VISUALIZATION OF FLOW BEHAVIOR AND ITS EFFECTED CONTOUR IN SUDDEN CONTRACTION, SUDDEN ENLARGEMENT AND SUDDEN ELBOW BY ANSYS

Navtej Singh*
Suneev Anil Bansal**
N.K Batra***

ABSTRACT

This paper describes an analytical approach to describe the areas where Pipes (used for flow) are mostly susceptible to damage. Their basic contours are discussed to know pressure values and velocity values at various sections of the pipe. ANSYS 13 software was used to plot the characteristics of the flow. Numerical simulations of the flow past pipes of various geometries were performed. Comparisons were made with the experimental and computational results presented in various studies previously. Paper tries to visualize the flow behavior in various geometric conditions of a pipe.

*Lecturer, Department of Mechanical Engineering, MM Engineering College, Mullana, Amabala

**Lecturer, Department of Mechanical Enginering, Q I E T, Mohali

***Professor & HOD, Department of Mechanical Engineering, MM Engineering College, Mullana, Amabala

1. INTRODUCTION

Pipes are the most efficient media for transporting fluid from one place to another. The efficiency of the system to minimize the losses in a network is as important as saving money these days. The researchers all over the world are working hard to find a solution to minimize these head losses (energy), total length of the network and pipe diameters. Lahiouel Y et al [1]. When a viscous fluid flows through a pipe, a part of its energy is spent in the maintenance of the flow. This energy is converted into thermal energy because of its internal friction and turbulence. This leads to the expression of the energy loss in terms of the fluid height known as the head loss. These losses are classified into major losses or linear head present throughout the length of the pipe and minor losses or singular head occurring due to minor appurtenance and accessories present in a pipe network. These appurtenances encountered by the fluid flow are sudden or gradual changes of the boundaries resulting in a change in magnitude, direction or distribution of the velocity of the flow. If pipes have many minor appurtenances but its length is small then its total minor head losses can be greater than frictional head loss. In pipes used for petroleum and water distribution system having considerable length, the terms- major head loss and minor head loss can be used without confusion. Lahiouel Y et al [2]. Time to time many researchers worked in order to achieve a general and precise formulation of the diverse type of head losses. Weisbach, 1855 [3] was the first one to give the concept of relation for head loss. Darcy[4] and Weisbach formulated equation- Darcy-Weisbach formula to measure accurate pipe friction loss. It is one of the most accurate formulas for measuring the pipe friction loss but is difficult to calculate and use than the other friction formulas. It has now become a standard equation for hydraulic engineers in practice. However, until then the establishment of the friction was still unresolved. Nikuradse [5] performed many experiments using smooth and artificially roughened pipes and drew a diagram, also known as Stanton diagram or Stanton-Pannel diagram. Colebrook [6] found that results of Nikuradse do not match with actual pipes. It was possible to use results for commercial pipes by introducing the concept of equivalent surface roughness. Colebrook then derived a relation known as Colebrook-white equation for finding friction factor in turbulent flow. Prandtl [7] found the relation expressing friction coefficient as a function of (e/D). It is now widely known as Karman-Prandtl equation. Rouse [8] plotted friction coefficient ($1/\sqrt{f}$) against the Karman number represented by:

$$K = R_e\sqrt{f} = D^{3/2}\sqrt{2gS/v}$$

He produced the necessary curves for the Colebrook transition zone. In 1944, L.F Moddy plotted charts using Colebrook-white equation known as moody charts or friction factor chart based on the roughness of the pipe against the Reynolds's number of the pipe. For flow between 2000<Re<100000(turbulent flow), Blasius suggested a relationship between the friction and Reynolds number. Hezen-Williams formula [10] was extremely popular with piping engineers because of its relatively simple calculation properties. However, the Hezen-Williams results rely upon the value of friction factor, C hw, is used in the formula and the C value can vary significantly from around 80 up to 130 and higher depending on the pipe material, pipe size and fluid velocity.

Various experiments have been carried out by different authors in order to determine the linear and singular head losses. They also found that because of turbulent eddies, vane contracta and friction with wall of pipe, these losses occur. In this paper, we have tried to visualize these losses in CFX software and verify these after conducting the experiments in the laboratory.

2. FLOW ANALYSIS IN FLUENT

In ANSYS 13 workbench, first of all we select the fluid flow (Fluent) from the analysis system. Then, this is sent into project schematic window. It contains design modular, mesh, setup, solution and results.

Solution can be obtained by the following steps:

i. Create the geometry in DM
ii. Meshing
iii. Setup
iv. Define Model
v. Define Material
vi. Define Cell Zone
vii. Boundary Condition
viii. Solve
ix. Initiation
x. Solve – Run Calculation
xi. Analyze results

3. DETAILED PROCEDURE

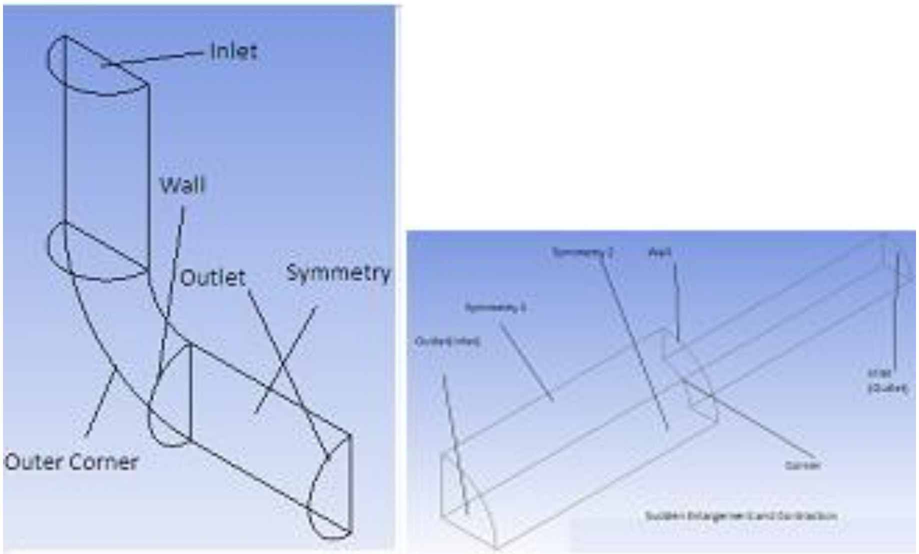

Figure1: Sudden enlargement, contraction and Elbow

Pipes are represented in 3D. Pipe Geometry is displayed in figure 1. Sudden Enlargement and sudden contraction is created from extrude and elbow from sweep. Geometry consists of a Wall, Symmetry 1&2, Inlet and Outlet. The radius and length of the pipe can be specified. After geometry, next step is Meshing. The mesh influences the accuracy, convergence and speed of the solution. Furthermore, the time it takes to create and mesh a model is often a significant portion of the time it takes to get results from a CAE solution. Therefore, the better and more automated the meshing tools, the better the solution. Tetrahedron (Patch Independent) method is used for these problems. This has the advantage that local numerical diffusion is reduced and is therefore suitable for complex flows, e.g. flow reversal [11]. During meshing, names are given to different parts of geometry like wall, symmetry 1&2, inlet and outlet by create name selection command. After this, inflation is inserted for providing thickness of the pipe to generate mesh and setup is initiated for the problem. Now on fluent window Double Precision is selected because Double Precision will be significant if tiny relative differences are significant. Examples where you would expect double precision to make a difference are natural convection (especially if modeled with the fully compressible option); meshes with a large difference between the largest and smallest element sizes; large geometries with small but significant features; flows with large pressure/velocity/temperature variations etc. Double precision is always more accurate, but does run slower (on 32 bit machines at least) and takes up more memory. Then, Viscous laminar model and k-epsilon (2 eqn) are selected from model command. The CFX is based on a coupled solver for solving the differential equations using the fully implicit Discretisation method and treating the hydrodynamic equations as one single system. To reduce the number of iterations required for convergence, a false-time stepping method is imposed which guides the approximate solutions in a physically based manner to a steady-state solution. Buoyancy is modeled using the Boussines approximation in which the forces are modeled as source terms in the momentum equations. Various models exist in CFX-5 code for modeling turbulent flow. Two-equation models based on the eddy-viscosity concept include the k-ε (Launder and Spalding, 1974), k-ω (Wilcox, 1998) and Shear Stress Transport (SST) k-ω based (Menter, 1994) models. Compared to the commonly used k-ε turbulence model, the k-ω model implies a new formulation for the near wall treatment which provides an automatic switch from a wall-function to a low-Reynolds number formulation based on the near-wall grid spacing. This makes it more accurate and more robust. The turbulence viscosity is assumed to be linked to the turbulence kinetic energy (kequation) and turbulent frequency (ω-equation) instead of the turbulence dissipation rate (ε-equation in the k-ε model). To overcome the sensitivity of the k-ω model to free stream conditions, the SST model was developed. It blends the k-ω model near the surface with the k-ε model in the outer region. In contrast, Reynolds Stress Turbulence models such as the standard Launder-Reece-Rodi Isotropic Production (LRR-IP) model (Launder et al, 1975) and Second Moment Closure-ω (SMC-ω) model (Wilcox, 1998), do not use the eddy-viscosity hypothesis, but solve transport equations for all components of the Reynolds stresses. This makes Reynolds Stress models more suited to complex flows. However, practice shows that they are often not superior to two-equation models because convergence difficulties often occur. The LRR-IP model is based on the k-ε model, whereas the SMC- ω is based on the k-ω model with the advantages already explained Zitzmann, T et al [12].

Now, the default fluid in CFX is air, but we want to define fluid as water. In the Create/edit Fluent database, water liquid is selected. In cell zone condition, default selection is solid, this is changed into fluid and water liquid is selected. In boundary condition inlet velocity magnitude is set along with Turbulence intensity and Hydraulic diameter. Before initialization from Residual monitors, plot and print to console are selected. Then, calculation with 500 number iteration and change reporting interval to 10 is run and the solution is calculated. These results are now confirmed by doing experiments in the lab. Water flowing from elbow, sudden enlargement and contraction is collected in a tank with 0.16 m^2 area. Diameter of the small and large pipe is 0.025m and 0.05m.

4. RESULTS AND ANALYSIS

Three set of conditions are discussed in this network of pipes. All three conditions viz sudden enlargement, sudden contraction & elbow frequently appear in the pipe network. Various changes in parameters of the pipe network are discussed. The numerical values thus achieved are shown in tables in respective cases.

4.1 Flow Behavior in Sudden Enlargement:

In sudden enlargement, diameter of 0.0125m is enlarged to 0.025m. Various velocity and pressure charts are drawn and shown in table1, figure 2 & 3.

Table 1 : Sudden Enlargement

Parameter	**Value**
Large Diameter (outlet)	0.025 m
Small Diameter (inlet)	0.0125 m
Length of small dia. pipe	0.1m
Length of large dia. Pipe	0.1m
Inlet Velocity	1.44 m/s
Min Pressure	-3.78e+002
Max. Pressure	1.35ee+00

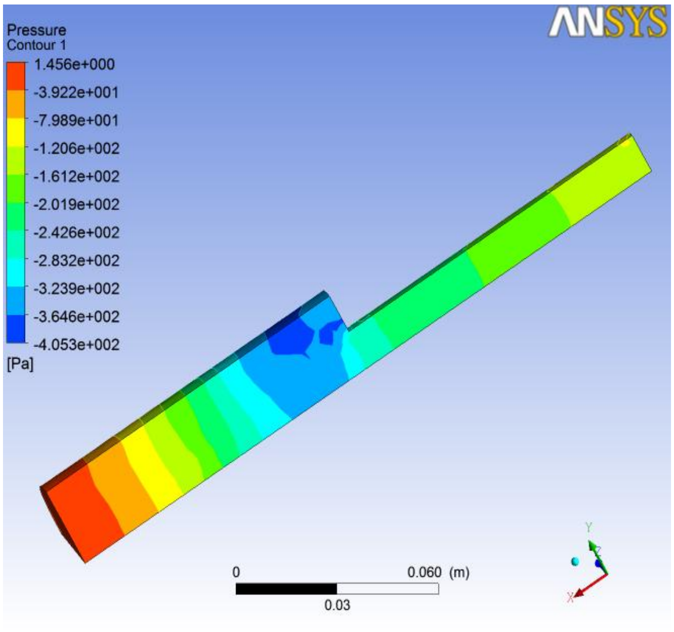

Figure 2: Sudden Enlargement (Pressure Behavior)

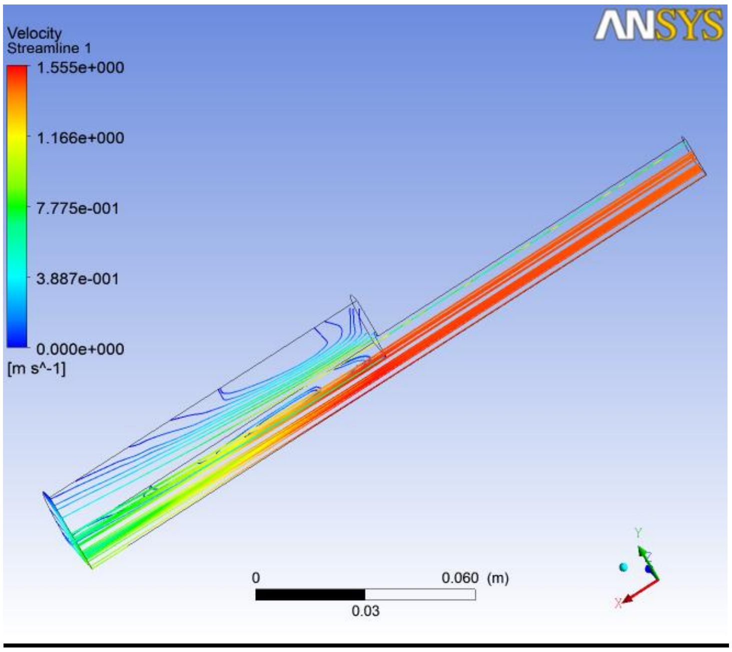

Figure 3: Sudden Enlargement (Velocity Behavior)

4.2 Flow Behavior in Sudden Contraction:

In sudden contraction, diameter of 0.025m is decreased to 0.0125m. Various velocity and pressure charts are drawn and shown in table 2, figure 4& 5.

Table 2 : Sudden Contraction

Parameter	**Value**
Large Diameter(inlet)	0.025 m
Small Diameter(outlet)	0.0125 m
Length of small dia. pipe	0.1m
Length of large dia. Pipe	0.1m
Inlet Velocity	1.44 m/s
Min Pressure	-9.4e+003
Max. Pressure	2.7e+004

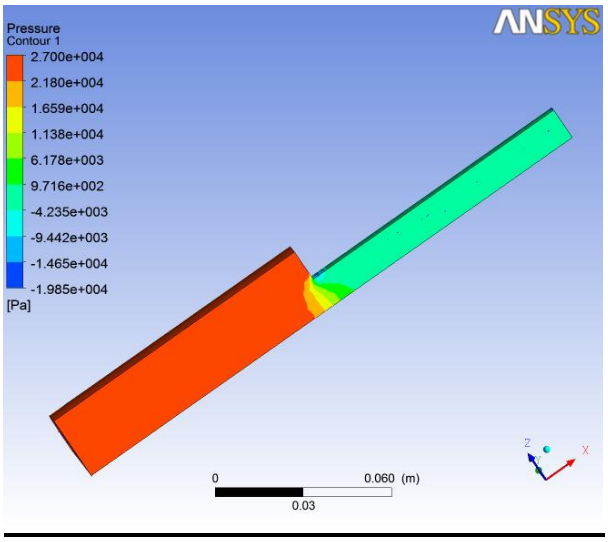

Figure 4 : Sudden Contraction (Pressure Behavior)

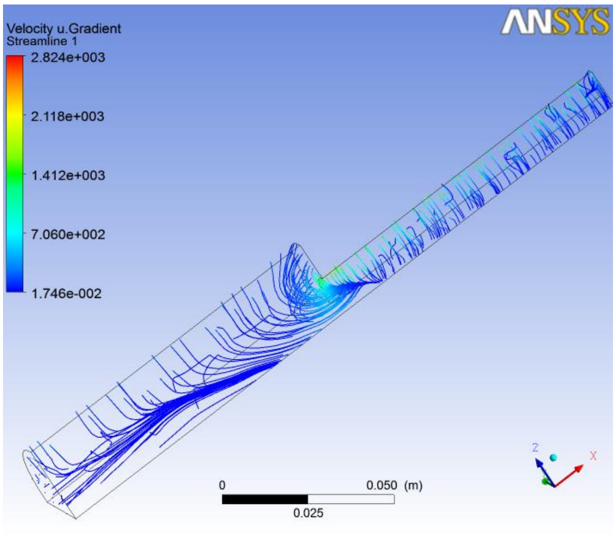

Figure 5: Sudden Contraction (Velocity Behavior)

4.3 Flow Behavior in Sudden Bend (Elbow):

In elbow, pipe diameter of 0.025m is taken. Radius of pipe bend is taken as 0.02625m. Various velocity and pressure charts are drawn and shown in table 3,figure 6 & 7.

Table 3 : Sudden Bend(Elbow)

Parameter	**Value**
Length L1	0.1m
Length L2	0.1m
Radius at corner	0.0525m
Radius of pipe	0.02625m
Inlet Velocity	3m/s
Min Pressure	-5.8e+003
Max. Pressure	4.8e+003

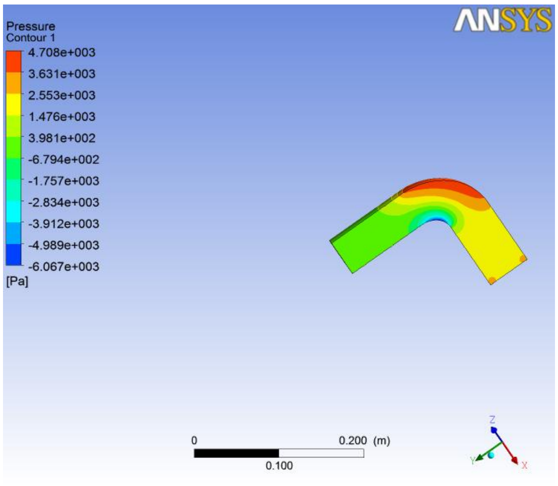

Figure 6 : Sudden Bend (Pressure Behavior)

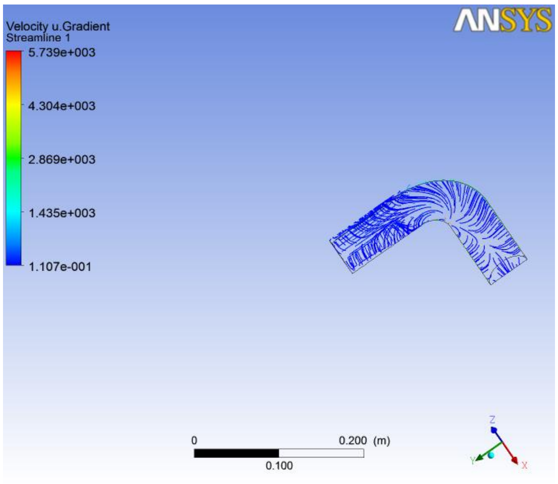

Figure 7: Sudden Bend (Velocity Behavior)

5. DISCUSSION & CONCLUSION

From the above analysis, it is observed that the flow is severely disrupted if there are contour changes occurring in the downstream flow in the pipe. Sudden enlargement creates more severe formation of flow eddies than sudden contraction. Also, the losses are more at the point where the enlargement in the pipe begins.

In the sudden contraction, vane contracta's are formed at the point of contraction and this point is the most susceptible point for pipe damage. In elbow, the outer corner at the bend is more susceptible to damage because the pressure is maximum at this contour. So, to increase the life of the pipe in cases of sudden contraction & enlargement, the pipes must be designed in view of the above observations making the corners more round so as to minimize the losses in the pipes. However, at the elbows, outer regions must be made thicker to increase strength of the bend to be able to bear the increased pressure changes at this contour.

To conclude, this examination results indicate that ANSYS can be used with high degree of accuracy to visualize the minor or singular head losses due to minor appurtenances and accessories present in a pipe network.

6. REFERENCES

[1] Lahiouel Y., Haddad A., Khezzar L., Chaoui K., Kondratas A.,"Development of method of routing fluid distribution networks", J. Mechanica, Vol. 43, N° 4, (2003), pp. 27-34.

[2] Lahiouel Y., Haddad A., Chaoui K., "Evaluation of head losses in fluid Transportation networks Sciences & Technologies B – N°23, juin (2005), pp. 89-94

[3] Weisbach J."Die Experimental Hydraulik", Freiberg, Germany: Engelhardt, (1855).

[4] Darcy Weisbach Formula http://www.pipeflow.com/pipe-pressure-drop-calculations/ pipe-friction-loss (12/15/2011)

[5] Nikuradse J., 1933,"Strmungsgesetze in Rauben Rohren", pp 361, Verein Deutsher Ingenieure, Forschungsheft, (1933).

[6] Colebrook C.F.,"Turbulent Flow in Pipes with Particular reference to the Transition Region between the Smooth and Rough Pipe Laws", J. Inst. of Civil Engrs.N° 11,(1939), pp.133-156.

[7] Bhave, P.R.,"Analysis of flow in water distribution networks", pp.25-68 Technomic Pub. Co., Inc., USA, (1991).

[8] Rouse H., "Evaluation of Boundary Rough Proc.2nd Hydraul. Conf., Bulletin N° 27, Univ. of Lowa, Lowa City, (1943).

[9] Moody L.F., "Friction Factors for Pipe Flow", Trans. American Society of Mechanical. Engineers., No. 66, (1944), pp. 671-684.

[10] Hazen-Williams Formula http://www.pipeflow.com/pipe-pressure-drop-calculations/ pipe-friction-loss (12/15/2011)

[11] Ansys ,Inc. http://http://www.idac.co.uk/products/downloads/ Meshing.pdf

[12] Tobias Zitzmann1, Malcolm Cook2, Peter Pfrommer1,Simon Rees2, Ljiljana Marjanovic2, "Simulation Of Steady-State Natural Convection Using CFD" Ninth International IBPSA Conference Montréal, Canada

[13] JONAS BREDBERG http://www.tfd.chalmers.se/~lada/postscript_files/ jonas_report_lowre.pdf