

“Review Study on Analysis of Venturimeter using Computational Fluid Dynamics (CFD) for Performance Improvement”

Manish R. Bhatkar¹, Preeti V. Ban²

¹Assistant Professor at Department of Civil Engineering, Jagadambha College of Engineering and Technology Yavatmal Maharashtra India

²Assistant Professor at Department of Civil Engineering, Jagadambha College of Engineering and Technology Yavatmal Maharashtra India

Abstract— Venturimeter is a device which is used for measuring rate of flow of fluid flowing through pipes. The application of these are found in various fields like Aviation, Automotive, chemical, petro chemical industries, etc. In automotive industry venturimeter is used to measure the fuel and air distribution in carburetor. In this paper we study the CFD and experimental calculations on venturimeter. In the present work, an attempt was made to study, prepare a computational model of a venturimeter, which can be used as an efficient and easy means for predicting the discharge coefficients for that computational fluid dynamics (CFD) software has been used as to perform the modeling and simulation of venturimeter. Simulation was carried out for a standard venturimeter and the results were compare with the standards.

Keywords: venturimeter, CFD

1. Introduction

1.1 General

A venturimeter is a device used for measuring the rate of flow of a fluid flowing through a pipe.



Fig.No.1. Venturimeter.

A short converging part: It is that portion of the venturi where the fluid gets converges.

Throat: It is the portion that lies in between the converging and diverging part of the venturi. The cross section of the throat is much less than the cross section of the converging and diverging parts. As the fluid enters in the throat, its velocity increases and pressure decreases.

Diverging part: It is the portion of the venturimeter (venturi) where the fluid gets diverges.

1.2 Principle of Venturimeter

The working of venturimeter is based on the principle of Bernoulli's equation. Bernoulli's Statement: It states that in a steady, ideal flow of an incompressible fluid, the total energy at any point of the fluid is constant. The total energy consists of pressure energy, kinetic energy and potential energy or datum energy.

Mathematically

Pressure energy + Kinetic energy + Potential energy = Constant

$$\frac{p}{\rho g} + \frac{v^2}{2g} + z = \text{Constant}$$

Here all the energies are taken per unit weight of the fluid. The Bernoulli's equation for the fluid passing through the section 1 and 2 are given by

$$\frac{p_1}{\rho g} + \frac{v_1^2}{2g} + z_1 = \frac{p_2}{\rho g} + \frac{v_2^2}{2g} + z_2$$

1.3 Working

The venturimeter is used to measure the rate of flow of a fluid flowing through the pipes. Let's understand how it does this measurement step by step. Here we have considered two cross section, first at the inlet and the second one is at the throat. The difference in the pressure heads of these two sections is used to calculate the rate of flow through venturimeter. As the water enters at the inlet section i.e. in the converging part it converges and reaches to the throat. The throat has the uniform cross section area and least cross section area in the venturimeter. As the water enters in the throat its velocity gets increases and due to increase in the velocity the pressure drops to the minimum. Now there is a

pressure difference of the fluid at the two sections. At the section 1 (i.e. at the inlet) the pressure of the fluid is maximum and the velocity is minimum. And at the section 2 (at the throat) the velocity of the fluid is maximum and the pressure is minimum. The pressure difference at the two section can be seen in the manometer attached at both the section. This pressure difference is used to calculate the rate flow of a fluid flowing through a pipe.

1.4 Concept of Computational Fluid Dynamics

Computational Fluid Dynamics (CFD) is the simulation of fluids engineering systems using modeling (mathematical physical problem formulation) and numerical methods (discretization methods, solvers, numerical parameters, and grid generations, etc.). The process is as figure .

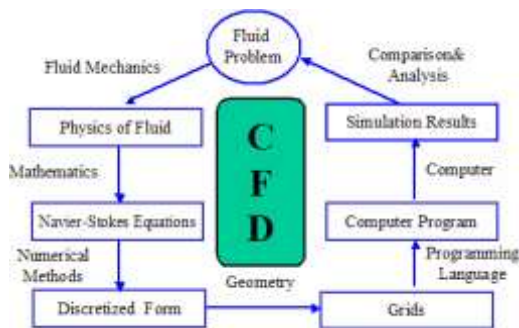


Fig.No.2 Flow Chart for CFD.

Firstly, we have a fluid problem. To solve this problem, we should know the physical properties of fluid by using Fluid Mechanics. Then we can use mathematical equations to describe these physical properties. This is Navier-Stokes Equation and it is the governing equation of CFD. As the Navier-Stokes Equation is analytical, human can understand it and solve them on a piece of paper. But if we want to solve this equation by computer, we have to translate it to the discretized form. The translators are numerical discretization methods, such as Finite Difference, Finite Element, Finite Volume methods. Consequently, we also need to divide our whole problem domain into many small parts because our discretization is based on them. Then, we can write programs to solve them. The typical languages are Fortran and C. Normally the programs are run on workstations or supercomputers. At the end, we can get our simulation results. We can compare and analyze the simulation results with experiments and the real problem. If the results are not sufficient to solve the problem, we have to repeat the process until find satisfied solution. This is the process of CFD.

TABLE 1 Advantages of CFD

	Simulation (CFD)	Experiment
Cost	Cheap	Expensive
Time	Short	Long
Scale	Any	Small/Middle
Information	All	Measured Point
Repeatable	Yes	Some
Safety	Yes	Some Dangerous

2. LITERATURE REVIEW

Patel Mitesh B (2018) In this paper we study conceptualization of this project is inspired by the experiments conducted for the calibration of Venturimeter and the loss of water head at the downstream of pipe flow in various hydraulic power plants. Also Venturimeter, are widely used in industry for flow measurements. In this present work, Computational Fluid Dynamics (CFD) has been used to compute the permanent pressure loss and relative pressure loss for 3D incompressible fluid for various designs of a classical Venturimeter. Further different parameters are defined to be varied to study effect of each in combination to minimize pressure drop in future work.

J. Ahamed Jahith (2019) In this paper we study and prepare the computational model of a venturimeter, which can be used as an efficient and easy for calibration of the instrument rather than of costly experimental methods. The research covers the following Points: to study the theory of the venturimeter and calculate the data theoretically by using Bernoulli's equation, to analyse the experimental data. The focus here is to analyse the pressure variations across the venturi section by means of Ansys Fluent 14.5, a commercial CFD code, which explores the use of computational methods to compute the flow parameters in the tube. The study aims at comparing the results calculated by both, the computational and experimental methods. And check the validity of Bernoulli's equation when applied to the steady flow of water in a tapered duct and to calibrate the venturi as a flow meter by calculating the coefficient of discharge.

H.Ameresh (2017) In this Paper we study that the dynamic flow analysis is to be carried by varying inlet diameter of venturimeter such as 25mm, 30mm and 35mm. Theoretical calculations are to be done for mass flow rate of air. And also obtained theoretical values are to be compared with ANSYS values. By using different inlet pressure the mass flow rates of Al2O3, water and air passing through various diameters of Venturimeter are to be calculated. In this work Unigraphics software is used for modeling of venturimeters with inlet diameters

of 25 mm, 30mm and 35mm. FEA software ANSYS is used for calculating mass flow rate values for venturimeters.

Jay Kumar(2014) In this paper we study that Venturi plays very Important Role in different field of engineering. Venturi has a number of industrial applications in which its design is important factor. One of the important factors that affect the fuel consumption is that design of venturi of carburettor. There is a need to design the Venturi with an effective analytical tool or software. In this work, there parameters namely pressure drop and Velocity discharge nozzle angle of the Venturi will be analyzed using computational fluid dynamics. For this analysis CFD will be done using two softwares namely GAMBIT and FLUENT. The results obtained from the softwares will be gives optimum result.

Arun R, (2015) In this paper we study that the venturimeter is a typical obstruction type flow meter, widely used in industries for flow measurements. The ISO standard (ISO-5167-1) provides the value of discharge coefficients for the classical venturimeters in turbulent flows with Reynolds number above 2×10^5 . But in case of viscous fluids, venturimeters are sometimes operated in laminar flow rather than turbulent flow at Reynolds number below the range covered by the standards. In this work, an attempt was made to study, prepare a computational model of a venturimeter, which can be used as an efficient and easy means for predicting the discharge coefficients at low Reynolds number. The computational fluid dynamics (CFD) software ANSYS FLUENT-14 has been used as a tool to perform the modeling and simulation of venturimeter. Simulation was carried out for a standard venturimeter and the results were validated with the standards. the results obtained from the simulations show that the discharge coefficient decreases rapidly as the Reynolds number decreases and also results were compared with the analytically proposed equation to calculate Cd at low Re and also same with the experimental data

Conclulsion

After the review of previous researches, it clear that CFD modeling and simulation was performed to calibration of venturimeter if CFD simulations could predict the performance of a venturimeter under non-ISO standard The results obtained from CFD were used to study the detailed information on venturimeter flow characteristics that could not be easily measured during experimental testing at very low Reynolds number. And lastely it is conclude that the computational model of a venturimeter, which can be used as an efficient and easy means for calibration of the instrument instead of costly experimental methods.

References

- [1] Patel Mitesh B "Analysis of venture-meter using Computational Fluid Dynamics (CFD) for performance improvement" Vol-4 Issue-2 2018 IJARIE
- [2] J. Ahamed EXPERIMENTAL STUDY ON FLOW THROUGH VENTURIMETER JIRJET Volume: 06 Issue: 03 | Mar 2019
- [3] H. Ameresh Investigation of Mass Flow Rate in Venturimeter Using CFD Vol. 7, Issue 12, (Part -7) December 2017 ijera
- [4] Jay Kumar, CFD Analysis of Flow Through Venturi IJRMET Vol. 4, Issue 2, Spl- 2 May - October 2014
- [5] Arun R Prediction of discharge coefficient of Venturimeter at low Reynolds numbers by analytical and CFD Method International Journal of Engineering and Technical Research (IJETR) Volume-3, Issue-5, May 2015