Numerical Simulation For Visualising Effect of Surface Roughness on Flow In Simple Pipe

ABHISHEK SHARMA[1], VISHAL GUPTA[2]*, ABHISHEK KUMAR JAIN[2] and SUDEEP KUMAR SINGH[3]

[1]National Institute of Technology, Rourkela.
[2]Maulana Azad National Institute of Technology, Bhopal (M.P.), India.
[3]Amity School of Engineering and Technology, Bijwasan.

(Received: February 10, 2015; Accepted: April 18, 2015)

ABSTRACT

With advances in computational power and mathematics, CFD has emerged as boon for design optimisation in various fields. We are surrounded with fluids from every side. And the physics of fluid is very difficult to understand if all the aspects of real life are considered. The flow even inside a simple pipe is very complex to observe practically. Head loss occurs due to friction in pipes and leads to loss of energy. In this paper, effect of surface roughness in pipes has been simulated and results in graphical and pictorial form have been presented. The simulation have been done with CFX code using SST turbulence model.

Key words: Computational fluid dynamics, steady state, frictional head loss.

INTRODUCTION

Earlier there were only experimental techniques available to predict the performance of turbo machinery and calculating the losses in flowing fluid even in pipes. The experiments were done on models. But observation of behaviour of flow was very difficult to observe and studying local parameters was very difficult[5]. With the advances in the field of computational mathematics and computational power, a lot of development has been taken place in the field of Computational Fluid Dynamics. Detailed flow analysis, study of local parameters, flow visualisation, detailed pressure and velocity distribution can be studied with the help of CFD[1,2]. There are many software packages available in the field of CFD[8]. The main objective of this paper is to discuss the effect of friction in pipes on flow.

Geometric Modeling and Common Input Data

Geometry in 2D or 3D is needed for numerical flow simulation depending the nature of problem. The geometry which is flow domain is descritised into small elements called mesh over which governing equations are solved.

Geometric Modeling

For studying the effect, geometry of pipe of 200 mm with diameter of 10 mm is considered. One side of pipe is described as inlet and other is kept open to atmosphere. Modeling has been done in ANSYS ICEM CFD 14.0. 3-D view of pipe is shown in Fig.1.

Meshing

Tetra mesh has been used for meshing 3D domain and for 2D surfaces triangular elements are used. Prism layer has been applied at pipe surface for capturing boundary layer effect. Mesh quality is checked for orthogonlity and aspect ratio to be within recommended values of ANSYS- CFX. Meshing of pipe is shown in Fig 2.

Common input data

Input data is needed for defining the working fluid and overall physics. The values used in present analysis is mentioned in Table 2.

Boundary conditions

Boundary such as inlet, outlet, wall, symmetry etc are needed to be defined and the value of obtained results also depends a lot on the values of boundaries defined. Flowing boundary conditions are defined for present work:

Inlet Boundary Condition

The mass flow rate and its direction with normal direction to the inlet of pipe is specified.

Outlet Boundary Condition

The reference pressure at the outlet was set equal to 1 atmospheric.

Wall Conditions

The walls of the domain is assumed to be smooth and no slip condition is assigned for smooth wall and its value is changed for varying the roughness of pipe.

Formulae used

Total head and loss coefficient are computed using the following formulae:

1. Total head at inlet of pipe

...(1)

2. Total head at outlet of pipe

...(2)

3. Head loss coefficient

...(3)

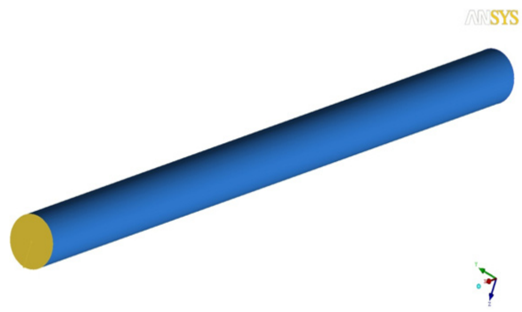

Fig.1: Geometry of Pipe

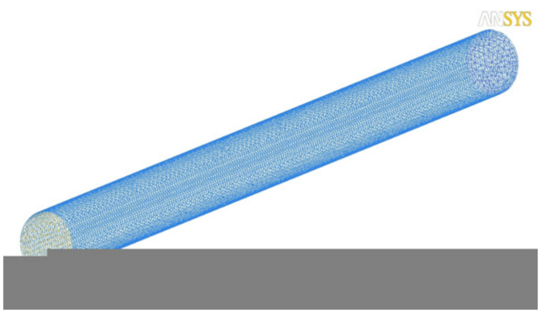

Fig.2: Meshing of Pipe

Table 1: Mesh data for pipe

Part Name	Number of nodes	Number of elements	Element type
Inlet	196	145105	TriangularQuadrilateral
Outlet	203	145111	TriangularQuadrilateral
Pipe wall	8772	17472	Triangular
Flow domain	56238	15162352416	TetrahedralWedges

Table 2: Common input data

Analysis type	Steady state
Domain Type	Fluid domain
Fluid type	Water
Reference Pressure	1 atm
Domain Motion Option	Stationary
Mesh Deformation Option	None
Fluid Temperature	25^0C
Turbulence Model	SST
Density of Water	997 Kg/m^3

Mesh independency test

Mesh independency test has been done by considering three mesh sizes for the domain. The simulation is done for discharge of 0.001 m^3/sec. The flow in pipe jet is assumed to be ideal, with a constant velocity profile. Results of mesh independency are given in Table 3.

Drop in pressure head is free of mesh above 39459 nodes but for better pictorial visualisation and for further simulation, mesh with 56238 nodes is considered.

Table 3: Mesh independency test

No. of elements	No. of nodes	Drop in pressure head	Time taken
61047	19976	2.68 m	2 minutes
103075	39459	2.76 m	4 minutes
204039	56238	2.76 m	12 minutes

Table 4: Variation in head loss coefficient and turbulent kinetic energy

Pipe type	Roughness size (mm)	Turbulent kinetic energy Inlet	Turbulent kinetic energy Outlet	Head loss coefficient
Smooth pipe	0.0000	0.5799	0.6247	0.24708293
PVC Pipe	0.0015	0.5806	0.6648	0.25556047
Steel pipe	0.0450	0.5874	1.2989	0.34785651
Galvanised iron pipe	0.1500	0.5965	2.0326	0.44445985
Cast iron pipe	0.2600	0.6077	2.5555	0.49256872
Concrete pipe	1.5000	0.5837	4.6547	0.61886265

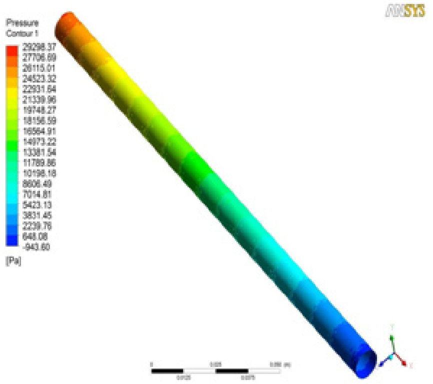

Fig.4: Variation pressure for smooth pipe

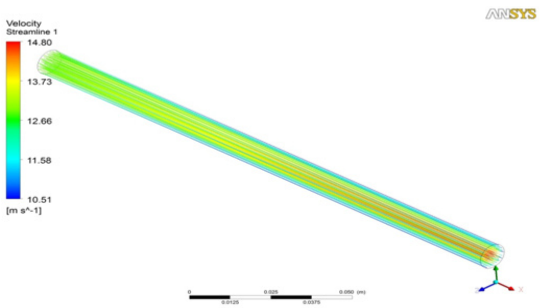

Fig.5: Water velocity streamlines for smooth pipe

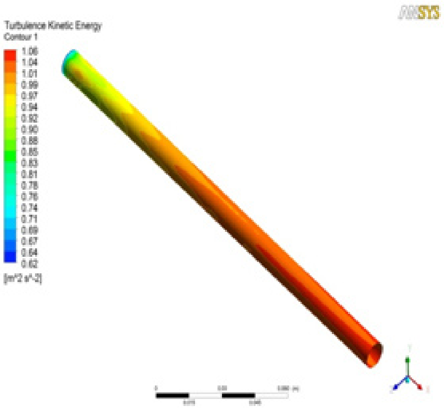

Fig.6: Variation in turbulent kinetic energy for PVC pipe

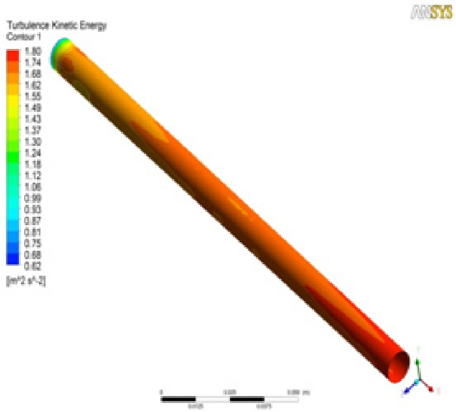

Fig.7: Variation in turbulent kinetic energy for steel pipe

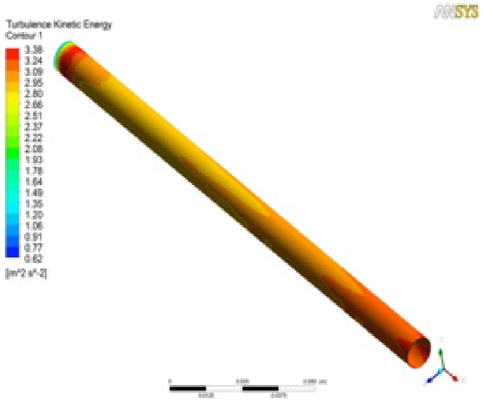

Fig.9: Variation in turbulent kinetic energy for cast iron pipe

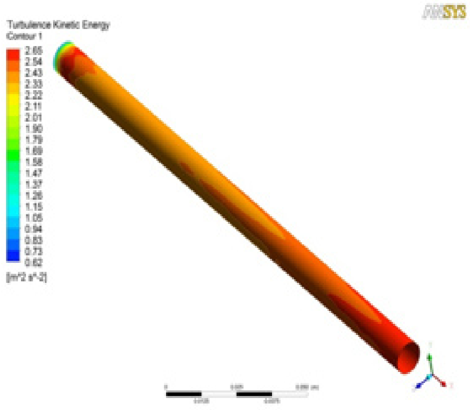

Fig.8: Variation in turbulent kinetic energy for galvanised iron pipe

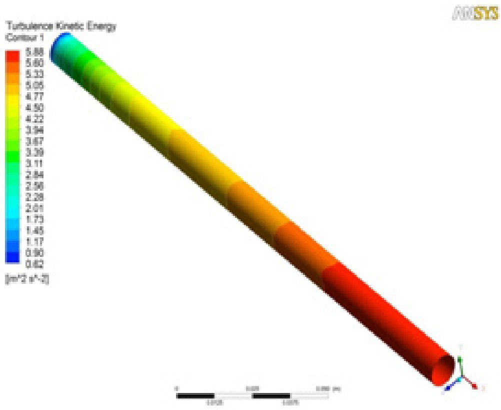

Fig.10: Variation in turbulent kinetic energy for concrete pipe

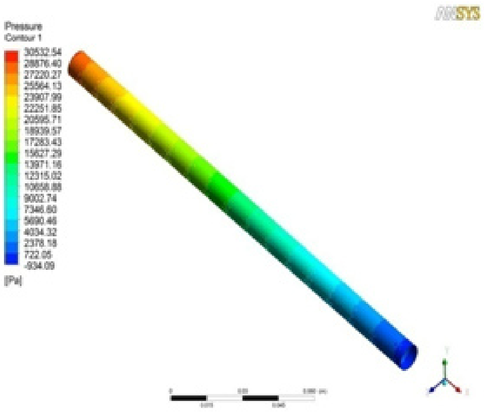

Fig.11: Variation in pressure for pvc pipe

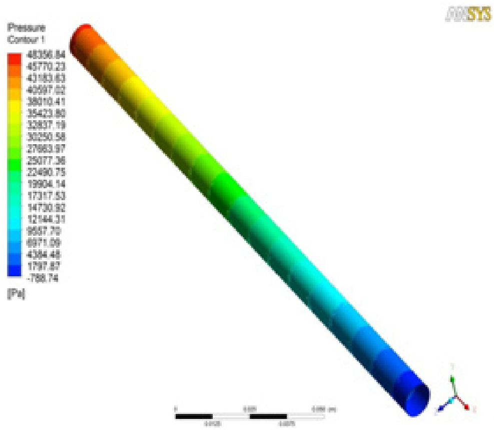

Fig.12: Variation in pressure for steel pipe

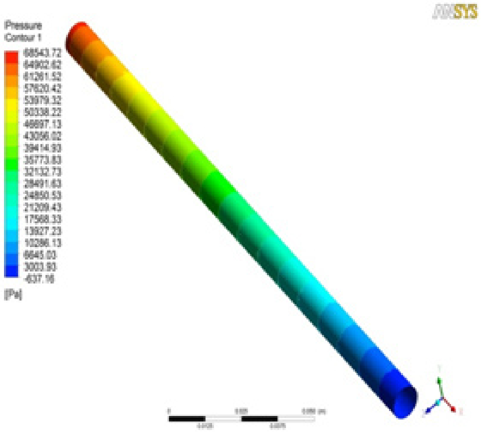

Fig.13: Variation in pressure for galvanised iron pipe

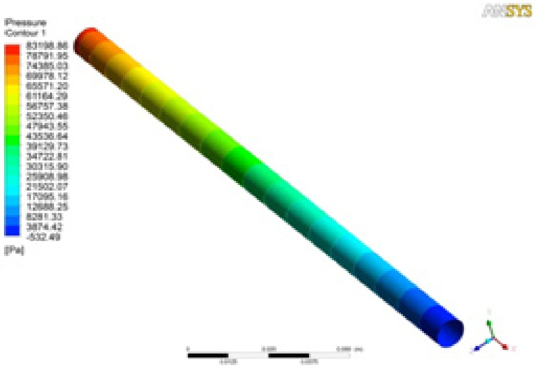

Fig.14: Variation in pressure for cast iron pipe

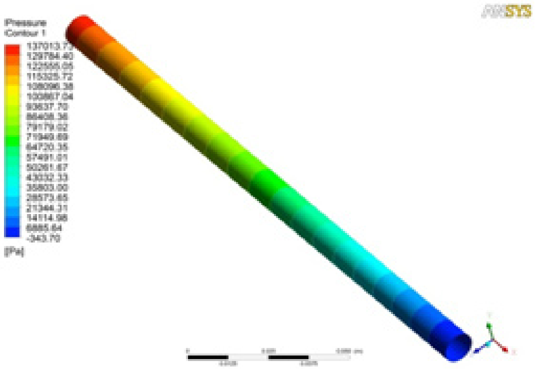

Fig.15: Variation in pressure for concrete pipe

RESULTS AND DISCUSSION

The analysis is carried out for six different values of pipe surface roughness. Roughness of pipe considered are 0 ìm (smooth pipe), 0.0015 mm (PVC and glass pipes), 0.045 mm (steel pipes), 0.15 mm (galvanised iron pipes), 0.26 mm (cast iron pipes), 1.5 mm (concrete pipes). The RMS residual was set to 10^{-6} for termination of the analysis. The analysis provided pressure, velocity and turbulent kinetic energy distribution within the flow domain and at boundaries.

As observed from Fig.3, it is seen that turbulent kinetic energy increases from inlet to outlet. This may be due to boundary layer effect. Fig.4 shows variation of pressure from inlet to outlet which decreases gradually. Water velocity streamlines in Fig.5 indicates highest velocity at centre of pipe and least at the surface of pipe. Velocity at boundary also decreases from inlet to outlet.

Variation in turbulent kinetic energy can be seen from Fig. 6 to Fig.10. The variation in turbulent kinetic energy from inlet to outlet increases as roughness of the surface of pipe increases.

Pressure decreases from inlet to outlet in all the cases as seen from Fig.11 to Fig.15. Highest pressure difference is observed for concrete pipe.

The average values of pressure, velocity and turbulent kinetic energy at inlet and outlet were obtained using function calculator in CFD- Post from simulation results. The values of loss coefficient is given in Table 4. It is observed from Table 4 that head loss coefficient decreases with increase in grain roughness size of pipe and is observed to be maximum for concrete pipe.

CONCLUSIONS

It is observed from numerical simulation results that pressure at the inlet of pipe is more as compared to outlet and decreases gradually. The comparison of head loss coefficients and pressure distribution indicates that PVC pipes lead to less loss of energy. It may also be concluded that CFD is good tool to predict the performance of pipes in less time. The effect of friction on the performance of turbo machines can also be studied with the help of CFD.

Nomenclature

CFD	Computational fluid dynamics
g	Acceleration due to gravity
H	Head
ñ	Density of the fluid
P	Pressure
TP	Total pressure
RMS	Root mean square
SST	Shear stress transport model
in	Inlet
out	Outlet

REFERENCES

1. Xiao Y X, Zeng C J, Zhang J, Yan Z G and Wang Z W, "Numerical analysis of the bucket surface roughness effects in Pelton turbine", 6th International Conference on Pumps and Fans with Compressors and Wind Turbines, IOP Conf. Series: *Materials Science and Engineering,* **52:** 052032 (2013).
2. Gupta V, Prasad V, "Numerical Investigations for Jet Flow Characteristics on Pelton Turbine Bucket", *International Journal of Emerging Technology and Advanced Engineering,* **2**(7): pp 364-370 (2012).
3. Rajak Upendra, Prasad Vishnu, Khare Ruchi, "Numerical Flow Simulation using Star CCM+", The International Institute for Science, Technology and Education Proceeding of International Conference on Recent Trends in *Applied Sciences with Engineering Applications,* **3**(6): pp 34-41 (2012).
4. ANSYS CFX 13 software manuals.
5. Modi P.N. and Seth S.M., Hydraulics and Fluid Mechanics including Hydraulic Machines, Standard Book House, Delhi (2011).
6. Bansal R.K., A Textbook of Fluid Mechanics and Hydraulic Machines, Laxmi Publications, Delhi (2010).
7. Garde R.J., Turbulent Flow, New Age International Pvt. Limited, New Delhi (2009).
8. Chapra S.C. and Canale R.P., Numerical Methods for Engineers, Tata Mc Graw Hill Publishing Company Ltd., New Delhi (2001).
9. Anderson John D., Computational Fluid Dynamics, McGraw-Hill Inc., New York (1995).
10. http://en.wikipedia.org/wiki/Friction_loss on 11/03/2015