Available online at www.elixirpublishers.com (Elixir International Journal)

Chemical Engineering

Elixir Chem. Engg. 66 (2014) 20721-20725

ISSN: 2229-712X

CFD Simulation on Vane Type Feed Inlet Device in a Column- Vapour Flow Analysed for Two Different Feed Streams

Hunny Bansal[1], Ameen Sayal[1], Parminder Singh[1] and Vipul Gupta[2]

[1]Department of Chemical Engineering, Thapar University, Patiala, 147004, Punjab, India.

[2]Department of Chemical Engineering, NIT Surat, 395007, Gujarat, India.

ARTICLE INFO

Article history:

Received: 23 November 2013;

Received in revised form:

25 November 2013;

Accepted: 15 January 2014;

Keywords

Vane type feed inlet device,
Mixing,
Turbulent flow,
Fluid Mechanics,
Gambit.

ABSTRACT

This paper attempts to design a vane type feed inlet device to study the flow of vapour in vacuum column containing feed inlet device. The flow of inlet feed has been analysed for pure vapour and mixture streams. The primary function of the flash zone is to separate the entrained liquid feed from the vapour. This is achieved with a feed entry device that changes the direction and reduces the feed velocity as well as by providing adequate height between the wash and the flash zones. Liquid flow behaviour, such as velocity distribution, in the feed inlet device is of considerable importance in determining column performance. Using the volume-average method, a computational fluid dynamics (CFD) model was proposed to describe the liquid flow behaviour in a structured vane type feed inlet device. The values of the geometrical parameters and operation Reynolds number of flow regime in the inlet device and column are reported. The model of inlet device in column has been developed and meshed in Gambit and the velocity distribution of the flow has been studied at different sections of the column. The proposed study of flow in a column containing feed inlet device has been done in Fluent.

Introduction

One of the recurring questions about refinery vacuum tower operation is the issue of flash zone entrainment and the operation of the wash section. As refiners push to optimize yields from the vacuum tower, the limits of existing column internals are often encountered. Although the operation of the vacuum tower needs ultimately to be judged as a whole, this article mainly focuses on the flash zone and wash sections of the column, since they are critical for high-yield operations [1].

The flash zone serves to transition the high-velocity, two-phase feed from the transfer line into the vacuum column in a manner that separates the liquid and routes it to the bottom of the column, while delivering the vapour uniformly to the upper sections of the column. The feed nozzle's orientation into the column can be radial or tangential. A tangential vane type feed inlet device having, relatively simple geometry, no moving mechanical parts, was invented to introduce gas/liquid mixtures into a vessel or column. The purpose of a vane type feed inlet device is to decrease the momentum of the feed, perform a first stage separation of solids and liquid from the vapour, and achieve an even vapour distribution across the vessel cross section. This is obtained by splitting the feed mixture into a series of flat jets.

Computational fluid dynamics (CFD) has become a widely accepted design tool, and its capability to solve process engineering problems is now being realized [2, 3]. While major challenges within this field, like the simulation of discrete particles, are still awaiting better understanding and models of general applicability, many problems of lower complexity can now be addressed by CFD. Among them the vapour feed flow and vapour-liquid feed flow into columns is a good example. CFD offers the opportunity to complete valuable experimental and empirical research, [4-7], by more rigorous but less expensive numerical experiments. First numerical investigations related to gas inlets date back to the early nineties [8] and this subject has also been addressed a few times [2, 9].

This paper presents CFD simulations which have been performed to assess the quality of vapour velocity profiles at various sections of the column. The vane type feed inlet device has been modelled and simulated with two types of feed - pure vapour feed and vapour-liquid mixture feed. The vapour velocity profiles at various sections of the column have been observed for both the different feed flow. The aim of the study was to optimise the separation efficiency without compromising the hydraulic capacity, particularly the pressure drop through the feed nozzle and the inlet device itself. The idea was to design a feature that would collect the separated liquid in a way to counterbalance the upward momentum of the ascending vapour.

Modelling of inlet device

The main aim of the CFD simulation is to analyse the uniformity of the vapour velocity profile right at the feed inlet device and various sections of the column. Therefore the study will mainly concentrate on the region of feed inlet device and column outlets. The packing has to be included in the model because it influences the flow below it. It is modelled as a vacuum column containing the feed inlet device located below the center of the column.

The influence of the liquid on the vapour velocity distribution in the open space between the inlet and the outlet is small and is therefore negligible for the pure vapour feed but is considerable for mixture feed. The vapour flow is assumed to be incompressible. Temperature variations are neglected. The effect of turbulence is taken into account using the standard k-e turbulence model. The boundaries and boundary conditions in the model used are described in Table. 1 and illustrated in Figure. 1.

The inlet and column geometries are symmetrical in many cases. This symmetry could be exploited in reducing the size of the numerical model by one half. It has however been observed

Tele: 09725699140
E-mail addresses: ameen.sayal@gmail.com

that flow fields with and without a symmetry condition do not always agree. While the geometry is symmetrical, there is no guarantee that the flow field keeps this symmetry, especially in high Reynolds number flows. Steady asymmetric mean flows or oscillating flows may evolve. This seems to happen mainly when the inlet nozzle diameter is large and the jet momentum of the entering flow is relatively low. Comparisons between CFD simulations with and without symmetry condition applied and LDA (Laser Doppler Anemometry) measurements in an air/water test rig (1m diameter) showed better agreement for the CFD results without the symmetry condition. To avoid oscillations in the results and tedious data collection over a long simulation time, a minor deviation from symmetry is introduced into the CFD model by shifting the nozzle off the symmetry plane by less than one percent of the column diameter.

Table. 1: Boundaries and boundary conditions. The colours refer to the surfaces in Fig. 1.

Boundary	Position	Boundary Condition (BC)
Feed inlet(green)	Cross section through nozzle, some diameters upstream	Uniform velocity profile, typical turbulence intensity and length scale
Vapour outlet(yellow)	Cross section through the column, some space above the packing bed	Free outlet
Sump (blue)	Liquid surface considered flat	Symmetry, no shear
Walls (white)	Column wall, nozzle wall	Adiabatic for mass and energy, log-law for turbulence

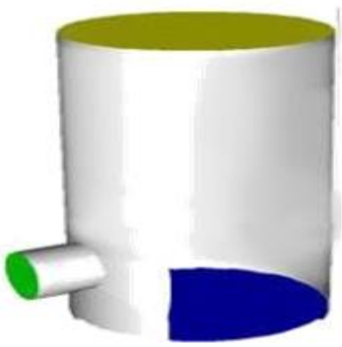

Figure. 1: Computational domain and boundary conditions

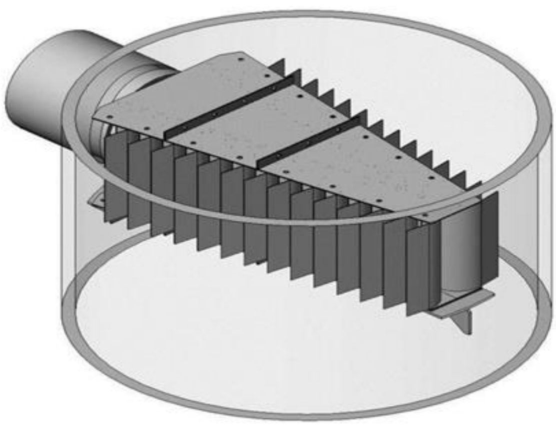

Figure. 2. Vane type feed inlet device shown inside a column. [10]

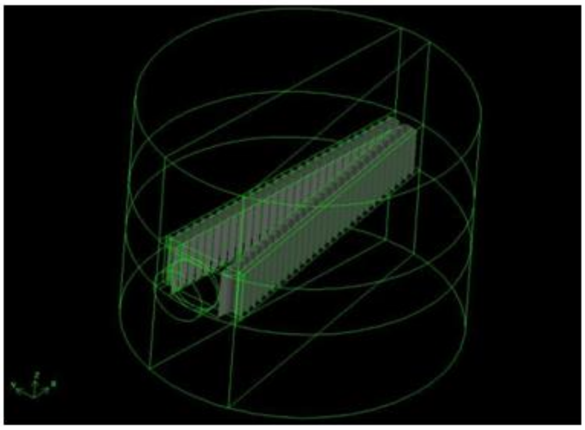

Figure 3. Unmeshed model of vane type feed inlet device in the column

Typical computational grids used consist of some hundred thousand up to 1.6 million finite volume cells, depending on the resolution required to capture geometrical details of the feed system. Simple geometries can be nicely modelled by structured multi-block grids. Grids are more difficult to generate for complex inlet devices. They can be handled by automatic grid generators using unstructured tetra grids, structured cooper grid and structured hexagonal grids at different sections of the inlet device and the column. Commercial CFD software' based on the finite volume approach was used for the simulations. Vapour feed systems studied is shown in Figure. 2. The measurement parameters of vane type feed inlet device developed in Gambit are shown in Table, 2. [11] The model developed of vane type feed inlet device has been shown. Figure 3 shows unmeshed model of vane type feed inlet device in the column, and the figure 4 shows a meshed model of vane type feed inlet device in the unmeshed column. Figure 5 shows a complete meshed (1.6 million grid elements) column containing inlet device.

Table 2: Measurement parameters of vane type feed inlet device

Edge angle	5 degree
Vane angle	7.8 degree
Length of inlet device	10800 cm
Diameter of column	12000 cm
Diameter of inlet	212 cm
Width of vane entrance	40 cm
No. of vanes per side	28
Vane radius	100 cm

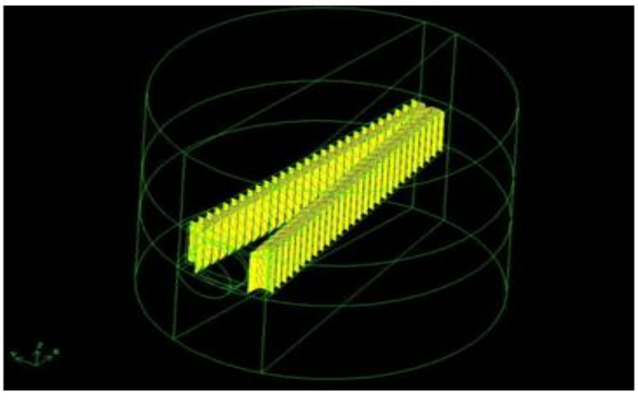

Figure 4. Meshed model of vane type feed inlet device in the unmeshed column

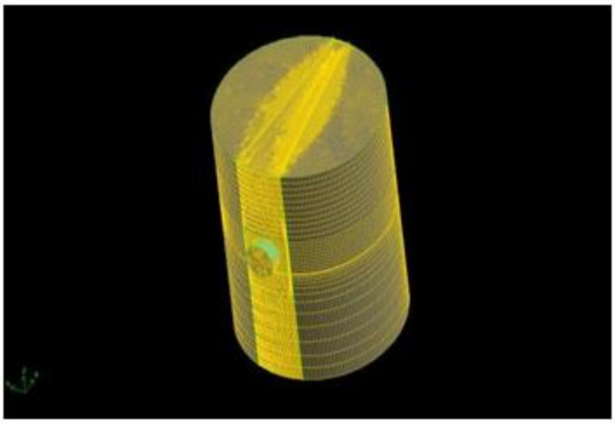

Figure 5. Complete meshed (1.6 million grid elements) column containing inlet device

Simulation of inlet device

After the model is developed, the nature of the feed and the physical properties of the feed material had been specified. The physical properties specified for the simulation are shown in Table, 3. [11]

Table 3: Physical properties specified for the simulation

Density	0.22 Kg/m^3
Vapour Flow	1734182 m^3/h
Re	1022236
Viscosity	0.000011 kg/(m·s)
Turbulent Intensity	2.84
Backflow Turbulent Intensity	2.837436
Velocity of Inlet	136.365 m/s
Pressure	1 atm

In this study, the simulations of feed inlet device in a column have been done and two simulations with two different feed have been carried out. In section 3.1, the flow distribution of vapour inside the column has been shown. The feed used in this section is pure vapour feed. The residual plot, contours of velocity magnitude at both outlets, velocity streamlines have been shown in this section. And in section 3.2, the feed used is a mixture of vapour and liquid and the flow distributions of vapour have been explained/shown. The residual plot, contours, velocity streamlines have been shown/explained as in previous section. The flow distributions through the contours of velocity magnitude at the inlet device for both kinds of feed have been shown in section 3.3. And a comparison of the contours of velocity magnitude taken from this simulation study and contours taken from earlier simulation study has been done in this section.

Pure vapour feed

Now, the two different kind of feed has been provided for simulation and the quality of vapour velocity contours for both have been assessed. First pure vapour stream was introduced in the column through vane type feed inlet device. The physical properties of water vapour taken as our feed material of simulation have been specified above. And then the simulation had been run with K-E model and using second order discretization. By varying the under-relaxation factors and discretization schemes, a number of times and after some 1600 iterations, convergence of order 1*e-4 had been reached. After reaching the convergence, developed system had been considered as stable and hydrodynamic behaviour of vapour in the column at different sections had been analysed by finding out the flow velocity contours and velocity magnitude streamlines.

Figure, 6 shows a residual plot between various convergence deciding parameters and number of iterations. The different y- ordinate parameters correspond to different colours. Figure, 7 shows contours of velocity magnitude for vapour at vapour outlet of the column. An evenly distributed vapour flow can be seen from the contours where blue colour refers to stationary conditions and red colour refers to 16 m/s velocity of the vapour.

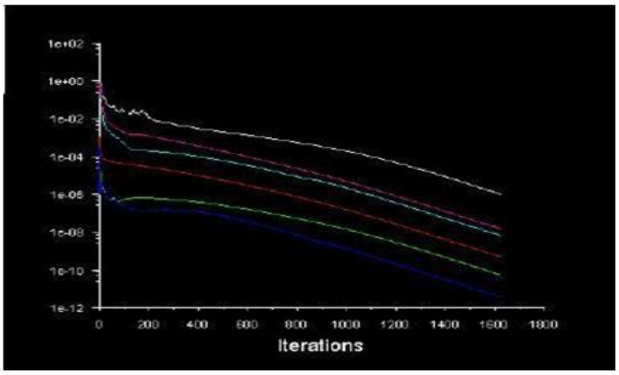

Figure 6. Residuals for vapour only feed. (White- continuity, Red- x.velocity, Green-y.velocity, Blue- z. velocity, Sky Blue- k, Pink- epsilon)

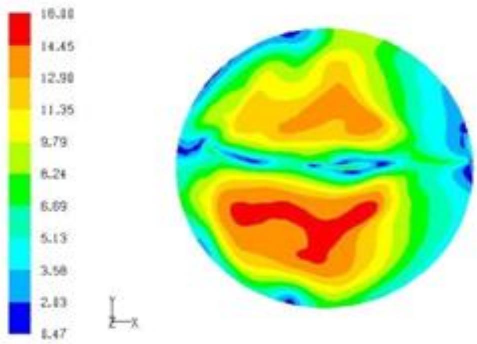

Figure 7. Contours of velocity magnitude at vapour outlet for pure vapour feed

Theoretically vapour should never come out of liquid outlet, but as CFD simulation provides nearby real life conditions, vapour flow was countered in the contours of velocity magnitude for vapour at liquid outlet which is shown in Figure, 8. The flow of vapour from the liquid outlet is also symmetrical from the center line of the contour where blue colour refers to stationary conditions and red colour refers to 29.03 m/s velocity of the vapour.

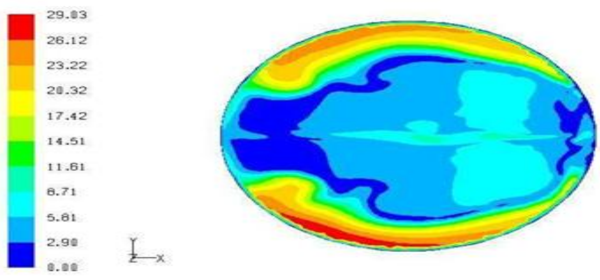

Figure 8. Contours of velocity magnitude at liquid outlet for pure vapour feed

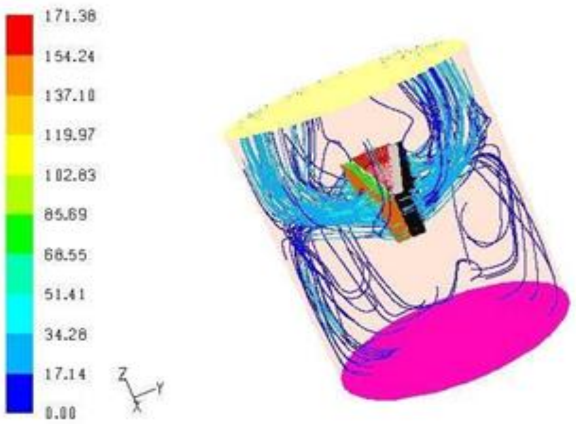

Figure 9. Contours of velocity magnitude streamlines for pure vapour feed

Contours of velocity magnitude streamlines shown in Figure, 9 shows the path which a vapour feed stream would follow after being splitted from the vane outlet of the inlet device. Streamlines of vapour flow shows an even vapor flow distribution on both sides of the inlet device.

Vapour-Liquid mixture feed

Second simulation had been carried out by introducing mixture feed in inlet device, the mixture being a vapour-liquid mixture. The physical properties of water vapour and liquid taken as our materials of simulation have been specified above and the properties which are not mentioned are used from default database of Fluent. And then the simulation had been run with K-E model and using second order discretization. By discretization scheme, a number of times and after some iterations, convergence of order 1*e-3 has been reached. After reaching the convergence, developed system had been considered as stable and hydrodynamic behaviour of vapour in the column at different sections had been analysed by finding out the flow velocity contours and velocity magnitude path lines. Figure, 10 shows a residual plot between various convergence deciding parameters and number of iterations. The different y- ordinate parameters correspond to different colours. Figure, 11 shows contours of velocity magnitude for vapour at vapour outlet of the column. An evenly distributed vapour flow can be seen from the contours where blue colour refers to stationary conditions and red colour refers to 7.41 m/s velocity of the vapour.

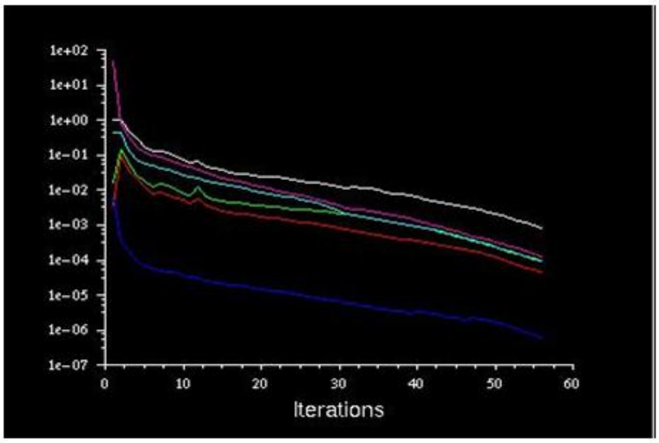

Figure. 10. Residuals for vapour-liquid feed. (White-continuity, Red- x.velocity, Green-y.velocity, Blue- z. velocity, Sky Blue- k, Pink- epsilon)

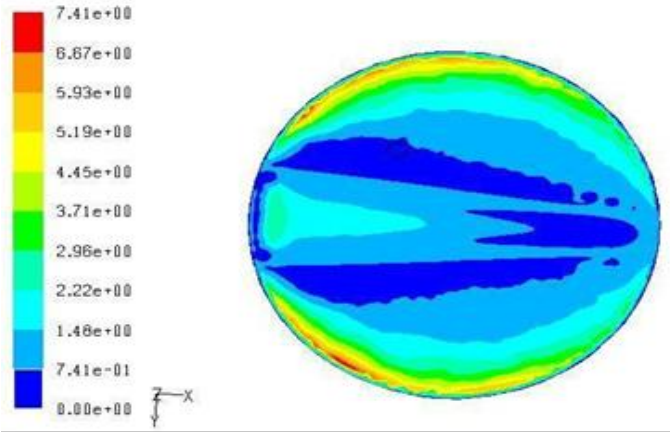

Figure 11. Contours of velocity magnitude at vapour outlet for mixture feed

Theoretically vapour should never come out of liquid outlet, but as CFD simulation provides nearby real life conditions, vapour flow was countered in the contours of velocity magnitude for vapour at liquid outlet which is shown in Figure, 12. The flow of vapour from the liquid outlet is also symmetrical from the center line of the contour where blue colour refers to stationary conditions and red colour refers to 15 m/s velocity of the vapour.

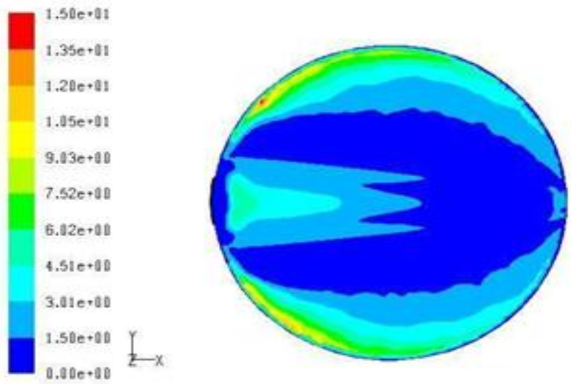

Figure 12. Contours of velocity magnitude at liquid outlet for mixture feed

Contours of velocity magnitude streamlines shown in Figure, 13 shows the path which a vapour feed stream would follow after being splitted from the vane outlet of the inlet device. Streamlines of vapour flow shows an even vapor flow distribution on both sides of the inlet device.

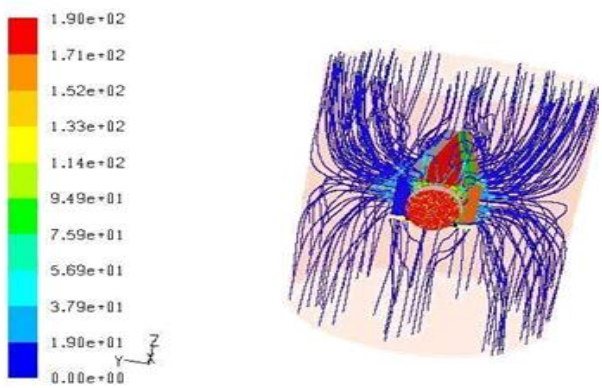

Figure. 13. Contours of velocity magnitude streamlines for mixture feed

Flow distribution comparison (Previous and new simulations)

Flow distribution of vapour at the inlet device for both kinds of feed streams had been studied through the contours of velocity magnitude shown in figure, 15 and figure, 16. Previous simulations to analyse the vapour velocity profile at the inlet device had been done by the sulzer, the contours of which are shown in figure, 14.

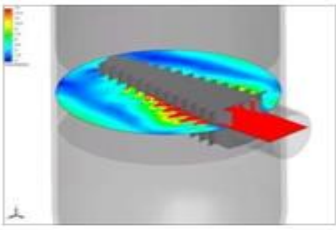

Figure. 14. Theoretical contours of velocity magnitude at inlet device

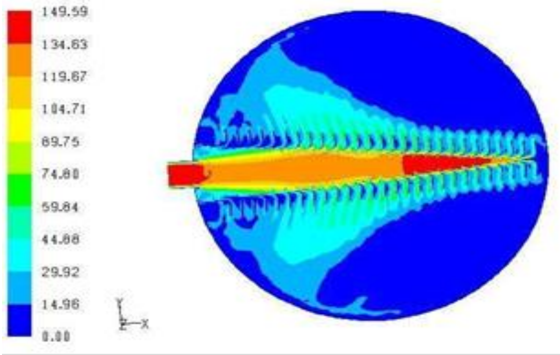

Figure 15. Contours of velocity magnitude at inlet device for pure vapour feed

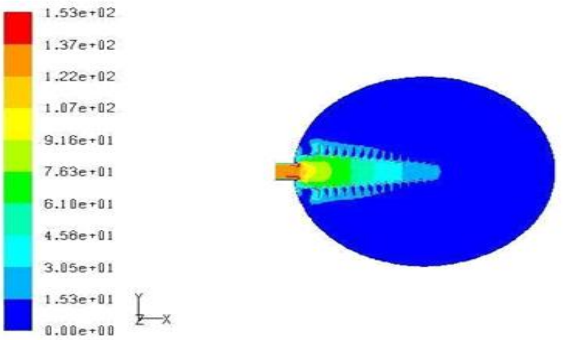

Figure. 16. Contours of velocity magnitude at inlet device for mixture feed

Conclusion

The purpose of a vane type feed inlet device is to decrease the momentum of the feed, perform a first stage separation of solids and liquid from the vapor, and achieve an even vapor distribution across the vessel cross section. This is obtained by splitting the feed mixture into a series of flat jets. A CFD simulation to see the action of vane type feed inlet device on the momentum of the feed and the separation had been done and the contours of velocity magnitude at various sections of a vacuum column had been taken. The CFD simulation had been done with introducing two types of feed, pure vapor feed and vapor-liquid mixture feed; and the contours of velocity magnitude had been taken and analyzed which shows our purpose to reduce the momentum of the feed and achieve an even vapor distribution across the vessel cross section had been reached. And a comparison of the simulated velocity magnitude contours had been done with the velocity magnitude contours taken from simulation study done by sulzer which shows an even flow distribution have been reached with a vane type feed inlet device in a vacuum column as the simulated results are comparable with the previous simulation results.

References

[1] Mark Pilling, Mario Roza and S. M. Wong, Sulzer Chemtech (2010).

[2] M. Casey, E. Lang, R. Mack, R. Schlegel, M. Wehrli (1998), Speedup, 12, 43 -51.

[3] M. Fleischli, R. Schlegel, M. Wehrli (2001), International Symposium on Mixing in Industrial Processes ISMIP4, Toulouse, May 14-16, France.

[4] L. A. Muir, C. L. Briens (1986), The Canadian J. Chem. Eng., 64, 1027 -1032.

[5] F. Moore, F. Rukovena (1987), Chemical Plants & Processing, August, 11 -15.

[6] L. Fan, G. Chen, S. Constanzo, A. Lee (1997), ICHEME Symposium Series 142, 899-910.

[7] X. Yuan, W. Li (1997), ICHEME Symposium Series 142, 931 -938.

[8] Ph. Suess (1992), ICHEME Symposium Series 128, A369 – 383.

[9] S. Vaidyanathan, M. Wehrli, H. Ulrich, R. Hunkeler (2001), Distillation Symposium, AIChE Spring National Meeting, April 24, Houston, Texas.

[10] Separation-Technology/Feed-Inlet-Devices/Shell-Schoepentoeter-and-Schoepentoeter-Plus.

[11] Confidential document, EIL. (Project at vocational training).