

CFD investigation on fluid flow analysis in fluid separator

Pranita Shinde¹, Pramod Kothmire², Yogesh Bhalerao³

School of Mechanical and Civil Engineering, MIT Academy of Engineering, Alandi, Pune

1. pranita.shinde1994@gmail.com

2. ppkothmire@mech.maepune.ac.in

3. ybhalerao@maepune.ac.in

Abstract-The analysis of fully developed flow in the two fluid separators is an important issue in the industry. Such as production, processing, and petrochemical. The role of the two fluid separator is to separate two different fluid by using an appropriate mechanism without changing quality. In this study, we have reviewed the different mechanism of two fluid separations such as gravity sedimentation, centrifugation, and electro kinetics, etc. The current work is focused on the design aspect of a fluid separator with respect to geometry and thermal design. CFD is used to simulate flow in a fluid separator and its results are verified experimentally. Flow rates used in the simulation have different values in interval 0.1 LPM. The study shows the best performance of fluid separator with respect to shape and flow rates. The given work helps to the correlate various design of separator in the industry with laboratory separators.

Keywords- CFD, Flow distributor, Flow rates, velocity, fluid separator.

I. INTRODUCTION

A separation process of oil and water is very well known to industry. There are many types of oil water separator. Design and selection of oil and water separator is considered after studying oil separation performance parameters and life cycle cost. The ratio of oil and water is one of the important parameters while separation. The produced oil from reservoir associated with salinity water, gas, sediments and contaminants [1]. Due to the presence of surfactants in oil such as dust, dirt, iron nails, and fine solids, this emulsion is quite stable [2]. Segregation of oil contaminants is necessary to achieve industry specification. Otherwise, the presence of salt causes corrosion, deactivation of catalysts, precipitation, etc. in separator. To date, there exist several techniques for enhancing the water-oil emulsion separation such as pH adjustment gravity or centrifugal settling [3], Heat treatment, and electrostatic demulsification [4, 5] which is done in plants. From the viewpoints of energy efficiency, electrostatic demulsification is used to demulsify emulsion and separate the water [6]. For resolving the separation of the oil-water emulsion, available treatment options facilities include mechanical, thermal, electrostatic and chemical or more often a combination of these[7].

In oil industries general demulsification procedure is found. To resolve water-in-oil emulsion into bulk phases of oil and water can be viewed as three stage process involving, (i) destabilization, (ii) coalescence and (iii) gravity separation. In short, destabilization is accomplished by adding heat or selected interfacial active chemical compound, due to that density difference between water and oil is increased. Viscosity of oil is also decreased by heating. Then weakening of the stabilization effect of natural emulsifiers started. And that form a film surrounding to the dispersed water droplets. The coalescence stage occurs when the breaking of the interfacial film is done which is, surrounded by water droplets. By application of high voltage electric field polarization of water, droplets are taken. Induce electric force between them to accelerate their motion towards each other. And allowing contact between droplets result in coalescence. As per stokes law, the formation of larger droplets can easily separate in this way [8]. Gravity separation is oldest type. Density difference between two immiscible liquid play a lead role. In this, the coalescence droplets of water to separate from oil require sufficient residence time and favorable flow pattern [9]. In order to improve understanding of mechanism numerous experimental and theoretical studies have studied. That may help to

influence of destabilization of the oil emulsion. “Bottle test” is experiments based traditional laboratory techniques [10], which consist of inspection, the separation amount of water from quiescent emulsion sample with time. This test is static in nature and does not correlate the flow of actual model of oil emulsion [11].

This paper mainly focused on the design aspect of separator using CFD with single-phase fluid, the above review shows that there is no unique design of two fluid separators. The current work focused on optimum shape and flow rate of the emulsion made up of two fluids with the help of CFD simulation we have investigated the optimum flow rates which should be kept in a fluid separator to get sufficient residence time for two fluid separations [12]. Also, we have co-relate our design of fluid separator with respect to a laboratory and industrial fluid separator. Though, caution has to be taken on the efficiency of separation of two fluids.

II) METHODOLOGY

2.1 Geometrical modelling

A geometrical model of a distributor design or problem statement is as shown in fig. 1. The main vessel (separator) having 3 liters capacity. On which batch wise experiments have done to find water-oil separation time. Here we used a 90:10 ratio for oil and water. Required separation time for this process is 6 minutes. To determine fluid separator parameters at first, we have decided to study distributor design (fig. 2) which helps to overcome the arising problems like a meshing of a complicated geometry, number of iterations in FLUENT.

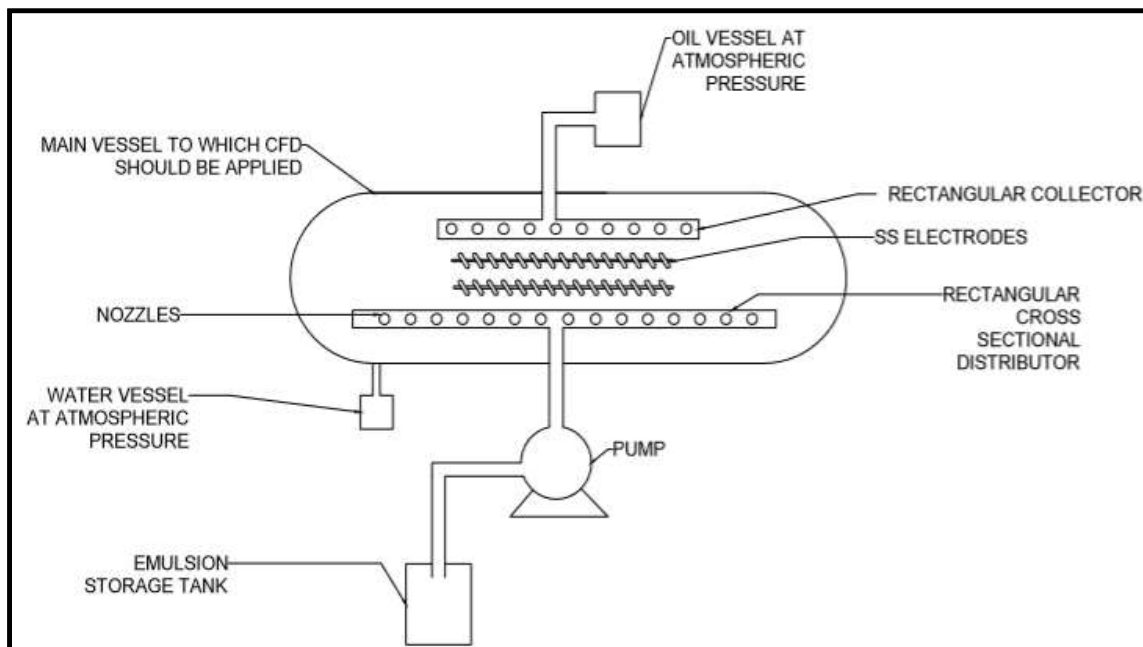


Fig.1 Schematic representation of fluid separator (distributor is at bottom)

To get the optimum result, straight (fig.2) and taper (fig.3) shape type distributors are need to study. Refer table 1 for distributor’s dimension. For both type distributor, segregated hole design (fig. 4) is used to reduce velocity variations.

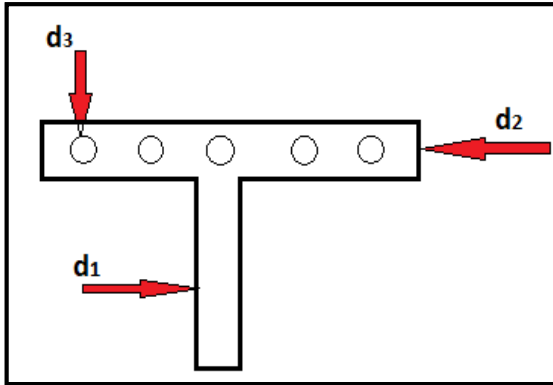


Fig. 2 Straight shape distributor

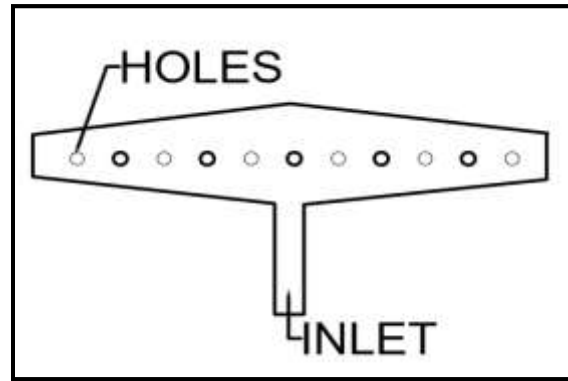


Fig.3 Taper shape distributor

Table 1. Straight and taper shape distributor dimension

Parameters	No.of holes (outlets)	Hole diameters (mm)	Vertical pipe (inlet) dimension (mm)	Horizontal distributor dimension (mm)
Straight	11	2	Length (l) =100 Diameter (d ₁) =10	Length (l) =140 Diameter (d ₂) =20
Taper				Length (l) =140 Diameter (d ₂) =20 and 10

Horizontal pipe of the distributor contains 11 number of segregated holes. Where, on one side 6 holes (1,3,5,7,9 and 11) are located at 10mm distance from the center and on another side, 5 holes (2,4,6,8 and 10) are located at 20 mm distance from the center. As mentioned, straight and taper shape of a distributor provided the same design of segregated holes.

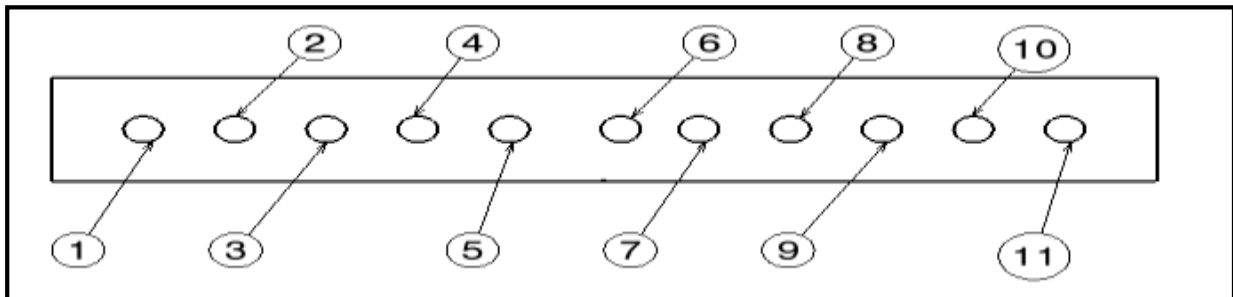


Fig. 4 Distributor with holes in segregated form

2.2 Meshing

For 2D and 3D (fig.5) modelling here we used AutoCAD and CATIAV5 respectively. From fig.2 and 3 we can observe that water enters at inlet and a stream of water split through holes of the distributor. The fine mesh generation was created on an assembly with 19647 nodes and 99335 elements. It is performed by ANSYS workbench as shown in fig.6. Fluent is used as solver. Tetrahedron mesh is used for this geometry. For taper shape (Fig.3) distributor also have similar meshing process.

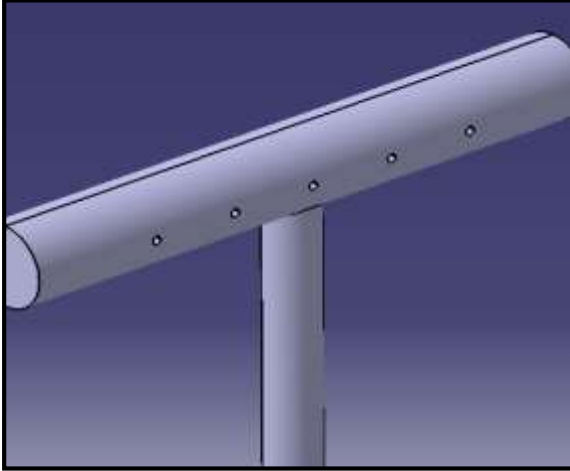


Fig.5 CATIA model of distributor

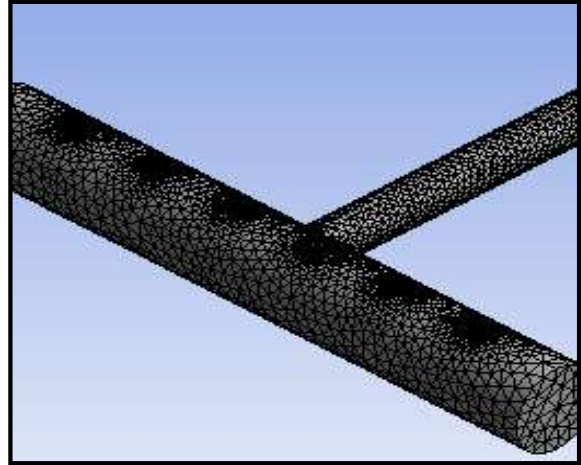


Fig.6 meshing of distributor

Table 2. Physical properties of water (20 °C)

Fluid	Density (kg/m ³)	Viscosity (kg/m-s)
Water	998.2	0.001003

2.3 Boundary conditions

The Boundary condition for inlet pipe (d_1) is velocity and at outlets (11 No. of holes) is gauge pressure, normal to boundary. After satisfaction of this condition, we can find out velocity and pressure value through each holes by changing inlet velocity conditions for different flow rates.

From continuity equation (1), we can find out inlet pipe (d_1) velocity.

$$Q = Av \tag{1}$$

Where, Q is Volumetric flow rate, A is Cross sectional area of inlet pipe and v is flow velocity. Now we have to find flow velocity by using eq. (2)

$$Q = \frac{V}{t} \tag{2}$$

In this, V is volume capacity of separator and t is separator time. From experiment, we get this values which help to find out volumetric flow rate (Q). Now from eq. (2), cross sectional area is calculated by using inlet pipe diameter.

Then, put values in eq. (1), we get flow velocity (v).

Table 3. Parameter for Estimation of values.

Parameters	Values
Volume capacity of separator (V)	3 Liters
Water-oil separation time (t)	6 min
Volumetric flow rate (Q)	0.5 LPM
Diameter of inlet pipe (d_1)	10 mm

From Table 3, we can easily calculate the value for flow velocity at different flow rates for both type of distributors.

III) RESULTS AND DISCUSSION

3.1 Straight and taper distributors with equal hole diameters

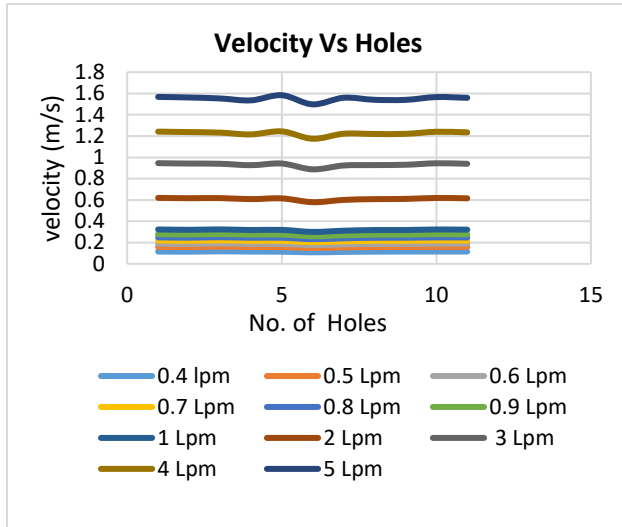


Fig.7a Velocity Vs Holes for straight distributor

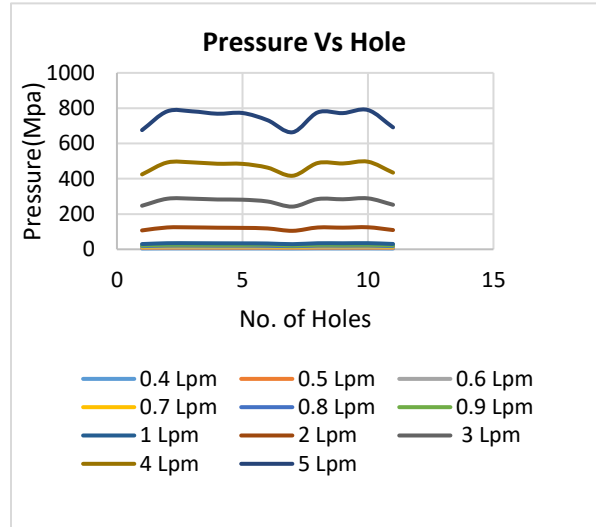


Fig.7b Pressure Vs Holes for straight distributor

Straight shape (Fig.7a and 7b) and taper shape (Fig. 8a and 8b) distributor shows velocity and pressure parameters respectively. (Refer Table 1 for dimension details). As we increased the flow rate, more velocity variation will occur in taper than a straight distributor (Fig 7a and 8a). The effect of fluid viscosity near the surface of the distributor which results to increase friction losses, by default increasing pressure drop also. Hence pressure variation is more in taper than a straight distributor (Fig 7b and 8b).

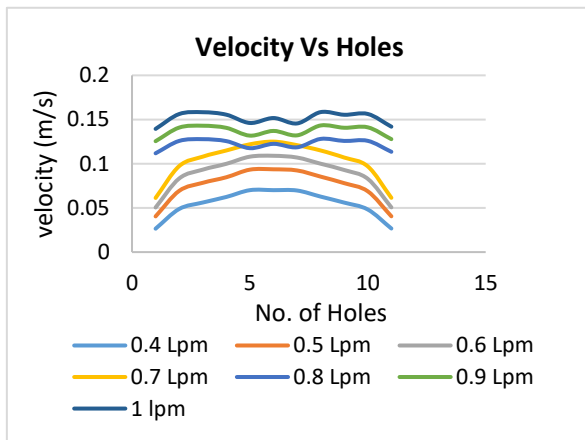


Fig.8a Velocity Vs Holes for taper distributor

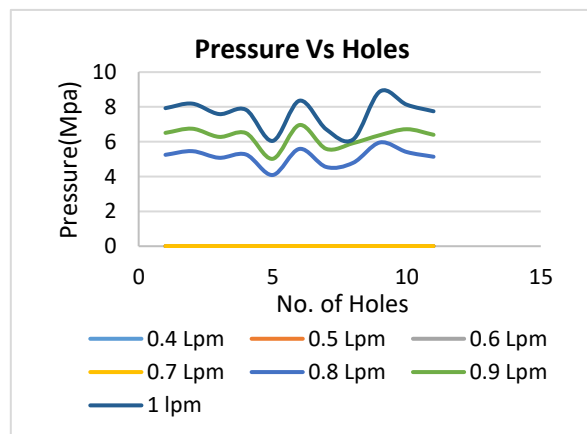


Fig.8b Pressure Vs Holes for taper distributor

3.2 Straight and taper distributors with different hole diameters

In this, we have to vary holes diameter for both distributors (fig 4). Others dimensions are same mentioned in Table 1. Following data in Table 4 will help to understand variations of hole diameters.

Table 4. Variation in hole diameters

Hole No's	Diameters(mm)
1 and 11	3
2, 6 and 10	3.5
3 and 9	4
4 and 8	4.5
5 and 7	5

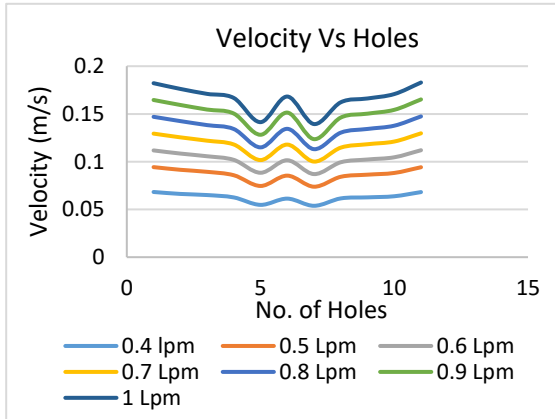


Fig.9a Velocity Vs Holes for straight distributor

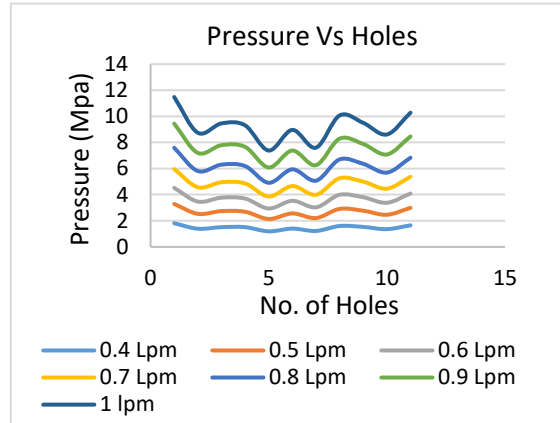


Fig.9b Pressure Vs Holes for straight distributor

Straight shape (Fig.9a and 9b) and taper shape (Fig. 10a and 10b) distributor shows velocity and pressure parameters respectively for variations in hole diameters. From fig 9a and 9b we can observe that at the end, hole diameters is less than holes near a center. It shows peak-valley shape distribution. Therefore increasing diameter of holes is reduced velocity and vice versa. From fig 10a and 10b, we can observe that due to taper shape and less hole diameters at the end the velocity is lower than center. And pressure is exact vice versa. From this figures, we can easily observe that straight shape distributor have minimum variations than taper shape.

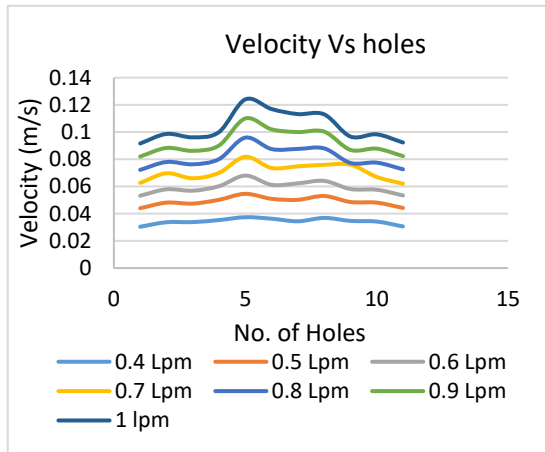


Fig.10a Velocity Vs Holes for Taper distributor

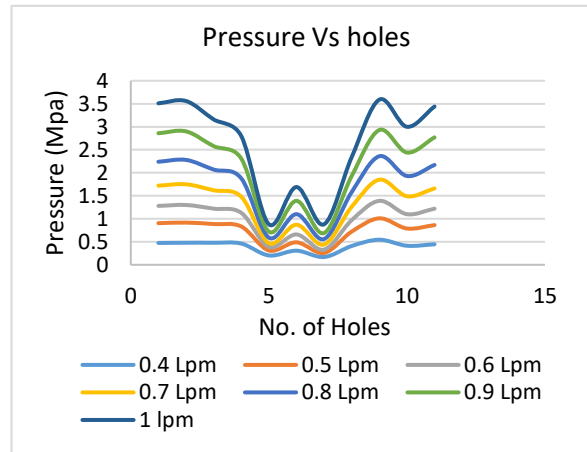


Fig. 10b Pressure Vs Holes for taper distributor

Design of distributor parameters is studied in various ways. Experimentally studied shows that water passes through holes of distributor have less variations than simulation-based study. After studying the impact of distributor design. Further, the process is to find the effect of drag force in main vessels. There are two parameters need to study

in drag force effect one is with a collector and without collector (fig 1). Hole diameters variation has major impact on distributor design. Here, we have used different flow rates values. It shows that, if we increased the flow rate variations will occur more. At point 0.5 LPM we get less variations for water single phase liquid. As of now we discussed effect of hole diameters at different flow rates. Now we studied the effect of change in inlet diameter (d_1) value with equal hole diameters.

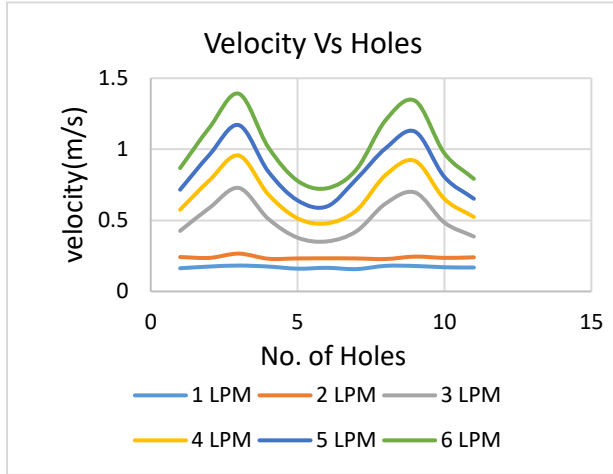


Fig.11a Velocity Vs Holes for Taper distributor

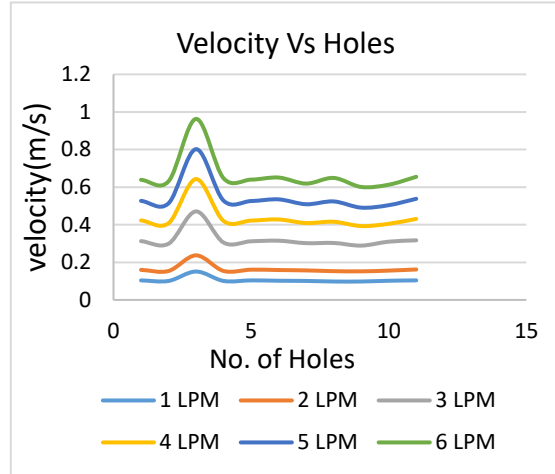


Fig. 11b Velocity Vs Holes for taper distributor

Straight shape (Fig.11a) and taper shape (Fig. 11b) distributor shows velocity parameters for 1 LPM to 6 LPM. In this we take the value of ($d_1=12\text{mm}$) (Ref. figure 2) for both distributors. Changing the inlet diameters shows a peak-valley distribution of flows. This study help to understand straight shape distributor is more useful than taper shape.

IV. CONCLUSION

To reduce velocity variation, simplifications are made in design of distributor with segregated holes. By using straight and taper shape distributor, velocity variations are observed. And design is confined for straight shape distributor. Inlet pipe diameter and holes diameter are major influenced factors affected on design of distributor and flow rate process. Straight shape distributor gives fully developed flow as compared to taper shape distributor. CFD analysis is one of the important tool for designing two fluid separator and it gives the proper design of two fluid separator and flow rates. Also it helps to scale up the lab studies to techno commercial design.

V .REFERENCES

- 1) Arnold, K. E., & Koszola, P. J. (1990, 2). Droplet-Settling vs. Retention-Time Theories for Sizing Oil/Water Separator. SPE Production Engineering, 5, 59-64. doi:10.2118/16640-pa
- 2) Cunha, R. E., Fortuny, M., Dariva, C., & Santos, A. F. (2008). Mathematical modeling of the destabilization of oil emulsions using population balance equation. Industrial & Engineering Chemistry Research, 47, 7094-7103.
- 3) Dezhi, S., Chung, J. S., Xiaodong, D., & Ding, Z. (1999). Demulsification of water-in-oil emulsion by wetting coalescence materials in stirred-and packed-columns. Colloids and Surfaces A: Physicochemical and Engineering Aspects, 150, 69-75.

- 4) Goto, M., Irie, J., Kondo, K., & Nakashio, F. (1989). Electrical demulsification of W/O emulsion by continuous tubular coalescer. *Journal of Chemical Engineering of Japan*, 22, 401-406.
- 5) Inc.), H. V. (1987). *Petroleum Engineering Handbook*. Society of Petroleum Engineers. Retrieved from <https://www.amazon.com/Petroleum-Engineering-Handbook-H-Bradley/dp/1555630103?SubscriptionId=AKIAIOBINVZYXZQZ2U3A&tag=chimbiori05-20&linkCode=xm2&camp=2025&creative=165953&creativeASIN=1555630103>
- 6) Less, S., & Vilagines, R. (2012, 1). The electrocoalescers technology: Advances, strengths and limitations for oil separation. *Journal of Petroleum Science and Engineering*, 81, 57-63. doi:10.1016/j.petrol.2011.12.003
- 7) Meidanshahi, V., Jahanmiri, A., & Rahimpour, M. R. (2012, 1). Modeling and Optimization of Two Stage AC Electrostatic Desalter. *Separation Science and Technology*, 47, 30-42. doi:10.1080/01496395.2011.614316
- 8) Arnold, K. E., & Koszola, P. J. (1990, 2). Droplet-Settling vs. Retention-Time Theories for Sizing Oil/Water Separator. *SPE Production Engineering*, 5, 59-64. doi:10.2118/16640-pa
- 9) Sjöblom, J., Aske, N., Auflem, I. H., Brandal, Ø., Havre, T. E., Sæther, Ø., . . . Kallevik, H. (2003). Our current understanding of water-in-oil emulsions.: Recent characterization techniques and high pressure performance. *Advances in Colloid and Interface Science*, 100, 399-473.
- 10) Kokal, S. L., & others. (2005). Crude oil emulsions: A state-of-the-art review. *SPE Production & facilities*, 20, 5-13.
- 11) Wojtanowicz, A. K., Xu, H., & Bassiouni, Z. (1994, 4). Segregated production method for oil wells with active water coning. *Journal of Petroleum Science and Engineering*, 11, 21-35. doi:10.1016/0920-4105(94)90060-4
- 12) Peronski, L., Bratley, R., Ingham, D. B., Ma, L., & Pourkashanian, M. (2009). Fluid Flow Analysis and Design of a Flow Distributor in a Domestic Gas Boiler Using a Commercial CFD Software