International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 02 Issue: 09 | Dec-2015 www.irjet.net p-ISSN: 2395-0072

CFD analysis of a single phase mixing of fluids without the aid of stirrers

Md Irfan[1], Mr. Kailash B A [2], Dr. Gowreesh Subramanya [3]

[1] P.G Student, Dept. of Thermal Power Engineering, VTU PG Centre Kalaburagi, Karnataka, India
[2] Asst. Professor, Dept. of Thermal Power Engineering, VTU PG Centre Kalaburagi, Karnataka, India
[3] Associate Professor, JSSATE Bangalore, Karnataka, India

***

Abstract - ***Numerical analysis of mixing processes is becoming increasingly important for understanding and optimizing industrial mixing problems. Due to the increase in cost and time for experimentation, a numerical simulation, CFD has been used which have been proven to be useful tool for analyzing industrial mixing process. In the present work an attempt is made to investigate the mixing of miscible and immiscible fluids in static mixer without any inserts, by considering low cost design. In the present design the potential head of the fluids itself is used for powering the fluid into the mixing chamber. Fluid enters the system through two inlets (inlet1 & inlet2) and leaves the system from a single outlet with a varied inlet shapes and mixing chamber. Study is carried out using commercial Computational Fluid Dynamic software for 6 designs. An inlet velocity at the two inlets i.e. velocity at inlet1 is 2m/s and that of the inlet2 is 3m/s. These velocities are varied and the mixing performance is studied. The simulation was operated out to check and investigate the various flow parameters such as velocity contour, pressure contour, turbulent kinetic energy, residence time and streamlines at various planes of the designed models. After the simulation it is observed that for model 3, the outlet velocity is 6.7128 m/s and the residence time is 20.21sec which is higher compared to the other models.***

Key Words: *Single Phase Mixing, CFD, Stirred tank, Fluid mechanics and Static mixers.*

1. INTRODUCTION

The mixing process in the industries is a single and unique process. The aim of this is to convert the heterogeneous quantity to homogeneous. In industrial process engineering mixing of liquids is a regular and frequent process. Therefore the tools used for mixing is dependent on the characteristic property of the liquids that are mixed.

Hence in order to master the mixing processes n the industries, it is very important to select a right choice of mixer i.e. equipment for mixing. The one-phased mixing comprises of low shear and high stream flow mixers to bring about engulfment of fluids, where as in multi-phase blending basically involves high-shear and low-stream flow blenders to develop tiny drops of fluids. Hence these droplets may generally occur in whether a turbulent regime or laminar or even in transitional flow, which is dependent on the Velocity & Reynolds No. (Re) of stream flow. Turbines and impellers types are frequently used for turbulent or transitional mixing, while helical ribbons, anchors and static mixers are used for laminar mixing.

1.1 Classical Methods for Fluid Mixing

Mixing of two miscible fluids is entirely simple and happens by dissemination. Such sort of blending does not make any issue. Basic shaking or blending is sufficient if in the event that the fluids are not promptly miscible or in the event that they have all together different viscosities then electric stirrer may be utilized.

Mixing in industrial is achieved in batches i.e. dynamic mixing, sometimes inline. The affective mixers require electric motors which works at accepted speeds of 1800 or 1500 RPM, and it is more rapid compared to normal stirring speed. Here mechanism of gears and gear box setup should be incorporated to lower the speed and to increase the torque. Mixing also requires multi-shaft mixers to blend the product completely.

A static mixer is precision engineering device that is used for the purpose of continuous mixing of fluid materials. A static mixer is basically a liquid mixing device but it can also be used to mix streams of gases, to disperse gases in liquid and also to mix immiscible liquids. This type of devices uses energy that is achieved through loss of pressure when the fluid flows through the static mixer. The common type of static mixing device consist of mixer element (i.e. inserts) contained in cylindrical tube or square housing. The size of the mixer can vary from 6 mm to 6 meters diameter. The basic construction materials used for the static mixer components are as follows stainless steel, Teflon, poly vinyl chloride, PVDF, polyacetal, CPVC and polypropylene.

1.2 Literature Review

Darryel Bouncer [1] used the DEM – Discrete element method for the simulation of Rushton impeller mixer. Where water was mixed in a baffled container/vessel. Here Darryel Bouncher used a steady state flow and the fluid drag forces are assumed. Darryel Boucher also studied the fluid particle behavior using an extra method called PEPT, which is abbreviated as positron emission particle tracking. He later concluded that behavior of water in the stirred container depends upon the velocities and trajectory of fluids.

Valentina Dore [3] he experimented on the circulation of flow patterns and mixing characteristics of a single phase mixing within water channels. The method he used is PIV – particle image velocimetry. He also used the velocity profile details and the compartmental model for understanding the mixing characteristics. Also he used to note down the multipoint details of velocity during PIV method.

2. Problem Definition

The First problems that were coming in previous works were length scale of mixing (i.e. the amount of two different fluids remains in contact with each other while the mixing process). Hence the interface time plays a vital role in liquid mixing. The contact time (scale length factor) must be well defined while experiments. Greater the contact time of two different fluids, the greater is the mixing ratio.

The second problem that we came across while literature review is region change. Two fluids have two different regions before they mix. Hence the amount of regime change is to be considered.

Also the third problem is to be considered that is impact of flow. The greater and higher is the impact, and then higher will be the turbulence. Therefore higher turbulence will tear the fluid components/particles making them mix correctly.

3. Specification of working Models

The following six models were analyzed with different shapes and dimension. The basic dimensions of the cylindrical chamber are 4m body diameter. Length and diameter of inlet and outlet are 1.5m and 1m respectively. Where as the square chamber has 4m area with same 1m inlet and outlets. Also the nozzle used in two models have nozzle angle of 18degree.

1. Cylindrical container with straight inlets – Model 1.

2. Cylindrical container with tangential inlets – Model 2.

3. Cylindrical container with tangential nozzle inlets – Model 3.

4. Square container with straight inlets – Model 4.

5. Square container with tangential inlets – Model 5.

6. Square container with tangential nozzle inlets – Model 6

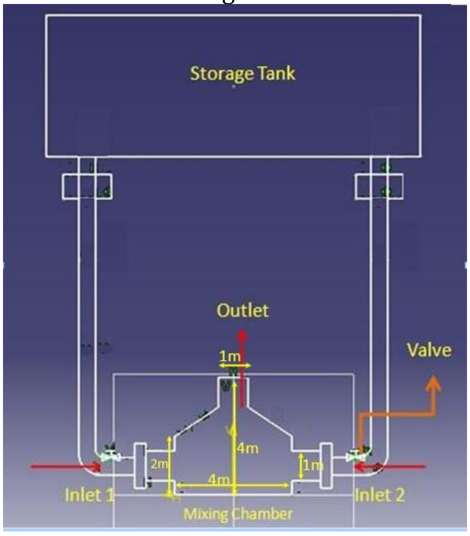

Fig -1: Basic working model

3.1 Methodology and CFD Simulation

The container geometry was created in the CATIA solid works and CFX was used for simulation of the fluid mixing inside the vessel. A CFD domain approach was incorporated with the grids done on the entire body with the tetrahedron meshing. The components had two domain/parts surfaces. The first was set of two inlets and the other was a single outlet. The surfaces of these two domains were set as fixed walls as the boundary conditions. Also boundary conditions used for the whole body were also mentioned as wall. The outlet pressure was kept constant. Inlet velocities were 2 m/s and 3 m/s respectively. Further with these parameters all the models were analyzed for 1000 iteration in the fluent solver. After completing the all the iteration and at the convergence point the models were saved. These models were then analyzed in the CFX post processor which gave the necessary information on all models such as Velocities, Pressure, Turbulent Kinetic Energy, Viscosity, and Residence time.

Table -1: properties of fluid materials

Properties	Values
Working fluid	Water liquid
Density (Kg/m^3)	998.2
Viscosity (Pa-s)	0.00103
Specific heat (J/kg-k)	400
Thermal conductivity	0.6

Table -2: Boundary conditions

Entity	Zone	Zone type	Value
Inlet velocity	Boundary	Inlet	2m/s
Outlet velocity	Boundary	Outlet	3m/s
Outlet pressure	Boundary	Wall reduction	zero

4. Results and Discussions

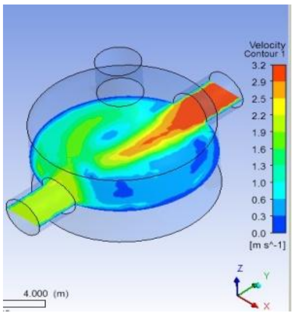

Fig -2:Velocity contour for model-1

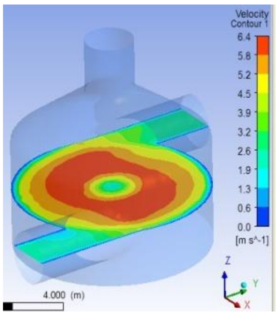

Fig -3:Velocity contour for model-2

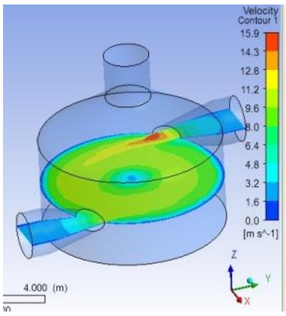

Fig -4:Velocity contour for model-3

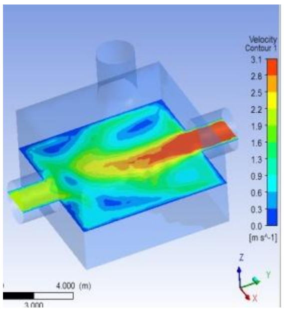

Fig -5:Velocity contour for model-4

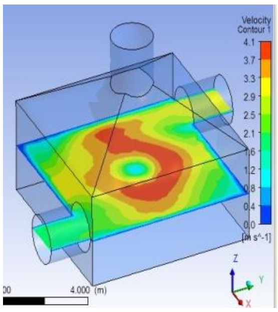

Fig -6:Velocity contour for model-5

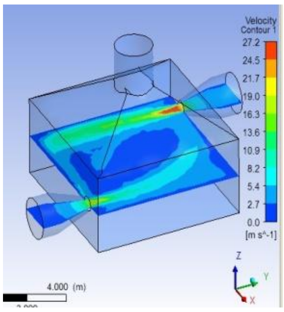

Fig -7:Velocity contour for model-6

We can see in the above six models the third model has high average outlet velocity compared to other models. Also cyclonic pattern of the streamlines is observed for third model (i.e. fig: 4), which helps in creating turbulence. For liquid-liquid flow in small channel, the liquid flow filed moves down stream due to gravity but after obtaining the max momentum. It later moves upstream at constant speed. The fluid inside circulates forming counter rotating vortices with closed streamlines. Due to nozzle inlets in third model the velocity obtained is high and this high velocity later develops turbulence. The maximum is the turbulence higher will be the fluid breaking. Whereas the square chambers doesn’t provide good circulating motion. At the edges of square chambers vortices are formed. Considering the vector plots we can say directly that the liquid water particles were making the loops under the 1st plane. The trajectory also has some irregularities due to high value of Reynolds number.

Table -3: Results of various parameters

Parameters	Inlet 1 Pressure (Pa)	Inlet 2 Pressure (Pa)	Inlet 1 Velocity (m/s)	Inlet 2 Velocity (m/s)	Outlet Velocity (m/s)
Model 1	194795	182516	2	3	5.0066
Model 2	187927	205461	2	3	6.4194
Model 3	217593	270765	2	3	6.7128
Model 4	203582	196654	2	3	5.0104
Model 5	434951	429622	2	3	6.1661
Model 6	186804	398390	2	3	5.0422

5. CONCLUSION

Flow parameters such as Residence time, Velocity contours, Pressure contours, Turbulent Kinetic Energy, Stream lines, Velocity vectors are plotted and analyzed for all the six designs. Based on the analysis following conclusions are drawn and it is found that Model 3 is the best.

- Residence time of the fluids inside the mixing chamber found to be higher due to the cyclonic nature of design.
- Nozzle shaped design at the inlet pipes allows in maximizing the fluid inlet velocity.
- Due to tangential inlets, fluid tends to take up centrifugal motion inside the mixing chamber and which in turn leads to mixing in a better way.
- Turbulence found to be higher due to the concentric flow pattern of both the fluids.
- Pressure drop observed to be nominal.

REFERENCES

[1] Darryel Bouncher, NimaGharib, Zouthong Deng, Thomas Leadbeater, Raymond *"Pept validation of a CFD-DEM model of a fine quartz particle behavior in stirred water*, procedia Engineering 102(2015) 1305-1315 Elsevier.

[2] Debangshu Guha, M.P. Dudukovic and P.A. Ramachandran 2007, *"CFD compartmental modeling of single phased stirred tank mixer"*Chemical Reaction Engineering Laboratory, Washington University in St.Louis,2007 MO:63130.

[3] Valentine Dore, DimitriousTsaolidis, PanagiotaAngeli, chemical Engineering science 80(2012) 334-341 *"mixing patterns in water plugs during water liquid segmented flow in microchannels".*

[4] Weixiong Chen, Chaoyin Shi, Mengqi Hu, Daotong Chong, Jinshi Wang, Junjie Yan *" Numerical and experimental analysis of two phase flow in ejector"*energyprocedia science direct 61(2014)-1298-1301-ICAE2014.

[5] Al ansary HAM, Jeter SM *Numerical and experimental analysis of single and two phase flow in ejectors,* HVAC and research 2004, 10:521-538.

[6] Dr. Bhawana Bhatt & S.S Agarwal *"Revised mixing"* NasdlNiscair Research 751/1/2007-Delhi Institute of Pharmaceutical Science and Research.

[7] ZdzislawJaworskia, HalinaMurasiewicz *"Numerical and experimental studies of liquid-liquid mixing",* Pomeranian University of Technology, Szczecin, AlejaPiastow 42, 71065 Szczecin, Poland.

[8] Leng D.E. Calabrese R.V., 2004. *"Immiscible liquid-liquid systems",* in: Handbook of Industrial.

[9] E.L. Paul, V.A. Atiemo-Obeng, S.M. Kresta, edsMixing. Science and Practice, , chapter 12, JohnWiley & Sons Inc., Hoboken, New Jersey, pp. 689-753.

[10] Song H. Han S.2005. *"General correlation for pressure drops in a Kenics static mixer".* Chem.EnggSci. 60, 5696-5704.Szalai E., Muzzio F., 2003."Fundamental approach to the design and optimization of staticmixers".AIChE J., 49, 2687–2699.