Effects of Geometrical Parameters on Performance Characteristics of Venturimeter

[1]Mohan Suryawanshi, [2]Akshay Bhinge, [3]Akshay Kokane, [4]Suryakant Dhaigude, [5]Rahul Bajaj

[1,2,3,4,5]Mechanical Engineering Department, Indira College of Engineering and Management.

Abstract: Most of the industries are concerned with flow measurement and accurate measurement of fluid is important to obtain specific proportion as per process requirement. Mostly venturimeter is used in industries because of well-known features offered by it. Venturimeter works on the principle of pressure difference i.e. Bernoulli's principle. By varying cross sectional area of flow, it creates pressure difference along its length which is used to calculate theoretical discharge. Head loss in venturimeter is very less as compared to other flow measuring devices. Because of its specific geometry eddies formation is avoided causing less head loss. Cd is an important parameter always referred in case of flow measuring devices. For venturimeter it varies from 0.95 to 0.99. By studying various parameters that affects the Cd efforts were done to have the best configuration of venturimeter for maximum Cd. CFD is tool used for the analysis of fluid flow. In this project efforts are being taken to improve the coefficient of discharge and reduce the frictional losses by experimental and CFD analysis of venturimeter.

Keywords: CFD, Venturimeter, SST and K-Epsilon model.

Nomenclature:

Subscripts

C_d: Coefficient of Discharge

d_p: level Difference in Venturimeter Limbs.

Greek Symbols

β: Dimentral Ratio(d/D)

Symbols

Small

d: Throat Diameter

Capital

D: Pipe Diameter

I. INTRODUCTION

Accurate flow measurement of fluid is most important in industries to obtain specific proportion as per process requirement. Flow measurement of various fluids is crucial like transporting materials from one point to another, mixing of materials, and chemical reactions etc.

Venturimeter is a converging diverging pipe with throat in between them. It consisted of a 21° conical contraction, a straight throat of diameter d and length l and then a 7 to 15° conical expansion. The discharge coefficient is near unity, and the non-recoverable loss is very small.

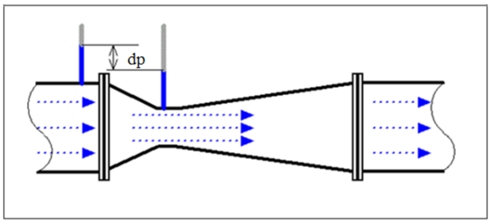

Figure 1 Venturimeter

In the venturi tube the fluid flow-rate is measured by reducing the cross-sectional flow area in the flow path, generating a pressure difference. After the constricted area, the fluid is passes through a pressure recovery exit section, where up to 90% of the differential pressure generated at the constricted area, is recovered. The venturi meter is an obstruction meter named in honor of Giovanni Venturi (1746–1822), an Italian physicist who first tested conical expansions and contractions. The original, or classical, venturi was invented by a U.S. engineer, Clemens Herschel, in 1898. The modern venturi nozzle consists of an ISA 1932 nozzle entrance and a conical expansion of half angle no greater than 15°.

Although experimental procedures offer a good result, they are often time consuming. Hence a more sophisticated method of testing the flow meter is through numerical methods. Due to variety of commercial CFD codes being available in the market, it is possible to obtain more accurate results which take less time. These results can then be compared with the initial experimental results to calibrate the instruments.

Number of studies have been performed for better knowing behaviour of venturimeter in different working conditions.

V. Subrahmanyam et al. (2014) [1] studied analytically four different models like venturimeter, orifice-meter, Rota-meter, nozzle to study the velocity, pressure, turbulence and mass flow rate and plotted graphs. Venturimeter are mostly used flow meters for measuring flow rate of fluid flowing through it. As mass flow rate and pressure drop are related to each other and hence variations of the pressure drop can be used to measure variations in the flow rate. He concluded pressure recovery better in venturimeter and FLUENT can be used with high degree of accuracy to visualize the various contours of velocity, pressure and turbulence.

Mohamed Aichouni et al. (2000) [2] did experimental investigation on the effects of non-standard operating conditions (approaching flow and internal geometrical irregularities) on the accuracy of the venturimeter. The results showed that the error caused by such non standards operating conditions can be very important which would lead to significant economic losses.

Nikhil Tamhankar et al. (1995) [3] did experimentation and CFD method and compare their result for the value of C_d. The study was aimed at comparing the results computed by both the computational and experimental methods. He concluded that the values of discharge coefficient obtained from experimental tests and CFD simulations were very close to each other.

Tarash Jain et al. (2014)[4] presented the three-dimensional analysis and optimization of the two types of venturimeter, viz. Circular Venturi and Slit Venturi on the basis of the various geometrical parameters using Computational Fluid Dynamics.

M.J. Reader-Harris et al. (2001) [5] manufactured 21 venturimeter in a range of diameter ratios from 0.4 to 0.75. Six were standard except for the convergent angles which were either 10.5° or 31.5°and rest were with dimensions around standard venturi. It was clear from data of gas measurement for the venturi tubes that convergent angle of 10.5° was much smoother than those from venturi tubes with the standard or the higher convergent angle.

Abhishek Kala et al. (2007) [6] experimentally studied pipes or conduit carrying sediment laden flow. Slurry-water mixture is very common in most of the industries, sewage carrying system etc. Suitability of flow meters i.e. venturimeter and flow meter need to be analyzed for sediment laden flow. Due to the presence of slurry or sediments, coefficient of discharge of flow meter will vary. Sediment laden flows generally create several problems because diluteness of solids in the liquid violets the continuums approach in the liquid - liquid flow analysis of conduits.

Brook et al. (1976) [7] analyzed the effect of solid liquid mixture in venturi and other meters. He carried out experiment in a non-standard type of venturi meter with a wide throat. He used the mixture of water and crushed Bakelite or water and basalt chippings. Better results were obtained in standard venturimeter.

Graf et al. (1967) [8] carried out an experiment on venturimeter to modify it for two phase flow. He used sand water mixture for the experiment by putting the venturimeter in horizontal position. He analyzed the data by plotting pressure drop against flow rate and a modified loss was correlated with sediment concentration.

Shook et al. (1982) [9] carried out experiment on stratified slurries flowing through horizontal venturi meters. It was found that for stratified slurries, the discharge coefficient was higher than that of homogeneous fluids. This effect was found to be increased by increasing slurry concentration and decreased by increasing the velocity of flow. By increasing concentration of slurry C_d of venturimeter increases.

F. Sanchez Silvao et al. (1991) [10] tested venturimeter in air-water and air-oil-water flow. The superficial velocities ranged up to 1.8 m/s for oil and water and up to 20 m/s for air. Conductance probes, made of two stainless steel rings mounted flush to the pipe wall and placed at different positions in the pipe and in the venturi throat allowed the measurement of the local liquid holdup for the case of air-water flow. Venturi nozzle can be employed for the measurement of the total mass flow-rate in two/three-phase flow systems.

II. EXPERIMENTATION

The experimental setup used was shown in fig. 2 below. Whole experimental setup is mounted on the water sump. Hydraulic pump is mounted on the sump. It delivers required water supply to whole apparatus.

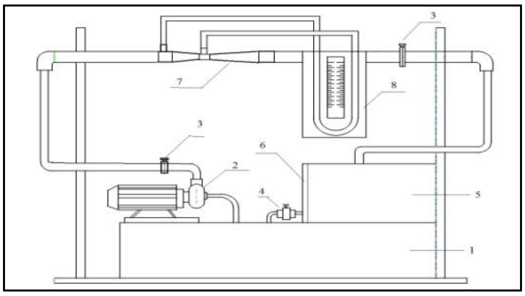

Figure 2 Experimental Setup of Venturimeter

[**1**-Sump Tank, **2**-Supply Pump, **3**-Flow Control Valve, **4**-Drain Valve, **5**-Collecting tank, **6**-Gauge Glass, **7**-Venturimeter, **8**-Manometer]

Water flow is maintained by the use of two flow control valves. One valve is attached at the outlet of pump and other at exit of venturimeter. Venturimeter is horizontally aligned. Manometer is attached between inlet and throat of venturimeter to measure pressure differential. Collecting tank is mounted on main water sump. It is used to measure actual discharge through venturimeter.

III. RESULTS AND DISCUSSION

We took two planes for pressure & velocity difference measurement at specified locations and contours obtained were as follows. Selected plane 1 was in the pipe region where it was needed to measure the pressure and velocity after CFD simulation. Also it was needed to measure pressure at throat section.

For accurate measurement it was needed that flow regime should be fully developed. So from entry in pipe section plane 1 was taken at a distance of 18 mm from inlet section of geometry while plane 2 was taken at a distance of 47 mm from inlet section so that flow reaching both the planes would be fully developed.

Table I. Location of Planes

Plane	Location	Distance in mm
Plane 1	Inlet Pipe	18
Plane 2	Throat	47

Pressure Contour

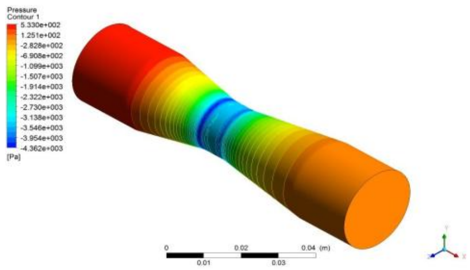

Figure 3 Pressure Contour SST Model [β=0.65]

Above pressure contour colour code in fig. 3 shows pressure distribution in various regions (i.e. convergent, throat, divergent & pipe). Pressure is maximum at inlet of pipe which is shown by red colour. In convergent section as area decreases gradually pressure decreases till throat region. In throat section area is constant due to which pressure is constant (i.e. minimum). In divergent section as area increases gradually pressure recovery takes place.

Difference in pressure between inlet & outlet of pipe because some pressure losses in sections is clearly visible in colour codes.

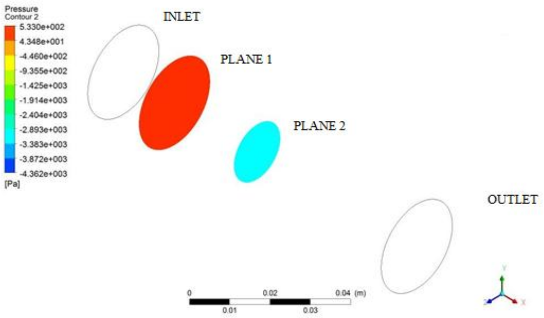

Figure 4 Pressure Contour at Specified Location, SST Model [β=0.65]

From contour shown in fig. 4 pressure at inlet is maximum and at throat is minimum because considering density of water across two planes constant, plane 1 have higher cross sectional area whereas plane 2 is at least cross sectional area.

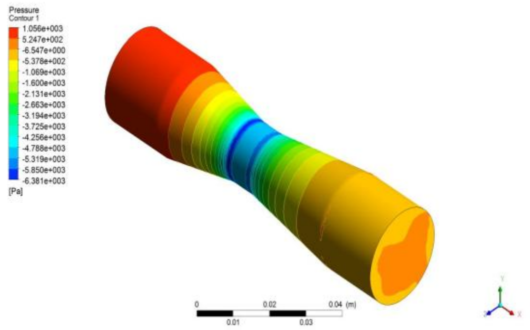

Figure 5 Pressure Contour k-ε Model [β=0.65]

Also, from Fig. 5 showing pressure contour it can be seen that inlet section is colored with orange color showing highest pressure value and at entry of throat section there is a blue color indicating least pressure value from pressure bar. It is because of cross-sectional area which is maximum at inlet and minimum at throat section. Pressure decreases with decrease in cross-sectional area.

Pressure contour shown in fig.6 below shows that there is difference in pressure between inlet & outlet of pipe because some pressure losses in sections.

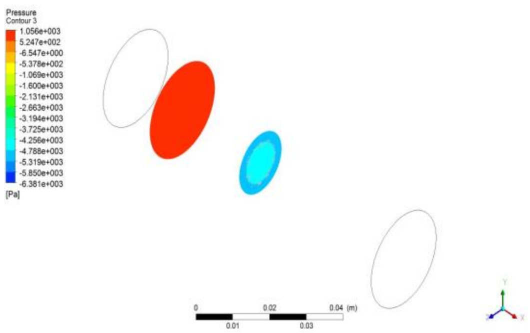

Figure 6 Pressure Contour at Specified Location, k-ε Model [β=0.65]

Pressure at inlet is maximum and at throat it is minimum because considering density of water across two planes constant, plane 1 has higher cross sectional area whereas plane 2 is at least cross sectional area.

Velocity Streamline

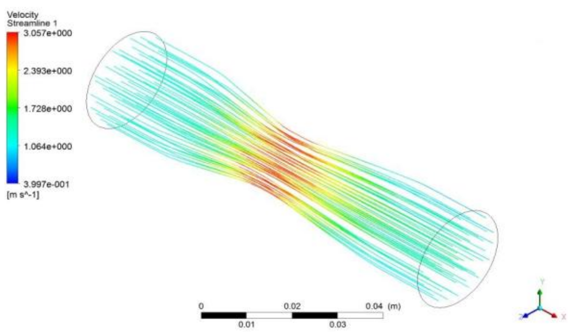

Figure 7 Velocity Streamline, SST Model [β=0.65]

From velocity streamlines in fig.7 and fig. 8 colour code shows velocity distribution in various regions (i.e. convergent, throat, divergent & pipe). Velocity is minimum at inlet of pipe which is shown by sky-blue streamlines. In convergent section as area decreases gradually velocity increases till throat. In throat section area is constant due to which velocity is constant (i.e. it is maximum). At outlet of diverging section streamlines become dark blue coloured indicating lesser velocity as compared to inlet section.

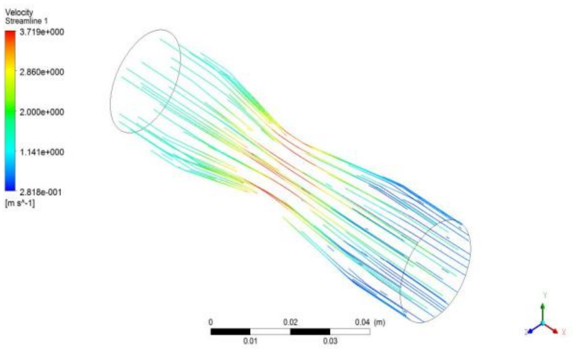

Figure 8 Velocity Streamline, k-ε Model [β=0.65]

Velocity Contour

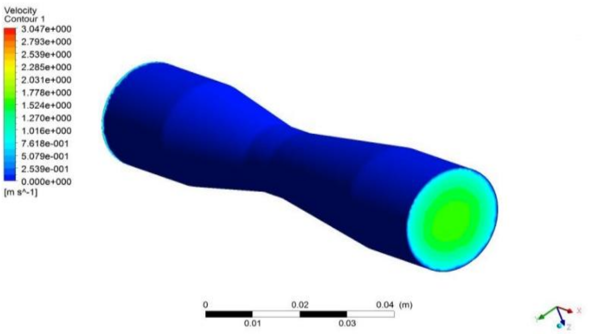

Figure 9 Velocity Contour, SST Model [β=0.65]

From above velocity contour color code in fig. 9 shows velocity distribution on wall is constant because of applied no slip boundary condition at wall.

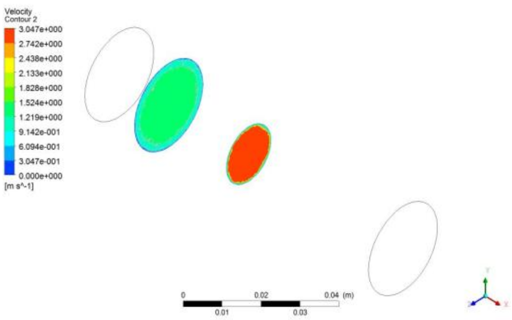

Figure 10 Velocity Contour at Specified Location, SST Model [β=0.65]

From contours in fig. 10 and fig. 11, it shows that at plane-1 velocity is less as compared to plane-2 (i.e. in throat). From continuity equation as area decreases velocity increases. At plane-1 cross sectional area is higher than throat section so velocity is lesser. Through convergent section area goes on decreasing and it has least value at throat section i.e. plane-2 so velocity is maximum at plane-2.

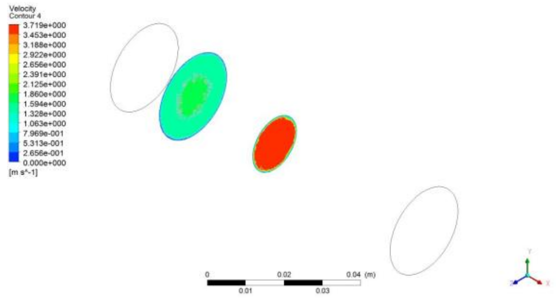

Figure 11 Velocity Contour at Specified Location, k-ε Model [β=0.65]

Results from Graph

Comparison of C_d between Experimental and CFD Analysis

From obtained results, graphs have been plotted for experimental and CFD analysis of venturimeter (for β=0.6).

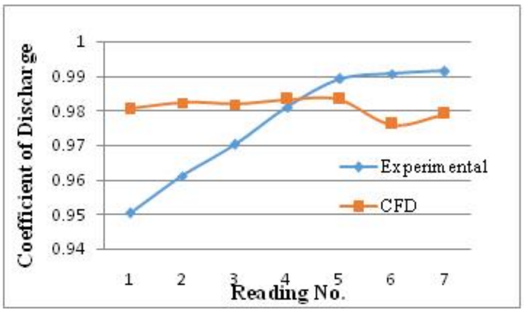

Figure 12 Comparison of Cd between Experimental and CFD Analysis

From fig. 12 it shows that Results obtained from experimental and CFD analyses are similar in about ±3% tolerance.

Coefficient of Discharge for Various β Values

By varying Geometrical parameter as follows:-

i. By changing throat diameter:- Firstly calculate Cd value for standard experimental setup whose β=0.6. Then we took random β values as 0.4, 0.5, 0.55, 0.65, 0.7, and 0.8 by varying throat diameter. We calculated Cd values for that various β values by using CFD tool.

ii. By changing length (i.e. Convergent, Divergent & throat) & Throat Diameter:- We took three different geometries by changing length of sections (i.e. Convergent, Divergent & throat) keeping total length of venturimeter constant (i.e. 63mm) according to length standards we changed further throat diameter.

For above combination C_d was calculated and plotted it as,

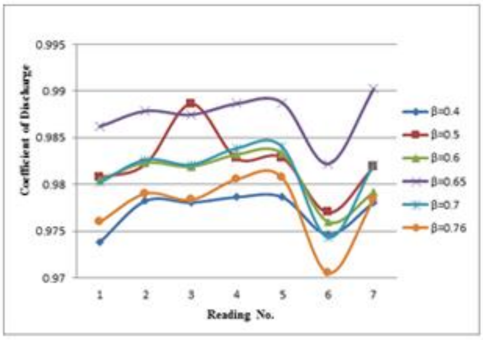

Figure 13 Coefficient of Discharge for Various β Values

From fig. 13 and results obtained from this work, coefficient of discharge is maximum at β=0.65.

C_d Vs Actual Discharge

As per calculation for coefficient of discharge for various β value we obtained best result for β=0.65.for this configuration we compare the C_d and actual discharge.

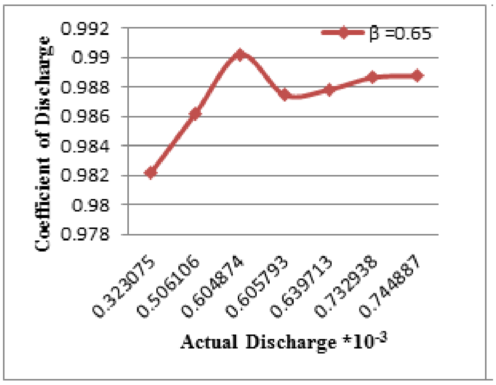

Figure 14 Comparison of Cd and Actual Discharge for β=0.65

Here fig. 14 shows Coefficient of discharge is maximum when actual discharge is 6.04874×10^{-4} m^3/sec.

Comparison of C_d and Reynolds Number for β=0.65

We took random values of Reynolds number from 1 to 100000, for this values we calculated velocity by keeping the parameters constant (i.e. density, diameter of pipe &viscosity), For this velocity we find coefficient of discharge.

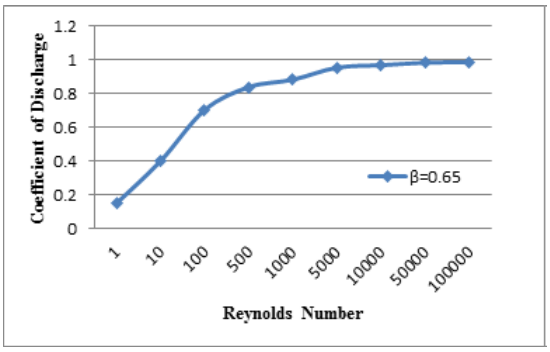

Figure 15 Cd Vs Reynolds Number

From above fig. 15 the value of coefficient of discharge increases with increase in Reynolds number.

Comparison between k-ε Model and SST Model.

Work done on two different models can be compared as,

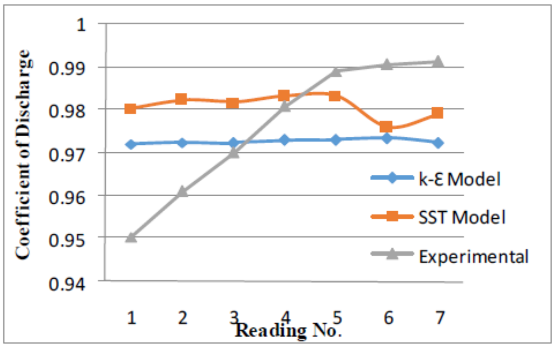

Figure 16 Comparison between k-ε model and SST model

Above fig. 16 shows error between k-ε models with respect to experimental is ± 2.27% and error between SST models with respect to experimental is ± 3.16%

IV. CONCLUSIONS

From this work conclusions can be drawn as follows:

i. As per results obtained from this work, coefficient of discharge is maximum at β=0.65.

ii. As per varying the inlet velocity and keeping the constant parameter (like density, viscosity & diameter) result obtained is that value of coefficient of discharge increases with increase in Reynolds number.

iii. Coefficient of discharge is maximum when actual discharge is $6.04874\times10^{-4}m^3$/sec.

iv. Results obtained from experimental and CFD analysis is in close proximity of ±3%.

v. Accuracy of flow measurement is increased.

vi. Due to superior geometry of venturimeter pressure loss due to flow measuring device is reduced.

vii. CFD approach reduces overall cost.

V. FUTURE SCOPE

From studying the previously done work and adding a bit of experimentation and CFD analysis more future work can be done on,

i. In venturimeter analysis working fluid for this experimental setup is water, for further analysis working fluid can be changed as per the applications such as oil, sediment etc.

ii. For different shapes of venturimeter the analysis can be performed and different results can be plotted.

iii. Analysis can be done for two phase mixture i.e. for steam and water mixture, air and water mixture.

REFERENCES

[1] V. Subrahmanyam, P. Hari Vijay "CFD Simulation on Different Geometries of Venturimeter" Internatonal Journal of Reserch in Engineering and Technology (IJRET), 2014.

[2] M. Aichouni, Harrison, S.T.L"Experimental Investigation of the Installation Effects on The Venturi Flowmeter Performance" Asme Fluids Engineering Conference, Jun. 2000.

[3] Nikhil T., Adarsha Hirijannaish "Experimental and CFD Analysis of Flow Through Venturimeter to Determine The Coefficient of Discharge " Internatinal Journal of Latest Trends in Engineering and Tech.(Ijltet), Vol. 3, 1995.

[4] T. Jain, J. Carpenter, V. Kumar Saharan "CFD Analysis and Optimization of Circular and Slit Venturi for Cavitational Activity" Journal of Material Science and Mechanical Engineering(Jmsme), 2014.

[5] M. J. Reader-Harris, W. C. Brunton, J. J. Gibson, D. Hodges, I. G. Nicholson "Discharge Coefficients of Venturi Tubes with Standard and Non-Standard Convergent Angles ," Flow Measurment and Instrumentation, Pp. 135-145, 2001.

[6] Abhishek Kala, Dr. S. K. Mittal, Prof M. K. Choudhary "Characteristics of Flow Meters with Sediment Laden Flow" International Journal of Engineering Reserch, Vol. 4, Pp. 240-243.

[7] N. Brook "Flow Measurement of Solidliquid" Proceedings of The Institution of Mechanical Engineers, Pp. 127-140, Jun. 1976.

[8] W. H. Graff "A Modified Venturimeter for Mesuring Two Phase Flow" Journal of Hydraulic Research, Pp. 161-187.

[9] C. A. Shook, J. H. Malshiya "Flow of Stratified Slurries Through Horizontal Venturimeter" The Canadian Journal of Chemical Engineering, Vol. 60, Jun. 1982.

[10] F. Sanchez Silvao, P. Andreussi, P. Di Marco "Total Mass Flow Rate Measurement in Multiphase Flow by Means of Venturimeter" Multiphase Production ,Elsevier, Pp. 145-155, 1991.

[11] http://www.engineeringtoolbox.com/flow-meters-d-493.html.

[12] http://en.m.wikipedia.org/wiki/different-types-of-boundary-condition-in-fluid-dymamics.

◈◈◈