NUMERICAL INVESTIGATION OF FLOW THROUGH MODIFIED VENTURI USING FLUENT

[1] Saravanan K G [2] Santhosh S [3] Gobikrishnan Udhayakumar

[1]Assistant Professor, Sona College of Technology, Salem, Tamil Nadu, India-636 005.

[2] PG student, Engineering Design, Sona College of Technology, Salem, Tamil Nadu, India-636 005.

[3]Assistant Professor, Sona College of Technology, Salem, Tamil Nadu, India-636 005.

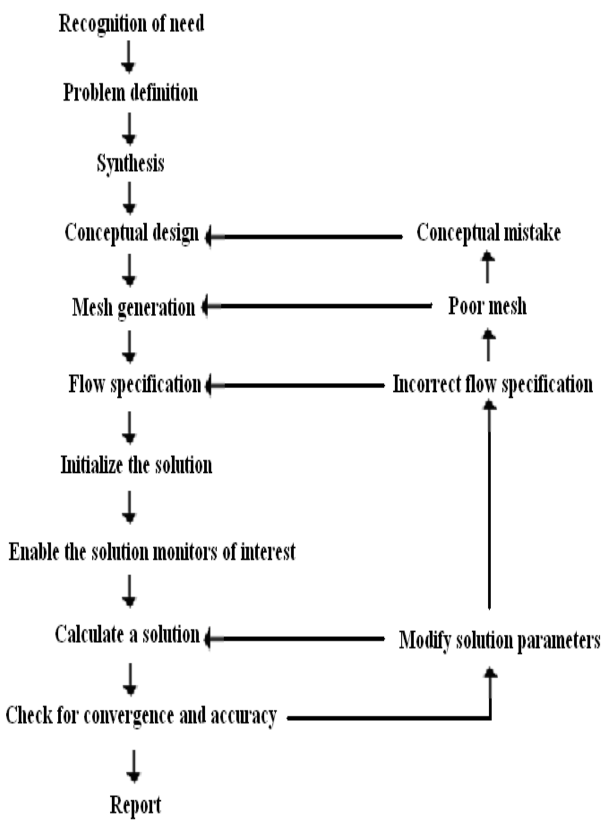
kgsmechanical@gmail.com

Abstract - **Mixing is a very crucial unit operation in process industry. The efficiency of the mixing depends on the type of agitator that will provide the required level of mixing in as short time as possible. Mixing time is usually the critical parameter in determining the efficiency of an agitated system. A CFD analysis yields values for species concentration, liquid velocity and temperature throughout the solution domain. This allows engineers to evaluate alternate designs and choose the optimum configuration. The industrial processing of rapid mixing of different solutions like dyes, chemicals, possessing different behavior properties were still achieved by using hydraulic equipments for separate solutions. This technique involves pumping each of the individual liquids and adjusting their flow rate to make them mix with correct proportions. This method can be improved further to meet the future requirements of conserving energy through the basic hydraulic principles. Prototype and pilot scale testing of stirred tank reactors can be a time consuming process and expensive process. In addition, many qualitative features of the flow fluid can be difficult to determine or measure experimentally.**

I. INTRODUCTION

The Physical aspects of any fluid flow are governed by three fundamental principles: Mass is conserved; Newton's second law and Energy is conserved. These fundamental principles can be expressed in terms of mathematical equations, which in their most general form are usually partial differential equations. Computational Fluid Dynamics (CFD) is the science of determining a numerical solution to the governing equations of fluid flow whilst advancing the solution through space or time to obtain a numerical description of the complete flow field of interest.**A.**

II.STEPS INVOLVED IN DESIGN AND ANALYSIS PROCESS



III.PROBLEM DEFINITION

Mixing can be carried out easily with less cost and reduced equipment using the venturi. Efficient mixing requires highly sophisticated devices.

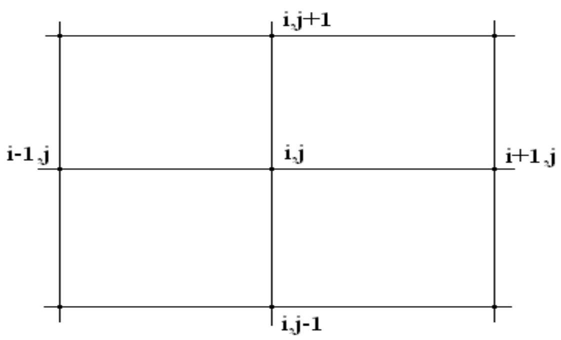
IV.WORKING METHODOLOGY OF CFD CODE

CFD codes are structured around the numerical algorithms that can tackle fluid flow problems. In order to provide easy access to their solving power all commercial CFD packages include sophisticated user interfaces to input problem parameters and to examine the results. Hence all codes contain three main elements.

1. Pre-processor
2. Solver
3. Post-processor

V.FINITE DIFFERENCE METHOD

Finite difference methods describe the unknown's F of the flow problem by means of point samples at the node points of a grid co-ordinate lines. Truncated Taylor series expansions are often used to generate finite difference approximations of derivatives of F in terms of the point samples F at each grid point and its immediate neighbors. Those derivatives appearing in the governing equations are replaced by finite differences yielding an algebraic equation for the values of F at each grid point.



VI.FINITE ELEMENT METHOD

Finite element methods use simple piecewise functions (e.g. linear or quadratic) valid on elements to describe the local variations of unknown flow variables F. The governing equation is precisely satisfied by the exact solution F. If the piecewise approximating functions for F are minimized in some sense by multiplying them by equations for the unknown coefficients of the approximating functions. The theory of finite elements has developed initially for structural analysis.

VII. SPECTRAL METHOD

Spectral methods approximate the unknowns by means of truncated Fourier series or series of Chebyshev polynomials. Unlike the finite difference or finite element approach the approximations are not local but valid throughout the entire computational domain. Again we replace the unknowns in the governing equation by the truncated series. The constraint that leads to algebraic equations for the coefficients of the Fourier or Chebyshev series is provided by a weighted residuals concept similar to the finite element method or by making the approximate function coincide with the exact solution at a number of grid points.

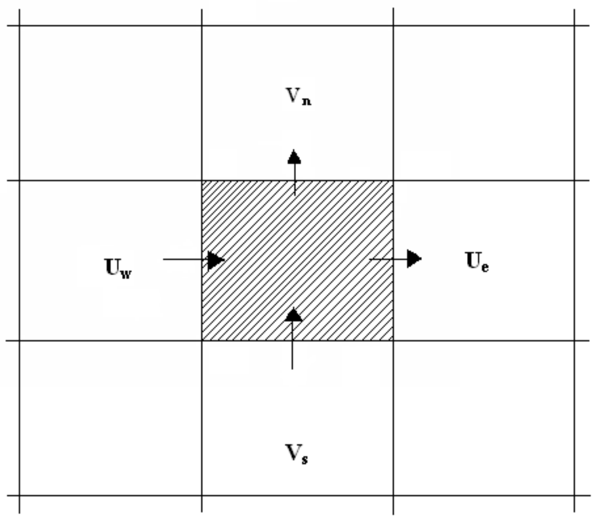
VII.FINITE VOLUME METHOD

The finite volume method was originally developed as a special finite difference formulation. It is central to four of the five main commercially available CFD codes: PHOENICS, FLUENT, FLOW3D and STAR-CD. The numerical algorithm consists of the following steps

- Formal integration of the governing equations of fluid flow over all the (finite) control volumes of the solution domain.
- Discertisation involves the substitution of a variety of finite difference type approximations for the terms in the

integrated equation representing flow processes such as convection, diffusion and sources. This converts the integral equations into a system of algebraic equations.

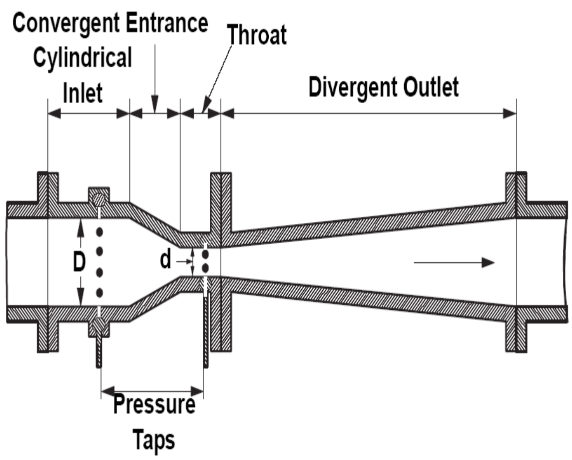
- Solution of the algebraic equations by an iterative method.



VIII.VENTURI METER AND ITS PRINCIPLE

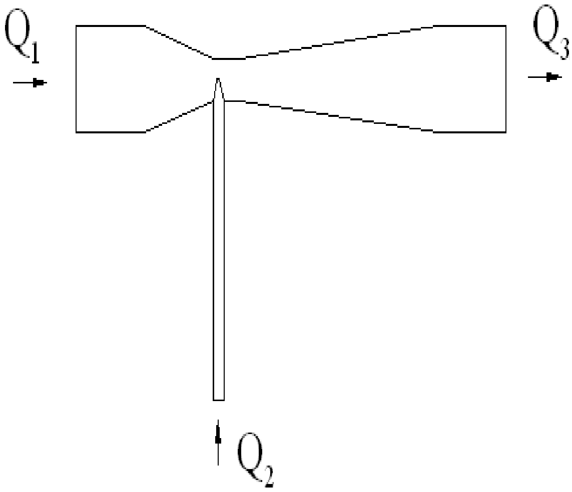
A venturi meter is a device used for measuring the rate of a flow of a fluid flowing through a pipe. It consists of three parts namely,

- A short converging part
- Throat
- Diverging part



A fluid passing through smoothly varying constrictions is subject to changes in velocity and pressure in order to satisfy the conservation of mass-flux (flow rate). The reduction in pressure in the constriction can be understood by conservation of energy: the fluid (or gas) gains kinetic energy as it enters the constriction, and that energy is supplied by a pressure gradient force from behind. The pressure gradient reduces the pressure in the constriction, in reaction to the acceleration. Likewise, as the fluid leaves the constriction, it is slowed by a pressure gradient force that raises the pressure back to the ambient level.

In this design a tube with a nozzle attached to the throat section. Since there is a low pressure at one end of the tube (throat section) and a comparatively high pressure at the other end of the tube, due to this pressure difference liquid is sucked in to the throat section. Using this principle mixing of liquid of two different temperatures can be obtained.



IX.PRANDTL NUMBER

Prandtl number is the ratio of momentum diffusivity to thermal diffusivity.

$$\mathrm{Pr} = \frac{\mu c_p}{k}$$

Where,

μ =Dynamic Viscosity of the fluid in Ns/m^2,

Cp=Co-efficient of thermal expansion in J/kgK,

k=Thermal conductivity in W/mK,

It provides a measure of the relative effectiveness of the momentum and energy transport by diffusion. Its value is near unity in most of the gases indicating that momentum and energy transfer by diffusion are of the same order.

X.REYNOLDS NUMBER

Reynolds number is defined as the ratio of inertia force to viscous force.

$$Re = \frac{\rho v D}{\mu}$$

Where,

ρ =Density of the fluid in kg/m^3,

v = Average Velocity in m/s,

μ = Dynamic Viscosity in Ns/m2,

Large values of Reynolds number denote high inertial forces where small values of Reynolds number are obtained with highly viscous fluids. Reynolds number determines the nature of the flow, whether laminar or turbulent.

XI. CONTOUR PLOTS

A disadvantage of xy plots as described above is that they usually do not illustrate the global nature of a set of CFD results all in one view. On the other hand, contour plots do provide such a global view.

A contour line is a line along which some property is constant. Contours are plotted such that difference between the quantitative values of dependent variable from one contour line to the adjacent contour line is held constant. In this fashion, in regions where the dependent variable is slowly changing in space, the adjacent contour lines, the adjacent contour lines are closely spaced together; in contrast, in region where the dependent variable is slowly changing in spaced, the adjacent lines are widely spaced.

Contour plots are clearly a superior graphical representation from this point of view. On the other hand, it requires more effort to read precise quantitative data from a contour plot as compared to a curve in a xy plot. Although each contour may be labeled as to the constant numerical values of the property it represents, obtaining of numerical values between contour lines requires some mental and/or numerical interpolation in space, an imprecise process to say the least.

The proliferation of contour plots with the advent of the computer is understandable; in CFD, contour plots are one of the most commonly found graphical representation of data.

XII.VECTOR PLOTS

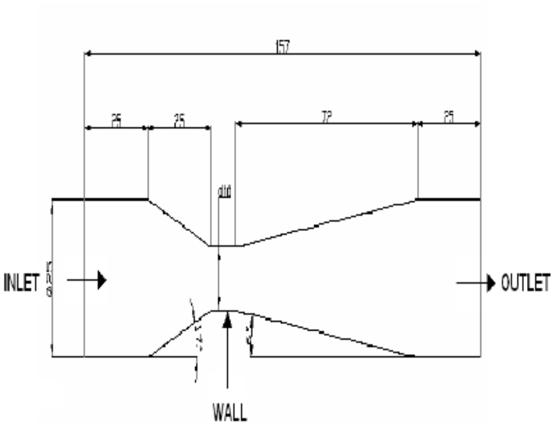
The Vector plots are used to indicate the direction of the fluid flow. The disadvantage of the contour plots is it does not show the fluid direction which can be determined using this vector plot.

XIII. XY PLOTS

xy plot are nothing but two-dimensional graphs. They represent the variation of one dependent variable versus another independent variable. xy plots are the simplest and the most straight forward category of computer graphical representation of CFD results. Although such graphs are not particularly sophisticated, they still remain the most precise quantitative way to present numerical data on a graph; that is, another person can readily read quantitative data from curves on a xy plot without making any mental or arithmetic interpolation.

XIV. PROBLEM DEFINITION

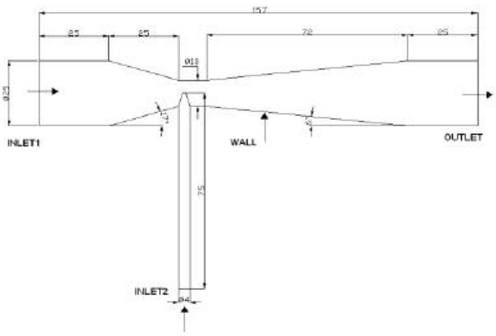
VENTURI



A venturi with the above specified dimension is used for determining the pressure distribution along the throat section. The boundary condition for the venturi is given in table below,

NAME	BOUNDARY CONDITIONS
Inlet	Pressure Inlet
Outlet	Pressure Outlet
Wall	Adiabatic Wall

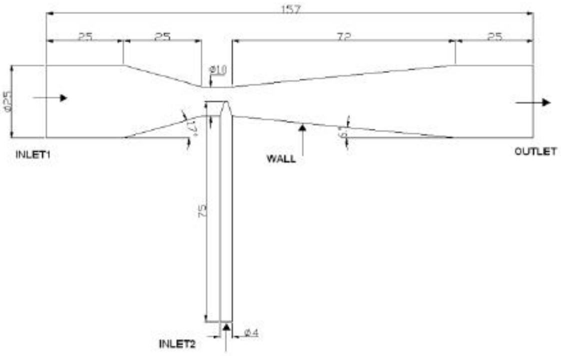
MODIFIED VENTURI CASE1



A modified venturi with the above specified dimension for which the nozzle being positioned at the beginning of throat section. The boundary condition for the modified venturi is given in table below,

NAME	BOUNDARY CONDITIONS
Inlet1	Pressure Inlet1
Inlet2	Pressure Intlet2
Outlet	Pressure Outlet
Wall	Adiabatic Wall

MODIFIED VENTURI CASE2



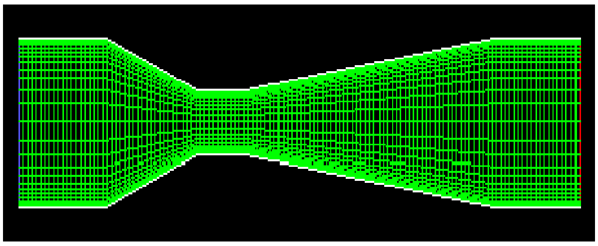
A modified venturi with the above specified dimension for which the nozzle being positioned at the end of throat section. The boundary condition for the modified venturi is given in table below,

NAME	BOUNDARY CONDITIONS
Inlet1	Pressure Inlet1
Inlet2	Pressure Intlet2
Outlet	Pressure Outlet
Wall	Adiabatic Wall

XIV.ANALYSIS

Analysis is done for the venturi flow configuration in order to determine the low pressure and the nozzle is being positioned at that predetermined location. The flow analysis is then extended to the modified venturi for the two different position of the nozzle at the throat section. The above two cases are being done and from the result obtained are then compared for the effective mixing. By selecting that case in which the mixing is efficient various parameters such as decreasing the throat diameter, increasing the throat length and increasing the Reynolds number of the flow through venturi are shown in the following figures and its effects are being discussed in the next chapter.

XV. MESHING GEOMETRY



- The edges of the models are being meshed using mesh edge icon under the mesh command icon.
- The faces of the models are being meshed using mesh face icon under the mesh command icon.

XVI. BOUNDARY CONDITIONS

The following boundary conditions are specified using specify boundary type option as given below,

NAME	BOUNDARY CONDITIONS
Inlet	Pressure Inlet
Outlet	Pressure Outlet
Wall	Adiabatic Wall

- The different zones that have to be created are then specified using the zone command icon and the boundary types are being specified.
- The left most vertical edge is selected; the name and boundary condition are given as Inlet and Pressure Inlet respectively.
- The right most vertical edge is selected; the name and boundary condition are given as Outlet and Pressure Outlet respectively.
- The remaining edges are selected; its name and boundary condition are given as Wall and Adiabatic Wall respectively.
- Now the file is saved using the same file name.
- The mesh file is then exported to be used with Fluent software.

XVII. FLUENT

Fluent uses the Finite-volume method to solve the governing equations for a fluid. It provides the capability for Computational Fluid Dynamics (CFD) to simulate fluid flow problems. In this problem, Fluent is used for solving and post-processing.

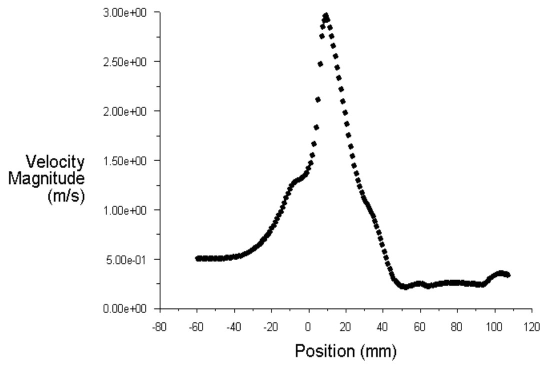
XVIII. RESULTS AND DISCUSSION

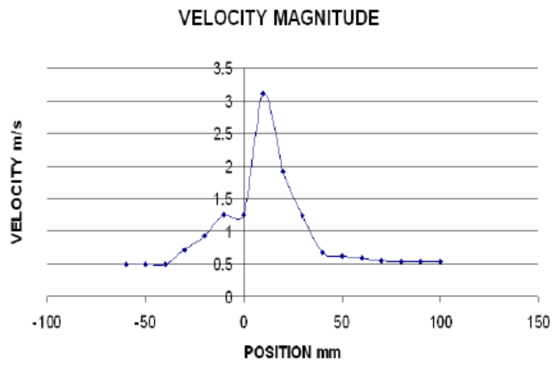
The computations are done for two cases of modified venturi. The position of the nozzle in the modified venturi is found by carrying out the flow analysis through venturi and the low pressure position is determined from the analysis. In the first case the nozzle was positioned at the beginning of the throat section, for the second case the nozzle was positioned at the end of the throat section and analysis was carried out for both the cases. The computations are extended for the case in which the effective mixing is obtained, by varying parameters such as increasing the Reynolds number (1560, 1785), decreasing the throat diameter (10mm, 8mm), increasing the throat length(10mm, 20mm). Numerical calculations are performed for certain representative cases. The contour and vector diagrams of the temperature, pressure and velocity distribution are reported and a detailed investigation was made on them. The effect on changing varies parameters are then inferred from the results and discussed.

A systematic grid refinement study is conducted to obtain grid independent solutions. This grid resolution is therefore used for all subsequent computations. The grids are more clustered towards the nozzle tip in all the cases being discussed. The observed information's about the grid are given below,

CASES	GRID SIZE	NODES
Venturi	28 x 162 (4536)	4347
Modified Venturi (case 1)	162 x 98 (15876)	4697
Modified Venturi (case 2)	158 x 96 (15168)	4675
Modified Venturi (throat diameter decreased)	161 x 98 (15778)	4515
Modified Venturi (throat length increased)	170 x 97 (16490)	4744

VELOCITY DISTRIBUTION AT THE DIFFERENT POSITION IN X-AXIS 2mm ABOVE THE NOZZLE TIP





XIX.CONCLUSION

Laminar flow through modified venturi for mixing has been studied in detailed through two-dimensional numerical simulation by changing various parameters such as increase in Reynolds number, decrease in throat diameter and increase in throat length. For both the nozzle position in the throat section of the venturi the flow configurations have been studied. Predicated flow has been presented in the form of the contours and vectors. The variations of the velocity are obtained theoretically and using the software is compared for certain representative cases.

There are noticeable changes in the flow for the positioning nozzle at different location at the throat section. Nozzle positioned at the end of the throat section yields better mixing when compared with the nozzle positioned at the beginning of the throat section. The mixing of the hot liquid and the cold liquid obtained indicates that mixing of different liquids can also be obtained using this modified venturi concept.

The increase in Reynolds number results in increase in the vacuum size and the slightly increase in the velocity magnitude. By decreasing the diameter the results inferred are better mixing and the lowest pressure of all cases considered is obtained. And for increasing the length of the throat section reveals to large vortex.

XX. REFERENCE

[1] C. Herschel, "The Venturi Meter", Builders' Iron Foundry, 1898.

[2] ISO 5167:2003 Parts 1 and 4 – Measurement of Fluid Flow by means of Pressure Differential Devices inserted in circular cross-section conduits running full, International Standards Organisation, 2003.

[3] BS 1042:1981 Measurement of fluid flow in closed conduits – Pressure Differential Devices, British Standards, 1981.

[4]R.W. Lockhart, and R.C. Martinelli, "Proposed correlation of data for isothermal two-phase, two-component flow in pipes", Chemical Engineering Progress, vol. 45(1), pp. 39-48, 1949.

[5] NFOGM Handbook of Multiphase Flow Metering Revision 2, Norwegian Society for Oil and Gas Measurement, March 2005.

[6] N. Barton, M. MacLeod, K. Zanker, and G. Stobie, "Erosion in Subsea Multiphase Flow Meters". The America's Workshop 2011.

[7] A.W. Jamieson, "Wet Gas Metering Key Note – Status and Trends on Technology and Applications", 19th North Sea Flow Measurement Workshop, 2001.

[8] G. Stobie, "Wet Gas Metering (and how not to do it)", Wet Gas Metering Seminar at the National Engineering Laboratory, East Kilbride, October 1996.

[9] ASME MFC-19G-2008 Technical Report – Wet Gas Flow metering Guideline, American Society for Mechanical Engineers, 2008.

[10] J.W. Murdock, "Two-Phase Measurement with Orifices", Journal of Basic Engineering, vol. 84, pp. 419-433, 1962.

[11] BS ISO 5168:2005 Measurement of fluid flow – Procedures for the evaluation of uncertainties, British Standards and International Standards Organisation, 2005.

[12] A.W. Jamieson, and P.F. Dickinson, "High Accuracy Wet Gas Metering", 11th North Sea Flow Measurement Workshop, 1993.

[13] Rick de Leeuw, "Liquid Correction of Venturi Meter Readings in Wet Gas Flow", 15th North Sea Flow Measurement Workshop, Paper 21, 1997.

[14] D. Chisholm, "A Theoretical Basis for the Lockhart-Martinelli Correlation for Two-Phase Flow", International Journal of Heat Mass Transfer, vol. 10, pp. 1767-1778, 1967.

[15] D. Chisholm, "Flow of Incompressible Two-Phase Mixtures through Sharp-Edged Orifices", Journal of Mechanical Engineering Science, vol. 9(1), 1967.