

CFD Analysis of Flow through T-Junction An overview

G.B.Nimadge

M.E. Student

Babasaheb Naik College of Engineering, Pusad, Dist- Yavatmal

Abstract— Study of different research papers related to flow of fluid through T-Junction of pipes, different losses suffered while passing through T-junction and reliability of classical formulas is carried out. T-junction is very common and important component of piping system. The present work focuses on quantifying the energy losses and the size and strength of recirculating zone. The flow in T-junction are highly complex and three dimensional, therefore requiring numerical or experimental treatment. So help of advance software's such as ANSYS will help to simulate such complex flow.

Key words: T-junction, Energy losses, ANSYS

I. INTRODUCTION

Pipe networks are very common in industries, where fluid or gases to be transported from one location to other. The energy loss may vary depending on the type of components coming across the network, material of pipe and the fluid that is being transported through the network. The aim of this review is to find out the effect of combining flow and dividing flow at T-junction and also affect of angle and material of pipe on energy loss. Using software we can have better view of flow inside the junction and study turbulence, kinetic energy, pressure loss etc. Such simulations save a lot of time and can be performed without actually doing the experiment. Depending on the inflow and outflow directions, the behavior of flow at the junction also changes. The following figure shows some possibilities of fluid entering and leaving the junction.

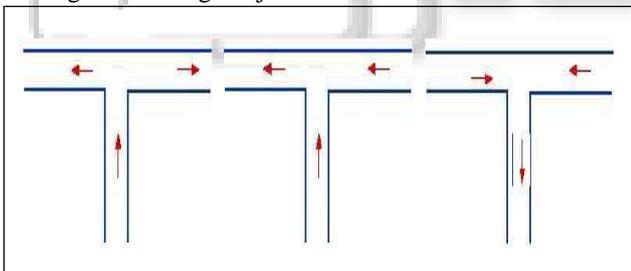


Fig. 1: Possibilities of fluid entering and leaving the Junction.

In fluid dynamics, head is the difference in elevation between two points in a column of fluid, and the resulting pressure of the fluid at the lower point. It is possible to express head in either units of height (e.g. meters) or in units of pressure such as Pascal's. When considering a flow, one says that head is lost if energy is dissipated, usually through turbulence; equations such as the Darcy-Weisbach equation have been used to calculate the head loss due to friction. Head losses are of two types major and minor. Major head losses (also called Frictional losses) are due to rough internal surface of pipe and occur over length of pipe. They are mainly due to friction. Minor losses are losses due to the change in fluid momentum. They are mainly due to pipe components due to bends, valves, sudden changes in pipe diameter, etc. Minor losses are usually negligible compared to friction losses in larger pipe systems.

II. COMPUTATIONAL FLUID DYNAMICS

Computational fluid dynamics, usually abbreviated as CFD, is a branch of fluid mechanics that uses numerical analysis and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. The Navier–Stokes equations are fundamental basis of almost all CFD problems. Methodology used in different approaches of CFD analysis includes

- 1) Defining the problem i.e. preprocessing.
- 2) Division of volume occupied by fluid into mesh that may be uniform or non-uniform.
- 3) After that boundary conditions are defined and simulation is done.
- 4) Finally postprocessor is used for analysis and visualization of results.

CFD deals with approximate numerical solution of governing equations based on the fundamental conservation laws of physics, namely mass, momentum and energy conservation. The CFD solution involves, Conversion of the governing equations for a continuum medium into a set of discrete algebraic equations using a process called discretization. Solution of the discrete equations can using a high speed digital computer to obtain the numerical solution to desired level of accuracy. [7]

III. NEED FOR CFD

Applying the fundamental laws of mechanics to a fluid gives the governing equations for a fluid. The conservation of mass equation and the conservation of momentum. These equations along with the conservation of energy equation form a set of coupled, nonlinear partial differential equations. It is not possible to solve these equations analytically for most engineering problems. However, it is possible to obtain approximate computer-based solutions to the governing equations for a variety of engineering problems. This is the subject matter of Computational Fluid Dynamics (CFD).

Pipes and ducts are the veins and arteries of mechanical systems such as power-plants, refineries, or HVAC systems. Without them, these systems could not exist. As in our own bodies, where the veins and arteries move blood through the pumping action of the heart, power plants require the circulation of a “working” fluid in order to provide its functionality. To increase the efficiency of such systems CFD is a useful tool.

IV. APPLICATIONS

CFD is being used for fundamental research as well as industrial R&D. CFD analysis forms an integral part of design cycle in most of the industries: from aerospace, chemical and transportation to bio-medical engineering. The length scales range from planetary boundary layers to

micro-channels in electronic equipments. Following is a short-list of some of more prominent applications of CFD:

- 1) Meteorology: weather forecasting.
- 2) Aerospace: design of wings to complete aircraft aerodynamic design.
- 3) Turbo machines: design of hydraulic, steam, gas, and wind turbines; design of pumps, compressors, blower, fans, diffusers, nozzles.
- 4) Engines: combustion modeling in internal combustion engines.
- 5) Electronics: cooling of micro-circuits.
- 6) Chemical process engineering.
- 7) Energy systems: analysis of thermal and nuclear power plants, modeling of accident situations for nuclear reactors.

V. CFD ANALYSIS

In CFD analysis of T-junction different authors have carried out their work by using different techniques. In experimental study of fluid mixing phenomena in T-junction, Hiroshi Ogawa specially mentioned some accidents caused by thermal fatigue, case of leakage in light water reactors (Japanese PWR Tomari-2 in 2003, French PWR CIVAUX in 1998) and also in sodium cooled reactors (French FBR PHENIX in 1992). According to his research mixing Tee is a component where temperature fluctuation occurs. A water experiment is performed in order to study influence of upstream elbow in main pipe near T-junction. Fluid temperature and velocity distributions in the pipe were measured by using a movable thermocouple and high speed PIV (particle image velocimetry). The flow pattern was classified into three patterns, wall jet, deflecting jet and impinging jet, measured flow velocity showed that biased flow velocity distribution and fluctuation due to the elbow influenced bending of branch pipe jet and the temperature fluctuation intensity around the jet. [3]

Recently computational fluid dynamics has been executed to investigate the transition boundaries of different flow patterns for oil-water mixing phenomena. Anand B. Desamala performed the geometry and meshing of the problem in GAMBIT and ANSYS FLUENT. Water lubrication technique is used to reduce the friction during the flow in pipelines of highly viscous oils. Classical formulas such as continuity equation, momentum equation all these equations are solved by volume of flow (VOF) method in fluent. Y-shaped fitting is also found in many pipelines. [1] Work on CFD analysis of Y-shaped branched pipe, including the effect of angle of turn or bend on it is carried out. To study the effect of angle only, Prof. Balvinder Singh and Prof. Harpreet Singh in their work considered all the three pipe branches if 1 inch internal diameter. After studying the effect of bend angle, pipe diameter, pipe length Reynolds number on resistance coefficient. They come to the conclusion that resistance coefficient vary with change in flow parameters. In CFD analysis of Y-shape joint three angles of 60° , 90° and 180° are selected. It is observed that angle of 45° a standard angle for Y-shape joint has minimum losses. [4]

The work carried out up till now includes CFD analysis of joints in pipe and comparison of analysis results with experimental or theoretical results. In most of the cases water experiments are carried out and in some cases oil and

water mixture is used. In CFD Simulation of a junction, investigating hydraulic properties of a T-junction, five operating conditions with different directions of flow were investigated. Boris Huber, constructed physical model for that purpose, where diameter of pipes is 192 mm in reservoir and turbine branches & 170mm in branch coming from the pump. The T-junction and the adjacent pipes were made of Plexiglas. The FLUENT 6.1.22 is used for numerical simulation. Nature scale reading and simulation readings were compared and hydraulic properties of investigated T-junction proved to be very satisfactory, with comparatively small loss coefficients.[2] One more analysis which focuses on the characterization of complex mixing phenomena at pipe intersections within pressurized water distribution networks, R.G.Austin and B.van Bloemen made an experimental setup consists of a cross junction with various sensors, pumps, and a data acquisition system to accurately measure solute concentration. Selected experimental results are compared to computational fluid dynamics CFD results. It shows that the complete mixing assumption can potentially create considerable errors in water quality modeling in particular with systems consisting of many cross junctions. This error is due to bifurcation of the incoming flows rather than perfect mixing. [5] Prediction of flows in 90° T-junction is performed numerically and results were compared with experimental results. Gyorgy Paal, Fernando T Pinho and Rodrigo Maia in their study observed that main cause of losses in T-junction are separation of flow and sharp corners. Conservation of mass and momentum were considered as governing equations. All the predictions are made by considering the flow as turbulent. For software part commercial code CFX 5.5 and for numerical simulations a full Reynolds Stress Model (RSM) and $k-\omega$ model are used. [3]

VI. ADVANTAGES

- 1) Using computational fluid dynamics we can predict the performance before modifying the actual setup.
- 2) It saves time and cost.
- 3) Analysis of complex three dimensional flows.
- 4) It can be used for virtually any problem and realistic operating conditions.

VII. CONCLUSION

The summary of present literature review is as follows, Losses in piping system are typically categorized as major and minor loss, particularly for T-unction, the angle between the branches forming T-junction plays an important role in energy losses. There is scope of improvement in the geometry. The major losses are due to material used for pipe i.e. friction loss, so by CFD analysis of T-junction and simulation of fluid flow proper material can be suggested for the pipe.

REFERENCES

- [1] Anand B. Desamala, "CFD Simulation and Validation of Flow Pattern Transition Boundaries during Moderately Viscous Oil-Water Two-Phase Flow through Horizontal Pipeline", World Academy of Science, Engineering and Technology 73 2013.

- [2] Boris Huber, CFD simulation of a T-junction, Institute of Hydraulic and Water Resources Engineering, Department of Hydraulic Engineering, Vienna University of technology, Austria.
- [3] Gyorgy Paal, Fernando T Pinho and Rodrigo Maia, "NUMERICAL PREDICTIONS OF TURBULENT FLOW IN A 90° TEE JUNCTION" The 12th International Conference on Fluid Flow Technologies Budapest, Hungary, September 3 - 6, 2003
- [4] Hiroshi Ogawa, "Experimental Study on Fluid Mixing Phenomena In T-Pipe Junction with Upstream Elbow", The 11th International Topical Meeting on Nuclear Reactor Thermal-Hydraulics (NURETH-11) Popes Palace Conference Center, Avignon, France, 2005.
- [5] Prof. Balvinder Singh, Prof Harpreet Singh "CFD analysis of Fluid Flow Parameters within A Y-Shaped Branched Pipe", International Journal of Latest Trends in Engineering and Technology (IJLTET).
- [6] R.G. Austin, B. van Bloemen, Waanders S. McKenna and C. Y. Choi, Mixing at Cross Junctions in Water Distribution Systems II, Journal Of Water Resources Planning And Management © Asce / May/June 2008.
- [7] <http://www.nptel.ac.in/courses>

