ORIGINAL RESEARCH PAPER

Engineering

CFD ANALYSIS ON DIFFERENT GEOMETRIES OF VENTURIMETER BY USING FLUENT

KEY WORDS: Venturimeter , CFD, Fluent , etc

Dr. K. Sudhakar Mech., Dean & Professor, Sree Visvesvaraya Institute of Technology & Science, Chowdarpally, Mahabubnagar – 509 204.

ABSTRACT

Venturimeters are used for measuring mass flow rate flowing through the venturimeter. Its design or geometry is important factor. Here the Geometry of venturimeter is directly proportional to fluid flow parameters like pressure, velocity, and turbulence. In this project to analysis or compare three different geometries of venturimeter by describing fluid flow parameters. The necessary numerical computations are accomplished by Fluent software product of Ansys and computational fluid dynamics (CFD) principles was used to analysis and plot the parameters of the flow of fluid through the different geometries of the venturimeter. Finally the result and conclusion are presented.

INTRODUCTION

A **venturimeter** is a device used for measuring the rate of a flow of a fluid flowing through a pipe.
It consists of three parts.

1.short converging part
2.Throat
3.Diverging part

It is based on the principle of Bernoulli's equation. Inside of the venturimeter pressure difference is created by reducing the cross-sectional area of the flow passage. The pressure difference is measured by using a differential U-tube manometer. This pressure difference helps in the determination of rate of flow of fluid or discharge through the pipe line. As the inlet area of the venturi is large than at the throat, the velocity at the throat increases resulting in decrease of pressure. By this, a pressure difference is created between the inlet and the throat of the venturi. When a fluid, whose flow rate is to be determined, is passed through a Venturi meter, there is a drop in the pressure between the Inlet section and Cylindrical Throat of Venturi meter. The drop in pressure can be measured using a differential pressure measuring instrument. Since this differential pressure is in direct proportion to the flow rate as per the Bernoulli's Equation hence the differential pressure instrument can be configured to display flow rate instead of showing differential pressure.

OPERATION OF VENTURIMETER

- The fluid flows inside the Inlet section of the Venturi meter having a pressure P1.
- As the fluid proceeds further into the Converging section, its pressure reduces gradually and it finally reaches a value of P2 at the end of the Converging section and enter the Cylindrical section.

The differential pressure sensor connected between the Inlet and the and the Cylindrical Throat section of the Venturi meter displays the difference in pressure (P1-P2). This difference in pressure is in direct proportion to the flow rate of the liquid flowing through the Venturi meter.

- Further the fluid passed through the Diverging recovery cone section and the velocity reduces thereby it regains its pressures. Designing a lesser angle of the Diverging recovery section, helps more in regaining the kinetic energy of the liquid.

The Venturi effect is a jet effect; as with a funnel the velocity of the fluid increases as the cross sectional area decreases, with the static pressure correspondingly decreasing. According to the laws governing fluid dynamics, a fluid's velocity must increase as it passes through a constriction to satisfy the principle of continuity, while its pressure must decrease to satisfy the principle of conservation of mechanical energy. Thus any gain in kinetic energy a fluid may accrue due to its increased velocity through a constriction is negated by a drop in pressure. When a fluid such as water flows through a tube that narrows to a smaller diameter, the partial restriction causes a higher pressure at the inlet than that at the narrow end. This pressure difference causes the fluid to accelerate toward the low pressure narrow section, in which it thus maintains a higher speed. The Venturi meter uses the direct relationship between pressure difference and fluid speeds to determine the volumetric flow rate.

The Venturi effect is a special case of Bernoulli's principle, in the case of fluid or air flow through a tube or pipe with a constriction in it. Bernoulli's principle can be derived from the principle of conservation of energy. This states that, in a steady flow, the sum of all forms of mechanical energy in a fluid along a streamline is the same at all points on that streamline. This requires that the sum of kinetic energy and potential energy remain constant. Thus an increase in the speed of the fluid occurs proportionately with an increase in both its dynamic pressure and kinetic energy, and a decrease in its static pressure and potential energy. If the fluid is flowing out of a reservoir, the sum of all forms of energy is the same on all streamlines because in a reservoir the energy per unit volume (the sum of pressure and gravitational potential g h) is the same everywhere [7]. Bernoulli's principle can also be derived directly from Newton's 2nd law. If a small volume of fluid is flowing horizontally from a region of high pressure to a region of low pressure, then there is more pressure behind than in front. This gives a net force on the volume, accelerating it along the streamline. Fluid particles are subject only to pressure and their own weight. If a fluid is flowing horizontally and along a section of a streamline, where the speed increases it can only be because the fluid on that section has moved from a region of higher pressure to a region of lower pressure.

FINITE ELEMENT METHOD

The basic idea in the Finite Element Method is to find the solution of complicated problem with relatively easy way. The Finite Element Method has been a powerful tool for the numerical solution of a wide range of engineering problems. Applications range from deformation and stress analysis of automotive, aircraft, building, defense, and missile and bridge structures to the field analysis of dynamics, stability, fracture mechanics, heat flux, fluid flow, magnetic flux, seepage and other flow problems. With the advances in computer technology and CAD systems, complex problems can be modeled with relative ease. Several alternate configurations can be tried out on a computer before the first prototype is built. The basics in engineering field are must to idealize the given structure for the required behavior. The proven knowledge in the computational aspects of the Finite Element Method is essential. In the Finite Element Method, the solution region is connected as built up of many small, interconnected sub regions called finite elements.

COMPUTATIONAL FLUID DYNAMICS

Computational Fluid Dynamics (CFD) is a computer-based tool for simulating the behavior of systems involving fluid flow, heat transfer, and other related physical processes. It works by solving the equations of fluid flow (in a special form) over a region of interest, with specified (known) conditions on the boundary of that region. The set of equations that describe the processes of momentum, heat and mass transfer are known as the Navier-Stokes equations. These partial differential equations were derived in the early nineteenth century and have no known general analytical solution but can be discretized and solved numerically.

Equations describing other processes, such as combustion, can also be solved in conjunction with the Navier-Stokes equations. Often, an approximating model is used to derive these additional equations, turbulence models being a particularly important example.

There are a number of different solution methods that are used in CFD codes. The most common, and the one on which CFX is based, is known as the finite volume technique. In this technique, the region of interest is divided into small sub-regions, called control volumes. The equations are discretized and solved iteratively for each control volume. As a result, an approximation of the value of each variable at specific points throughout the domain can be obtained. In this way, one derives a full picture of the behavior of the flow.

PROCEDURE
GEOMENTRY
MESH
FLUENT SETUP
SOLUTION
RESULTS
MODELING AND RESULTS

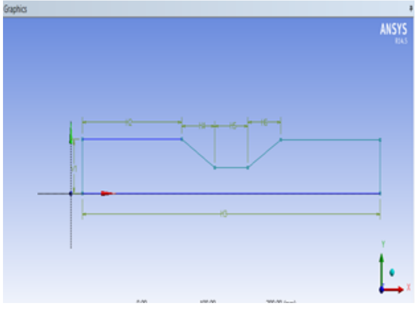


Figure 1 SKETCHING

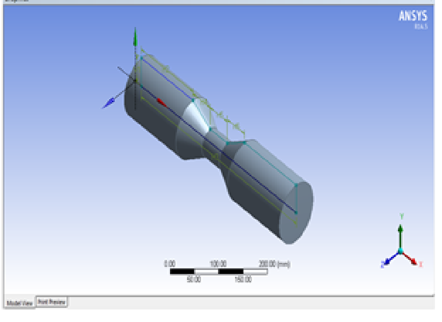


Figure 2 3D MODEL

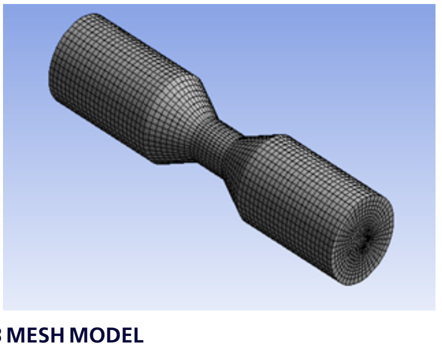


Figure 3 MESH MODEL

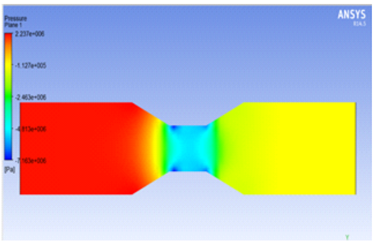


Figure 4 PRESSURE DISTRIBUTION GEOMENTRY 1

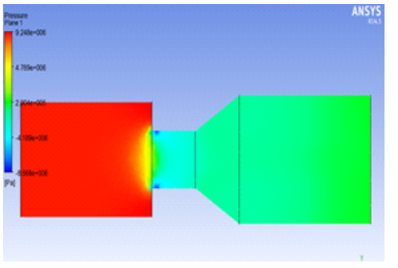


Figure 5 PRESSURE DISTRIBUTION GEOMENTRY 2

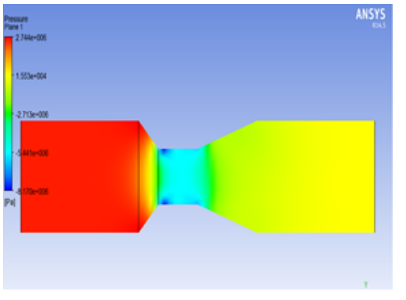


Figure 6 PRESSURE DISTRIBUTION GEOMENTRY 3

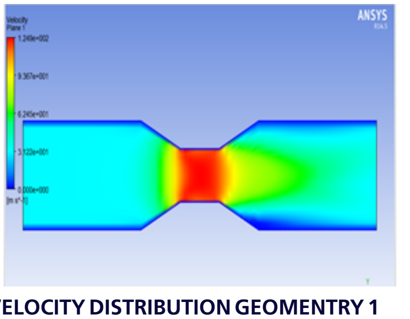


Figure 7 VELOCITY DISTRIBUTION GEOMENTRY 1

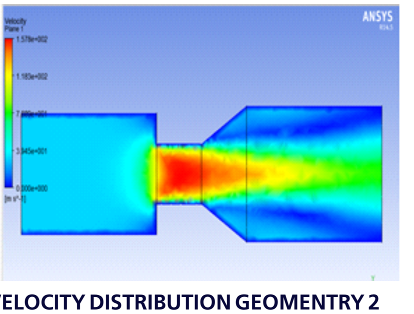
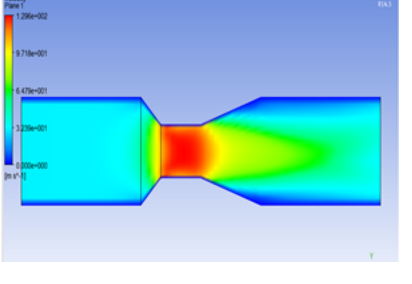


Figure 8 VELOCITY DISTRIBUTION GEOMENTRY 2



VOLUME-6 | ISSUE-7 | JULY-2017 | ISSN - 2250-1991 | IF : 5.761 | IC Value : 79.96

Figure 9 VELOCITY DISRIBUTION GEOMENTRY 3

CONCLUSION

During this project we have acquired knowledge over Ansys CFD and its respective solvers, by utilizing our knowledge over Ansys CFD and Fluent solver which is a kind of CFD solver which is used for numerical calculations, we have analyzed three different geometries of the venturimeter and we have represented results in the form of contour plots.

FUTURE SCOPE

It will be a complete guidance for solving fluidity problems under any flow movements.
CFD is a great science and has a great future,
-Fighters, bombers, cargo aircrafts,
-Helicopters,
-Air breathing engines,

Through these CFD analysis we can make sure that this analysis are very useful in design of venturimeter for different applications.

REFERENCES

The sites which were used while doing this project
1. Ansys and fluent study guides.
2. www.wikipedia.com
3. www.scribd.com
4. www.slideshare.com