CFD Simulation and Experimental Study of New Developed Centrifugal Trays

N. Naziri, R. Zadghaffari, and H. Naziri

Abstract—**In this work, a computational fluid dynamics (CFD) model is developed to predict the hydrodynamics of industrial-scale Centrifugal tray that is a new kind of tray and can be used in distillation and absorption towers. Three-dimensional two-phase flow of gas and liquid is considered in which the interaction was modeled based on the concept of phasic volume defined in the Volume of Fluid (VOF) multiphase model through k-epsilon Simulation. All governing equations including surface tension and wall adhesion are solved simultaneously using the FLUENT code. Gas-liquid interfaces and the existence of froth regime are clearly visualized via VOF model. Due to the importance of trays, it is vital to predict lower operating range limits for such gas-liquid contacting devices. Weeping phenomenon is observed in bubbling regime and occurs at low vapor flow rates. To approach this aim, continuity and momentum equations are solved simultaneously for the fluid on tray and the results were compared with data of sieve trays. In the Comparison of sieve type trays with centrifugal chimney type, in low loading vapor, no weeping was observed in the centrifugal tray. The simulation results show that this innovated centrifugal tray can be used for solving the weeping problem and increasing the performance of tray towers**

Index Terms—**Computational fluid dynamics, centrifugal tray, volume of fluid model, weeping.**

I. Introduction

Distillation is the most widely used separation technique and usually the first choice for separating mixtures. Distillation tray design is a combination of theoretical and empirical challenges. Good design leads to good phase contacts with acceptable mass transfer and efficiency, thus trays must have flexibility to operate in satisfactory region of operating conditions. Such regions are called as operating windows or performance diagram of trays. The vapor and liquid rates will determine the operating window of trays. At low vapor rate trays will weep, at high rates, froth touches the above upper tray and entrainment will start. Upper and lower operating limits of the trays are highly affected by these two phenomena. Since many distillation columns operate at less than their design capacity, it is important to determine the weeping for trays. The dry tray pressure drop and the weep fraction are two vital hydraulic parameters that determine the lower operating limit for a tray. At reduced vapor rates, weeping becomes a problem in tray columns because it reduces tray efficiency and so adversely affects the separation achieved by the column [1]-[3].

One of the most modern trays that are designed by using latest science and modern technologies is centrifugal tray with mechanism of centrifugal force and special design in order to separate phases. Some of the features of this kind of tray include high capacity, efficiency, mechanical resistance and also reduction of foaming and priming. These trays have considerable differences with common trays from the view point of geometrical construction, so that this varying construction has great influence on the extent of separation, range and their operational quality. The centrifugal tray belongs to a new generation of mass transfer trays that can increase the capacity and efficiency profile of tray columns significantly. This increase is achieved by a special vapor and liquid distribution and also contact system installed on this tray.

The goal of this research was to examine the CFD models ability for prediction of the Centrifugal trays' lower operating limit in industrial scale and the result were compared with the CFD data of Rahimi et al [4].The main objective is to find the extent to which CFD can be used as a design and prediction tool for real behavior of industrial trays.

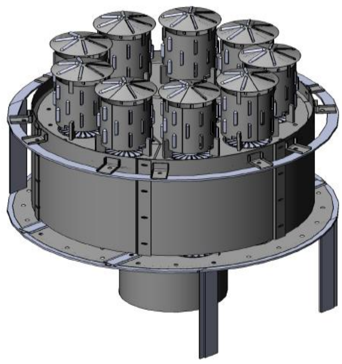

Fig. 1. Isometric view of computational geometry

II. Computational Geometry

Three-dimensional geometry is developed using commercial software called GAMBIT. The computational domain models an industrial-scale centrifugal tray with the diameter of 1.2 m. Detail specifications of the Centrifugal tray geometry are presented in Table I. Only 1/9 tray is modeled following observation of symmetric flow. Such configuration results in relatively less computational cost and time. Fig.1 shows the configuration of the system that has been simulated. The computational domain consists of hexahedral meshing elements. The number of computational cells generated from 698458 cells are shown in Fig. 2.

Manuscript received May 10, 2012; revised June 20, 2012.

N. Naziri is the Technical Manager of Azar Energy Comapany (e-mail: info@azarenergy.com).

R. Zadghaffari is with the Department, Azar Energy Co., Tabriz, Iran (e-mail: info@azarenergy.com).

H. Naziri is with R&D Department, Azar Energy Co, Tabriz, Iran (e-mail: info@azarenergy.com).

TABLE I: CENTRIFUGAL TRAY SPECIFICATIONS

Description	Dimension
Diameter	1.2 (m)
Weir height	0.06 (m)
Tray Spacing	0.6 (m)
Bubbling area	0.6 (m²)

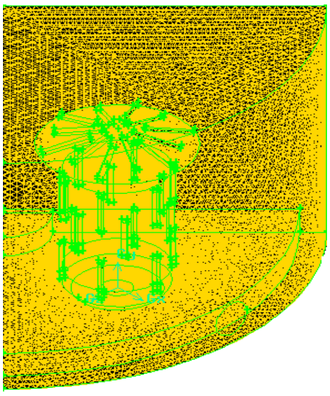

Fig. 2. Mesh configurations of the computational domain

III. Governing Equations

In the proposed VOF multiphase model the tracking of gas-liquid interfaces is accomplished by the solution of a continuity equation for the volume fraction (α) of the two phases. Since the main focus of this work is on the hydrodynamic behavior of Centrifugal trays, mass and energy transfers are not considered. Hence for the qth phase, the continuity equation has the following form:

$$\frac{\partial \alpha_q \rho_q}{\partial t} + \left(\nabla \cdot \alpha_q \rho_q u\right) = 0 \quad (1)$$

where u represents velocity. Having constant density (ρ) and viscosity (μ), (1) reduces to:

$$\left(\nabla \cdot \alpha_q u\right) = 0 \quad (2)$$

The volume fraction equation only solves for the secondary liquid phase whilst the primary gas phase volume fraction is computed based on the following constraint:

$$\sum_{q=1}^{n} \alpha_q = 1 \quad (3)$$

The momentum equation is given by:

$$\rho\left(\frac{u_i}{\partial t} + u(\nabla \cdot u)\right) = -\nabla p + \mu \nabla^2 u + \rho g + F_{vol} \quad (4)$$

where P indicates pressure whilst g and F_{vol} represents gravitational acceleration and volume force respectively. Continuum surface force (CSF) [5] was used to model the surface tension which was taken into account via the source term, F_{vol} , in the momentum equation. For two-phase flow, the volume force is defined by:

$$F_{vol} = \sigma_{ij} \frac{\rho k_i \nabla \alpha_i}{0.5(\rho_i + \rho_j)} \quad (5)$$

where subscript i and j denotes volume phases; σ represents the surface tension coefficient; k represents the curvature defined by the divergence of the unit normal, $\hat{n}$:

$$k = \nabla \cdot \hat{n} \quad (6)$$

where

$$\hat{n} = \frac{n}{|n|} \quad (7)$$

In which

$$n = \nabla \alpha_q \quad (8)$$

The significance of surface tension was determined by evaluating the Weber number, W_e which is given by:

$$W_e = \frac{\sigma}{\rho_L L U^2} \quad (9)$$

where ρ_L represents liquid density; U represents the free-stream velocity and L represents the clearance under the downcomer. $W_e >> 1$ indicates that the presence of surface tension is significant and should not be neglected.

When applying the time average procedure in the Navier-Stokes equations, an extra term appears due to the turbulent fluctuation of the velocities, which needs to be represented by a constitutive equation. This term is known as the Reynolds flux. We consider that the fluctuations (turbulence) reflect the formation and dispersion of small swarms of bubbles, and that the Reynolds stresses can be linearly related to the mean velocity gradients (eddy viscosity hypothesis) as in the relationship between the stress and strain tensors in laminar Newtonian flow, and thus an effective viscosity can be assumed:

$$\mu_{eff} = \mu + \mu_t \quad (10)$$

The turbulence viscosities were related to the mean flow variables by using the standard k- ε model. The k - ε model is well treated in literature [6, 7].The standard k-ε model is related to the turbulent kinetic energy and its dissipation rate as follows:

$$\mu_t = C_\mu \rho\left(\frac{K^2}{\varepsilon}\right) \quad (11)$$

where k is the turbulent kinetic energy and ε is the dissipation rate of the turbulent kinetic energy. The conservation equation for turbulent kinetic energy and its dissipation rate can be written as:

$$\frac{\partial}{\partial t}(r_\alpha \rho_\alpha K_\alpha) + \nabla \cdot \left(r_\alpha \left(\rho_\alpha U_\alpha K_\alpha - \left(\mu + \frac{\mu_{t\alpha}}{\sigma_k}\right) \nabla K_\alpha\right)\right) = r_\alpha (P_\alpha - \rho_\alpha \varepsilon_\alpha) + T_{\alpha\beta}^{(K)} \quad (12)$$

$$\frac{\partial}{\partial t}(r_\alpha \rho_\alpha \varepsilon_\alpha) + \nabla \cdot \left(r_\alpha \rho_\alpha U_\alpha \varepsilon_\alpha - \left(\mu + \frac{\mu_{t\alpha}}{\sigma_{\varepsilon 1}}\right) \nabla \varepsilon_\alpha\right) = r_\alpha \frac{\varepsilon_\alpha}{K_\alpha}(C_{\varepsilon 1} P_\alpha - C_{\varepsilon 2} \rho_\alpha \varepsilon_\alpha) + T_{\alpha\beta}^{(\varepsilon)} \quad (13)$$

where $C_{\varepsilon 1}$, $C_{\varepsilon 2}$, σ_k and $\sigma_{\varepsilon 1}$ are the model constants and $T_{\alpha\beta}^{(k)}$ and $T_{\alpha\beta}^{(\varepsilon)}$ are coefficients of mass transfer between phases. All of the above governing equations, (1) – (13), were solved simultaneously using the commercial code called FLUENT 6.3.

IV. Boundary Conditions

A. Liquid Inlet

Proper boundary conditions have to be defined in order to solve the equations set uniquely. Velocity-inlet was specified as the liquid inlet boundary condition. Experimentally, tray by tray measurement of liquid mass flow rate entering from downcomer clearance is difficult to conduct and accuracy is always an issue. Most of the time, it is more convenient to express this quantity in term of liquid volumetric flow rate, *QL*, given by:

$$u_{L,in} = \frac{Q_L}{A_{da}} \qquad (14)$$

The area under the downconmer apron, A_{da} is given by the product of downcomer clearance height, hcl and weir length, lw.

B. Gas Inlet

Similarly, velocity-inlet was specified like the gas inlet boundary condition. Direct measurement of gas velocity is often difficult to conduct experimentally; hence, a more convenient way is to express this quantity in term of superficial F-factor, often represented by F_S. This quantity is defined by the product of gas superficial velocity, V_S and square root of gas phase density, ρ_V as shown in the following:

$$F_S = V_S\sqrt{\rho_V} \qquad (15)$$

Taking V_S as subject, (15) is then rearranged to :

$$V_S = \frac{F_S}{\sqrt{\rho_V}} \qquad (16)$$

For any given vapor load, the vapor superficial velocity can be calculated via (16). Having the bubbling area A_B as the basis of superficial velocity, the gas inlet velocity is then given by:

$$u_{h,i} = \left(\frac{V_S A_B}{2N_H}\right)\frac{1}{A_{h,i}} \qquad (17)$$

N_H Represents the number of holes in the model geometry and $A_{h,i}$ stands for the whole area.

C. Liquid and Gas Outlet

Pressure-outlet was specified as the liquid- and gas- outlet boundary conditions with backflow volume fraction of unity assigned at the secondary liquid-phase. Liquid and vapor flow across the tray are driven by the pressure gradient between the inlet and outlet of both entity.

D. Geometry Wall

No slip wall condition was specified to all wall boundaries.

V. Operating Conditions and Solution Algorithms

The gas and liquid were simulated at atmospheric pressure. Air and water are employed as the working fluids with air being specified as the primary gas phase whilst water being assigned as the secondary liquid phase. The simulation is initialized by patching the liquid inlet surface with $\alpha_L = 1$ so that gas-liquid interfaces can be tracked immediately from the point of release. Apart from this, each time step was assumed to have been fully converged whenever the continuity equation absolute criterion attained the value of 0.001. Solution algorithms are introduced to enhance or accelerate the convergence of this simulation. The pressure velocity coupling was handled by the Semi-implicit Method for Pressure-linked Equation (SIMPLE) algorithm whilst Pressure Implicit with Splitting Operators (PRESTO) is used as the discretization method.

VI. Results Analyses and Discussions

Weeping is liquid descending through the tray perforations. Under weeping conditions, part of the liquid flows over the outlet weir while the rest descends through the perforations. The liquid descending through the perforations short-circuits the primary contacting zone, causing a reduction in tray efficiency. At the tray floor, the static liquid head tends to force liquid down through the perforations. The vapor pressure drop counteracts the downward force and acts to keep liquid on the tray. Weeping takes place when the liquid head on the tray exceeds the pressure drop that is holding the liquid on the tray. With the pressure essentially constant in the vapor space below, weeping occurs at those locations where the liquid head is temporarily high, even when the average head on the plate does not exceed the tray vapor pressure drop[3], [8].

Fig. 3 shows weeping rate as a function of F_S at liquid rates of 60m3/mh. The results were compared with data of sieve trays [4]. Details of trays specifications are given in Table II.

TABLE II: Specifications of Test Sieve Trays

Tray diameter	Plate active area	hole area%	Number of holes	Downcomer area
1.2 m	1.0078 m ²	7.04	560	0.061608 m ²
Number of sieve trays	Hole diameter	Tray thickness	Weir height	Downcomer clearance
2	12.7 mm	2mm	50mm	40mm

Results show that in low loading of vapor, weeping does not occur in centrifugal tray. The most important factors that make this centrifugal tray unique are as follow:

- In centrifugal tray, liquid flow on tray has irregular behavior and rotating flow of liquid can be seen completely (Fig. 4)
- According to rotating flow, phase distribution of liquid on tray becomes uniform and resistance against gas flow becomes the same in the whole surface.
- There is a possibility of gas penetration from all the existing holes in the surface of the tray and in fact

efficient surface of tray will be increased and dead surfaces in tray will be eliminated.

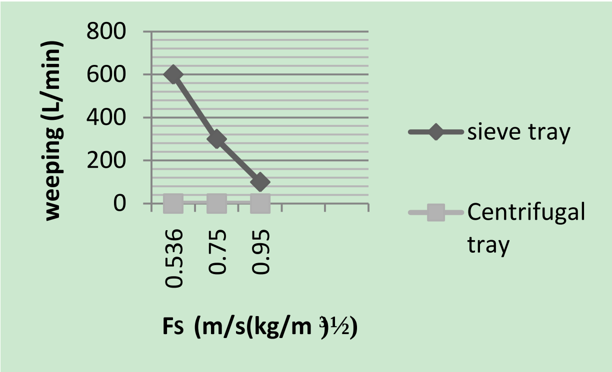

Fig. 3. CFD result for the weeping rate

One of the advantages of using vapor to influence liquid flow is that trays need to take some amount of vapor side pressure drop to maintain enough resistance to prevent weeping of liquid through the tray orifices.

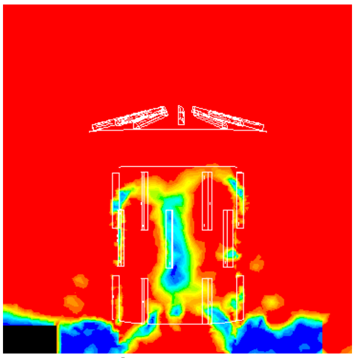

Fig. 4. Phase contours at z =0

The next figures present data of the main tray characteristic. Dry tray pressure drop is a hydraulic parameter which affects the weeping occurrence. Fig. 5 shows CFD results for the dry pressure drop as a function of the F-factor. Weeping occurs when the pressure drop of the passing vapor through the tray deck is insufficient to support the liquid. Therefore, by increasing the pressure drop, the liquid turndown ratio increases.

According to procedure of gas and liquid current on centrifugal tray, these two help each other to move and flow. So resistance of liquid phase against the gas phase will be very less and gas needs to move upward. So pressure drop in tower will be less than common value.

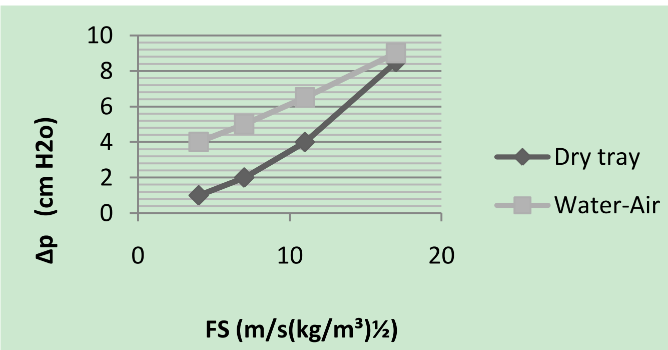

Fig. 5. Pressure drop for the centrifugal tray

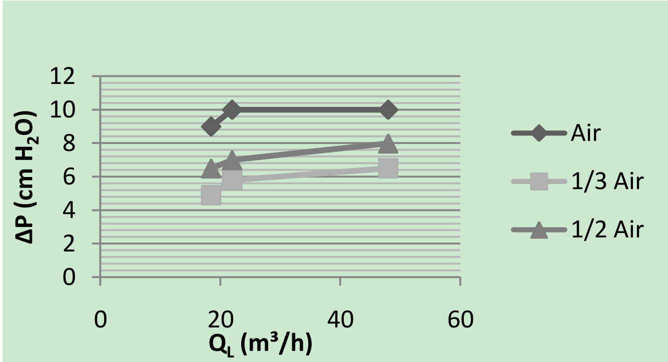

Fig. 6. Performance characteristics

One of the main problems that is seen in gas sweetening systems with Amine and also in the case of using common trays is the problem of entrainment and excluding of Amine accompanied by gas flow from the column that would result in Amine losses and require to be made up.

In Table III, the results of using new centrifugal tray in eliminating the problems of entrainment and excluding of Amine from system are presented during 9 month field study.

TABLE III: BEFORE *VS.* AFTER REVAMP PERFORMANCE COMPARISON

Centrifugal tray		Valve tray							
9	8	7	6	5	4	3	2	1	Month
0	0	13.5	8	6	14	18	16	16	Amine (Barrel)

Results of table III show that after the exchanging of valve trays with centrifugal tray, the problem of valve tray and Amine losses that is caused by not proper separation of gas and liquid in trays was completely solved and the performance of column was improved.

VII. NOMENCLATURE

A_B = tray bubbling area, m ²
A_{da} = down comer clearance area, m ²
A_h = hole area, m ²
$C_{\varepsilon 1}, C_{\varepsilon 2}$ = model constants
F_S = F factor $= V_S\sqrt{\rho_V}$, m/s(kg/m ³)½
F_{vol} = Volume force= $\sigma_{ij}\,\frac{\rho k_i \nabla\alpha_i}{0.5(\rho_i+\rho_j)}$
g = gravity acceleration, m/s ²
hcl= clearance height, m
k = turbulent kinetic energy
lw =weir length, m
$\hat{n}$ = unit normal
N_H = number of holes
P = total pressure, Nm
Q_L = liquid volumetric flow rate, m ³/s
$T_{\alpha\beta}^{(k)}$, $T_{\alpha\beta}^{(\varepsilon)}$ = coefficients of mass transfer between phases, kmol/m ²s
t = time, s
u = velocity, m/s
u_h = gas velocity, m/s
u_L = liquid velocity, m/s
VS = gas phase superficial velocity based on bubbling area, m/s
W_e = Weber number
Greek letters

α= volume fraction of phases
ρ= density, kg/m ^{3}
ρ_V = gas phase density
ρ_L = liquid phase density
μ = viscosity, kg.m^{-1}.s^{-1}
μ_{eff} = effective viscosity, kg.m^{-1}.s^{-1}
μ_t = turbulence viscosities, kg.m^{-1}.s^{-1}
ε= dissipation rate of the turbulent kinetic energy
σ = surface tension coefficient
σ_k = model constants

REFERENCE

[1] M. W. Biddulph, "Oscillating Behavior on Distillation trays," *AIChE J*, vol. 21, no. 41, 1975.

[2] D. R. Summers, "High Capacity Tray Revamp of C_2 Splitter," *Sulzer Chemtech USA*, Inc, AIChE Annual Austin, Texas, 2004

[3] H. Z. Kister, "Distillation Design," *McGraw-Hill*, 1992.

[4] R. Rahimi, T. Zarei, and N. Naziri, "A study on sieve tray lower operating limit," *presented at Distillation Absorption, Eindhoven*, The Netherlands September 12-15, 2010

[5] J. U. Brackbill, D. B. Kothe, and C. Zemach, "A continuum method for modelling surface tension," *Journal of Computational Physics*, vol. 100, pp. 335-354, 1992.

[6] S. B. Pope, "Turbulant flows," *Cambridge Univ. Press, Cambridge*, 2000.

[7] D. C. Wilcox, "Turbulence modelling for CFD," *DCW Industries Inc*, La canada, CA, 1993.

[8] M. J. Lockett, "Distillation Tray Fundamentals," *Cambridge University Press*, 1986.

Nader Naziri Shiraz was born in Iran, 1956. He was graduated from Institute of Technology of Tabriz, Iran in Mechanical course,1979 and has a self study in chemical Engineering. He is the TECHNICAL MANAGER of Azar Energy Company. He has 25 years of experience in troubleshooting and revamping of refineries and petrochemicals of Iran such as 1. troubleshooting of foaling problem in Styrene Monomer plant of Tabriz Petrochemical, Iran, 2. Increasing the capacity and decreasing of H_2S by using new centrifugal trays innovated by Nader Naziri in Amine tower of Tabriz Petrochemical, Iran, … . he has 4 patents in the field of chemical engineering and also different articles including "A study on sieve tray lower operating limit" presented at Distillation Absorption conference, Eindhoven , The Netherlands, 12-15 September 2010.He published a book, *corrosion in refineries* , Tabriz, Iran:Azar Energy,2010. Nader Naziri is a member of R&D Society of the Iranian Industries and Mines, Iran Inventors association and Society of Iranian petroleum industry equipment manufacturers. He received silver award from 1st International Technology, Invention & Innovation show of West Asia, 2011 and also an award from 1st Iran Oil refining Forum, Tehran, Iran,2007. You can visit www.azarenergy.com for more information.

Ramin Zadghaffari Tabriz was born in Iran, 1974. He holds PhD in chemical engineering from Sahand university of technology, Iran Ramin Zadghaffari is MANAGER of Paya Process Co.,Tabriz, Iran and also the MANAGER of CFD DEPARTMENT for Azar Energy Co., Iran He has 12 years of experience in the design and engineering of chemical, petrochemical and refining projects. He has authored several papers and presentations in various fields of chemical engineering.

Hamed Naziri Azarshahr was born in Iran, 1991. He is a student of chemical engineering in Sahand University of Technology, Iran. Hamed Naziri is the MANAGER of EXPERIMENTAL SECTION of R&D DEPARTMENT for Azar Energy Company from 2008. He has a patent on the invention of "Co-Current Trays". He also has an article of "A Computational Fluid Dynamics and an Experimental Approach to the Effects of Push Valves on Sieve Trays" presented at Distillation Absorption conference, Eindhoven , The Netherlands, 12-15 September 2010