transition length is considerably less than this the overall excess loss appears to be increased, the change occurring either in the bend itself diameters of the bend outlet. At outlet from the bend the secondary circulation is tending to invert the normal stagnation pressure distribution and losses are expected to remain high. In fact the results indicate that they differ little from those in the bend itself for as much as & diameters downstream. By this time frictional damping will have reduced the rotational effects and the excess loss drops accordingly: so too does the rate at which the velocity profile is changing so that even after diameter it is still not entirely symmetrical.

The 90^0 bend shows an increase in the loss per unit length in the bend. At bend outlet the value drops, appearing to be roughly constant over 25 mm diameter.

Although there is (theoretically) no secondary circulation at the bend outlet the particles there are displaced by a maximum amount and losses must be expected in association with the distorted pressure distribution. It is difficult to assess the accuracy of the present results but the inlet static pressures check with the total pressure differences, and the other readings, besides being reasonably consistent among themselves, arc only used for comparative purposes. In the light of this it would certainly seem that the recorded increase is significant. Some observations can be made from the analysis results, however. Firstly there is the existence of a region immediately upstream of the bend outlet in which there is an adverse pressure gradient.

In every system this region extends over at least a quarter of the circumference of the pipe wall but for the shortest transition lengths (zero and it appears to have spread to more than half. This adverse pressure gradient probably indicates a partial choking of the pipe just as if the flow had been separated from the wall. If choking is also associated with the development of a pressure maximum on the inside of the bend the changes in the total pressure profiles at the bend outlet(with transition length) in the extent and position of the region of "separation". It is noticeable that almost immediately after the bend wall static pressures become independent of 0, (though this does not mean that static pressures are necessarily constant over the whole cross-section). For long transition lengths it may be assumed that the exit pressure is uniform and atmospheric; a pressure variation may however exist at the outlet is small.

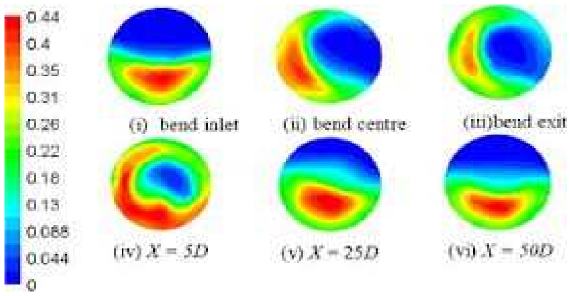


Fig.No.4 – CFD results of Pipe Bend

Conclusion-

In assessing pressure losses in a bent circular pipe both bend outlet conditions and pipe outlet conditions are important as well as the inlet velocity profile. LOSS coefficients depend on some parameter which varies with the phase of the secondary circulation at the bend outlet, Also there is a contribution to the loss in the downstream transition region of a given system not merely associated with the value of the secondary circulation at the bend outlet but also the displacement of the fluid particles there. Due in part, perhaps, to changes in displacement but more particularly to choking effects in the pipe an increase in the loss coefficient is observed as the downstream transition length is shortened. For purposes of correlation therefore it would seem to be necessary to have transition lengths of diameters or more. A total pressure gradient perpendicular to the plane ef the bend is useful in predicting the secondary flow pattern further downstream. Thus, in the long bend, a mechanism for preventing the secondary flow increasing indefinitely, and for allowing theflow to become fully developed, is provided by the formation of total pressure gradients, opposite in sign to those at the start of the bend, and the consequent production of vorticity of opposite rotational sense.

References

1. M.ROWE "Measurements and computations of flow in pipe bends" J. Fluid Mech. (1970), vol. 43, part 4, pp. 771-783.

2. Moira E. Martin and E. C. Deverson. "The Effect of Bend Outlet Conditionson the Pressure Losses in Bent Circular Pipes" A.R.C. Technical Report ministry of aviation C.P.No.505 (19,502)

3. Dean, W. R. "Flow through curved pipes". Phil. Mag. Vo1.7, p.208.

4. White C, M, "Streamline flow through curved pipes". Proc. Roy. Sot. A, Vo1.123, p.645.