Paper ID: FM-06

Analysis of Flow in Complex Pipe System

Abdul Mohaimen Safi
Department of Mechanical Engineering, BUET, Dhaka- 1000, Bangladesh
E-mail: mohaimensafi@gmail.com

Abstract

In hydraulic engineering the analysis of water flow through a closed, pressurized piping system is a common task. This paper presents an analysis of flow in complex pipe system using finite element method. A computer model has been developed. The plot of velocity profile within the pipe and the graph of variation of velocity of the outlet pipe have been studied. This model is good opportunity to study variety of practical design situations including farm irrigation pipe networks and municipal water systems. This model is flexible in its ability to handle different design problems including branched and multi-looped complex pipe networks.
.

Keywords: *hydraulic engineering, complex pipe system, finite element method.*

1. Introduction

Pipe network analysis is a calculation of fluid flows and pressure drops in complex piping systems. Analysis of piping systems is important for public utilities supplying water to consumers, natural gas distribution planning, or any system of piping where consistent delivery pressures and flow rates are important. 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. When a real fluid flows through a pipe, a part of its energy is converted into thermal energy due to internal friction and turbulence. This leads to the expression of energy loss in terms of the fluid height known as head loss [1]. Head losses are divided into two main categories, “major losses” associated with per length of the pipe and “minor losses” associated with minor appurtenance and accessories like bend, fittings, valves etc. These accessories cause change in magnitude, direction and distribution of the flow. For relatively short pipe systems, with a relatively large number of accessories like bends and fittings, minor losses can easily exceed major losses. Pipe network analysis uses iterative method regardless of the fluid being delivered. The importance and complexity of these calculations increases as a pipe network grows, and customers expect uninterrupted delivery [2]. Velocity and pressure drops can be measured experimentally, but laboratory measurements do not always translate well to real-world systems of overlapping loops, many delivery points and a constantly changing pipe network. Analytical solution for every problem is not possible for complex shape and boundary condition. Hence, numerical method had become popular with the researcher in the last few decades. Analysis of flow in a complex pipe system requires solution of partial differential equations. In this present paper for the solution of these partial differential equations Finite Element Method (FEM) has used. A finite element model of a problem gives a piece wise approximation to the governing equations. Since these elements can be put together in a variety of ways, they can be used to represent exceedingly complex shapes. In the present paper numerical simulation for flow in a complex pipe system has been investigated using ANSYS one of the most popular finite element method software.

2. Description of the problem

The geometry of the problem is shown in Fig 1. A two dimensional analysis has been performed assuming the fluid to be incompressible, and constant property of fluid throughout the pipe. In all the simulation water of density ρ= 1000 kg/m^3 and viscosity μ= 1 kg/m-s at a reference temperature of 25^{0}C have been taken. Uniform velocity distribution is assumed for the inlets shown in Fig 1. A stationary no-slip boundary condition is imposed on the walls of the pipe system. At outlet pressure outlet boundary condition is used and diffusion fluxes for the variables in the exit direction are set to zero.

3. Governing equations

The details of the governing equations and the treatment of the interface are obtained from ANSYS user guide [3].

Continuity equation:

$$\frac{\partial(\rho)}{\partial t} + \nabla.(\rho U) = \sum_q S_q \tag{1}$$

Where, ρ, U, t, S are density, velocity, time and mass source, respectively. In the present case S is zero.

Momentum equation:

The single momentum equation is solved in the computational domain and the resulting velocity field is shared amongst the phases. The general momentum equation can be written as:

$$\frac{\partial(\rho U)}{\partial t} + \nabla.(\rho U.U) = -\nabla P + \nabla.[\mu(\nabla U + \nabla \mathrm{U}^{\mathrm{T}})] + (\rho g) + F \tag{2}$$

Where P, g, F, μ are pressure in the flow field, acceleration due to gravity, body force acting on the system and viscosity of the flowing fluid respectively.

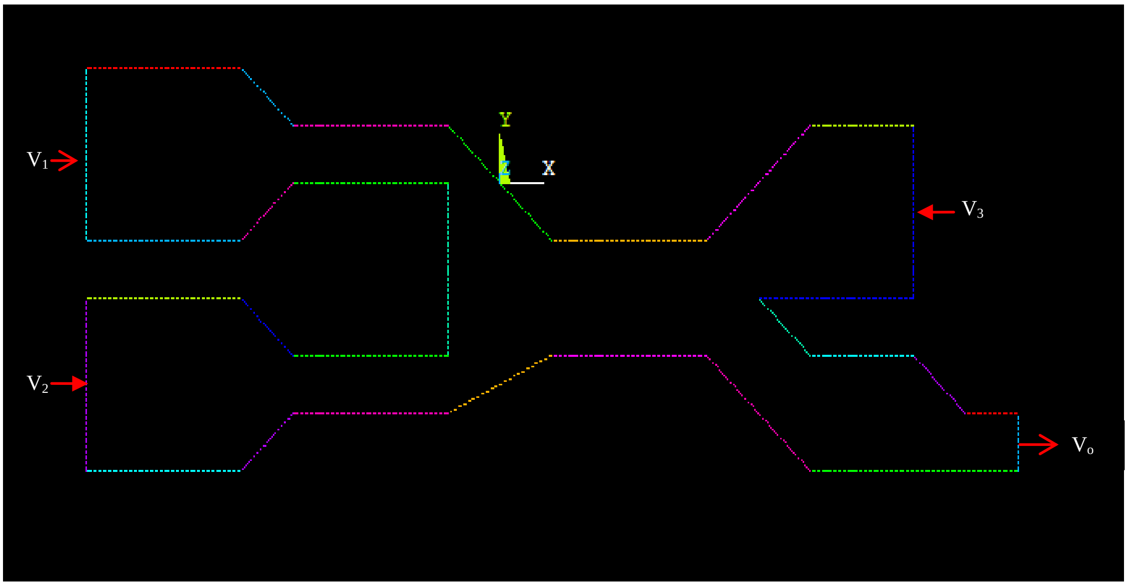

Fig. 1. Model geometry

4. Solution methodology

For the analysis FLOTRAN element FLUID141has been used as FLUID141 efficiently handle transient or steady state fluid/thermal systems that involve fluid and/or non-fluid regions. The FLUID141 element has these characteristics: 1) Dimensions: 2-D, 2) Shape: Quadrilateral, four nodes or triangle, three nodes 3) Degrees of freedom: Fluid velocity, pressure, temperature, turbulent kinetic energy, turbulent energy dissipation, multiple species mass fractions for up to six fluids. The conservation equations for viscous fluid flow and energy are solved in the fluid region, while only the energy equation is solved in the non-fluid region. For the FLOTRAN CFD elements, the velocities are obtained from the conservation of momentum principle and the pressure is obtained from the conservation of mass principle. The temperature if required is obtained from the law of conservation of element discretization of the governing equation for each degree of freedom is solved separately [3]. The commonly used κ-ε turbulent model was applied in the simulation. This model is based on the concept that turbulence consists of small eddies which are continuously forming and dissipating. This model is numerically more robust than the Reynolds stress turbulent model in terms of convergence and stability. Even though the Reynolds numbers in the simulations are low however, it was thought that the flows are generally turbulent due to regular disturbances. The Ansys software package offers various mesh types. For this study the fine mesh option was used in order to obtain simulated results of high accuracy. The maximum grid length was set to 0.0001m. The maximum number of iterations was set to 1000 and the target relative residual was set to 0.0001. The blend factor was specified to 2nd order. These settings were determined based on many trial simulations. Intel Core 2 Duo, 2.0 MHz processing machine has been used for the entire computation.

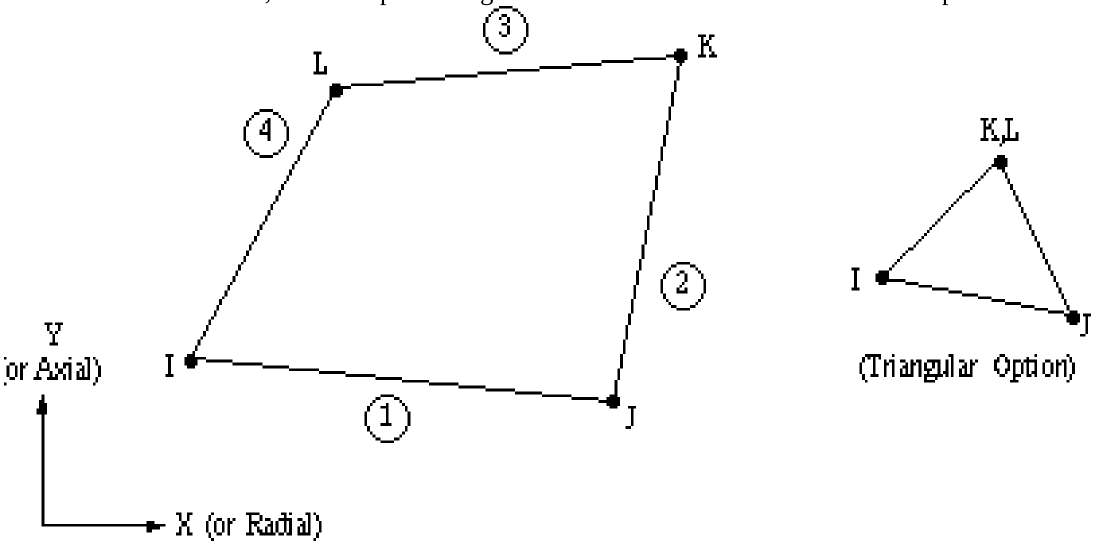

Fig. 2. FLOTRAN element FLUID141

5. Results and discussions

The solution velocity and pressure for each nodal point within the tolerance limit 0.0001 has been obtained. The variations of values of outlet velocity V_o with the change of non-dimensional ratio of inlet velocities V_3/V_1 were plotted for outlet section (designated by yellow arrow). The selected section is shown in Fig. 3.

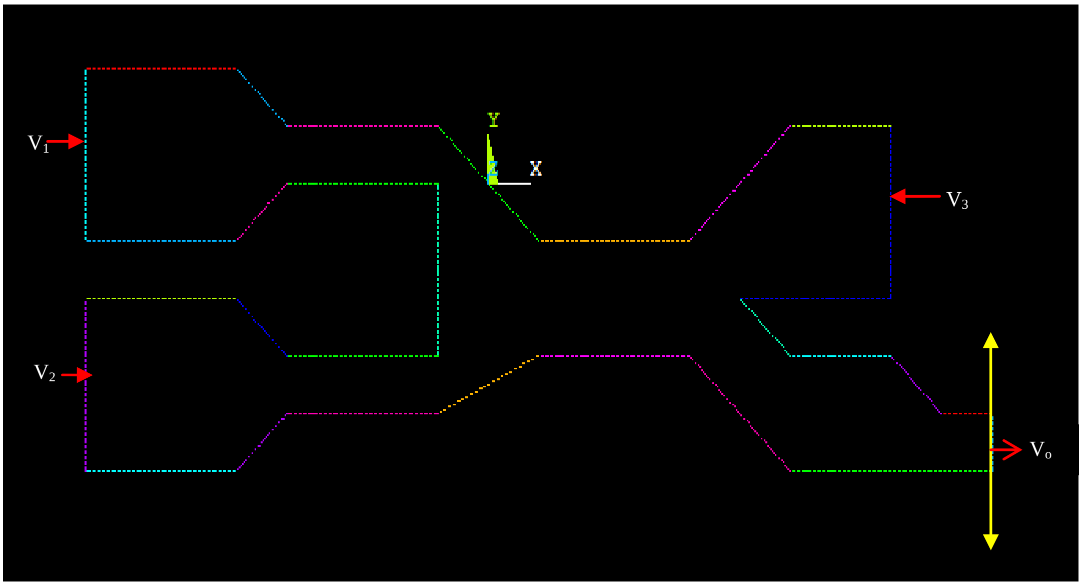

Fig. 3. Selected section chosen for studying designated by yellow arrow

In Fig. 4(a) Variation of outlet velocity with respect to different V_3/V_1 ratio has been presented. For constant dimensionless distant Y/D_o along the radius of the pipe. The result for V_3/V_1= 0.18 shown in Fig. 4(a) shows that outlet velocity profile changes with respect to V_2/V_1. For V_2/V_1 ratio upto 1 the velocity profile remain somewhat parabolic then with the increase of V_2/V_1 ratio the profile changes. Also it appears that for constant value of Y/D_o the maximum value of V_0/V_1 is found at the midpoint along the radius of the pipe. The result for V_3/V_1= 0.8 is shown in Fig. 4(b). Here again we see that V_2/V_1 ratio controls the outlet velocity profile. From the results of V_3/V_1= 1 and V_3/V_1= 5 are shown in Fig. 4(c) and 4(d). It can be seen that also for V_3/V_1= ratio upto 1 the velocity profile remains same and after that it changes drastically. Also it can be seen that for V_3/V_1

ratio greater than 1, V_o/V_1 ratio is maximum between approximately Y/D_o= 0.2 to Y/D_o= 0.8. So it can be concluded that for both V_3/V_1 and V_2/V_1 ratio less than 1 flow become smooth and ordered which indicate the characteristics of laminar flow.

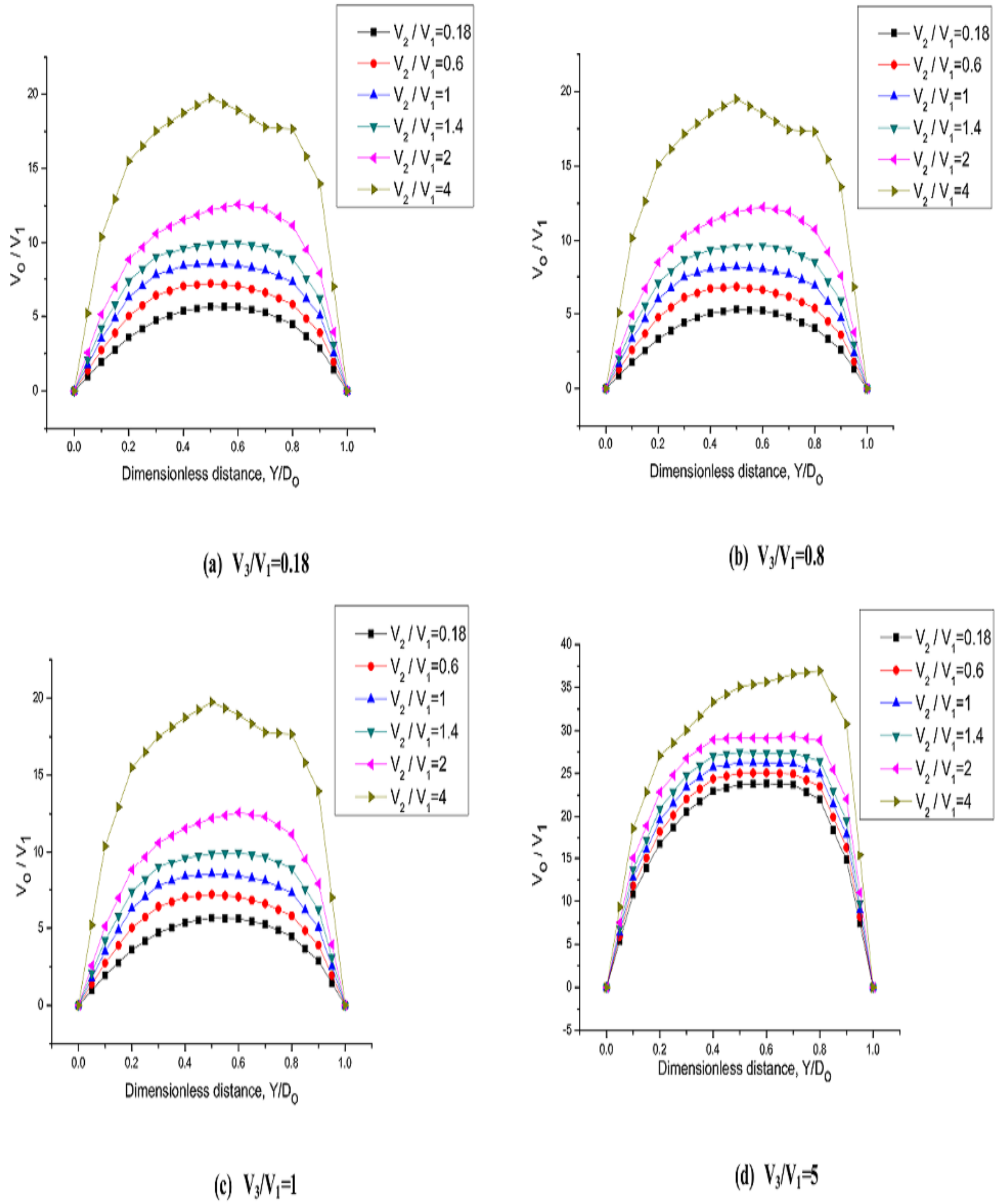

Fig. 4. Variation of outlet velocity for different V_3/V_1 ratio

The vector plot of velocity distribution inside the pipe model has been presented for different V_3/V_1 ratios with respect to V_2/V_1 = 1 in Fig. 5. The vector plot for V_3/V_1 = 0.18 shown in Fig. 5(a) shows that for V_3/V_1 ratio less than V_2/V_1 ratio flow is dominated by V_2/V_1 ratio and losses in the pipe appears to be small. When inlet pipes velocity is same i.e. when V_3/V_1 = 1 and V_2/V_1 = 1 shown Fig. 5(b) velocity profile throughout the pipe started to change this may be because of the effects sudden enlargement and sudden contraction presence at various position in the pipe becomes dominant.. From the vector plot for V_3/V_1 = 1.6 shown in Fig. 5(c) it appears that for V_3/V_1 ratio larger than V_3/V_1 flow aberrates from the smooth and ordered characteristics and higher the V_3/V_1 ratio becomes higher the aberration which can be clearly seen from the Fig. 5(d).

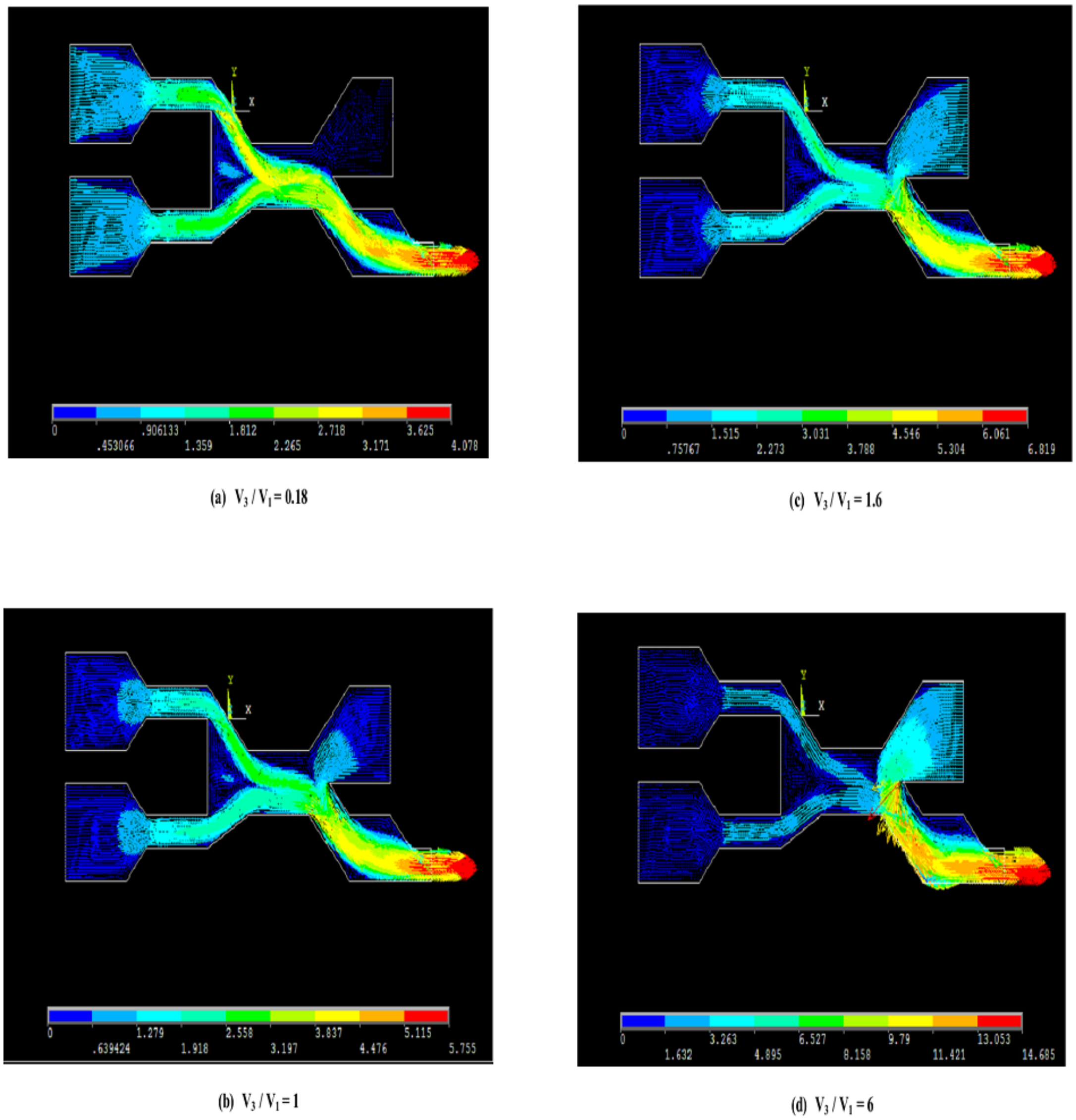

Fig. 5. Vector plot of velocity distribution inside the pipe for $V_2 / V_1 = 1$

6. Conclusions

The present study provides the solution for velocity and the pattern of flow in a complex pipe system using finite element approach. Both quantitative and qualitative results of pressure vessel obtained which establish the soundness and appropriateness of the present finite element approach. This solution technique is valid for any complex system of pipe.

7. References

[1] Navtej Singh, Suneev Anil Bansal, N.K Batra, "Visualization of Flow Behavior and Its Effected Contour in Sudden Contraction, Sudden Enlargement and Sudden Elbow By ANSYS", International Journal of Engineering and Research Applications (IJERA), Vol.2, ISSUE 5, ISSN: 2249-3905, 2012.

[2] D Rohitendra K Singh, "A Study of Air Flow in a Network of Pipes Used in Aspirated Smoke Detectors", M.Sc. Thesis, Victoria University, Australia, 2009.

[3] "ANSYS" Theory Manual and Expanded ANSYS Workbook.