J. Basic. Appl. Sci. Res., 2(7)7030-7040, 2012
© 2012, TextRoad Publication

ISSN 2090-4304
Journal of Basic and Applied Scientific Research
www.textroad.com

Numerical Study of Fluid Flow in Dew Point Setting Unit Separators in Sarkhoun and Qeshm Gas Treating Company

Y.Bakhshan[1], J.Khorshidi[*], H.Parvin[2], I.Naderipour[2], A.Ghobadi[2]

[1]Faculty of Science and Technology. University of Hormozgan, Iran
[2]Sarkhoun and Qeshm Gas Company (SQGC).Iran

ABSTRACT

In this paper, in order to research the effects of separator plates geometry on fluid flow behavior is used from three types of plate with different tips -Round, sharp and flat (main geometry)-for simulation was used 2dimension and 3dimension with numerical finite volume model. Then plate has been modeled 2 dimensional and for purpose of droplet tracking in continuous phase (CH_4) has been used discrete phase model (DPM) with turbulence Reynolds stresses model (RSM). Also Gambit software is used for mesh generation and boundary conditions and fluent software is used for solution of governing equations. After simulation, effect of growing of droplet, plate tip geometry, density of liquid droplet in continuous phase, distance between plates and pressure drop was studied. The result of research compared with experimental result in outlet of separator at Sarkhoun and Qeshm gas treating company that has 90% compatibility. The simulation result has shown that plate with flat tip has more compatibility with relation to other plates in that has acceptable pressure drop and separation efficiency. Also effective parameters in improvement of separation process are growing and increase density of liquid droplet that plates with flat tip duo to their special geometry have more compatibility for separation with present condition. The result of research is shown that in applicable condition (Rate of flow and properties of fluid), performance of flat tip plate is better than two other types.

KEY WORDS*:* simulation, separation, separator, plates.

INTRODUCTION

Separators are among equipments are used in order to separate phases of a mixture (solid-liquid -gas) in process industries, especially oil and gas industries from structure point of view separators are divided to 3 groups: horizontal separators are used to separate gas from mixture with low mass ratio of vapor to liquid in feed flow due to high separating power ، this separator is designed and used in most oil and gas (utilization) units in two types، low and high pressures [2, 3]. Horizontal separators contain 3 separating parts A،B،C as shown in figure 1.

Different facilities are installed inside of these separators to improve their performance. vane type separators are kinds of internal facilities that their wave shape vanes ، causes liquid phase separates from gas (figure 2).in these separators liquids drops passes with gas flow due to their weights liquid drops Gravity could not change direction with gas flow rapidly and as a result they stick to each other on the vanes and finally separate from gas flow and fall down Pay attention to horizontal separators widespread applications.

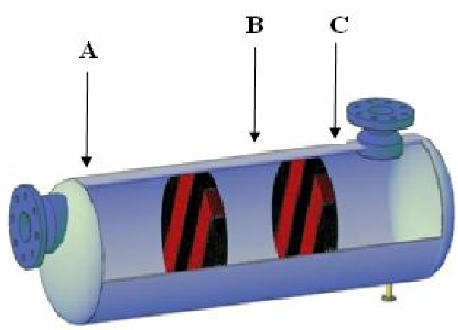


Fig.1. Horizontal separator internal parts

Horizontal separators are used widely on industry and many scientists are working to improve their performance and rapid improvement and available super computers with high speed processing ،using numerical

***Corresponding Author:** J.Khorshidi, Faculty of Science and Technology. University of Hormozgan, Iran.
E-mail: JKhorshidi@yahoo.com

method in computer calculations is considered as a profitable way for engineering tools design. Calculative fluid dynamic science becomes as a powerful tool for fluid flow behavior and heat transfer and complicated equations analysis for researchers and engineers and it has had a considerable importance during the last decade. Lee Jao and cooperators [4] studied liquid drops behavior state in a two phases vapor – water vane type separator with numerical method Ted Fronky and cooperators [5] studied flow simulation in separators and formation and liquid drops separating state using the calculative fluid dynamic method. This research shows appropriate agreement between simulation results and experimental results.

Fig.2. One ordinary trap vane type

Also computer simulation helped Nat co company engineering to design oil – gas separator ،50% smaller than the last models [6] in this research fluid flow study and water with gas separating state in a two phases vane type separators and also the effect of shape change of separating part blades on separating performance of liquid drops from gas phase will be considered.

Geometry

Studied separator consist of a cylindrical shape reservoir with 1500mm diameter and 5100mm length and two groups separator plate ،each with 200mm width .these blades are settled with 10mm distance .so that they fill all reservoir section area .view of this separator with all details and dimensions are given in figure3 and table 1 .

Table 1: Separator geometry

Length	5100 mm
Diameter	1500 mm
Distance between two baffles	170 mm
Distance between inlet unit first baffle	200 mm
Baffle thickness	1680 mm
Distance between outlet unit first baffle	4530 mm

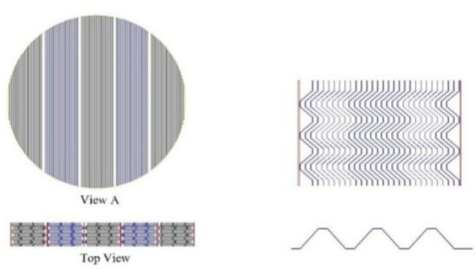


Fig.3. flat tip plate

Mesh generation

One of the important consideration in problem numerical solution ، is production and mesh accuracy of selected geometry .on the other hand in a numerical solution it is always necessary to have the calculation of way of problem mesh . i.e. the mesh should be selected so small as the mesh size decreasing could not effect on problem responses .To do so two parameters separating system performance and pressure drop for solution mesh quality study have been used. In figure 4 wave shape path of mesh has been shown.

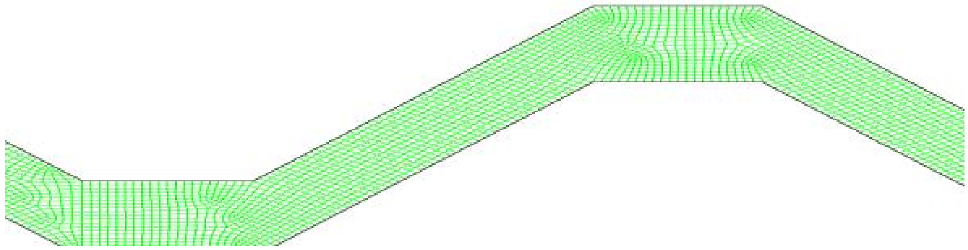


Fig.4. mesh generation

To study the effect of mesh and finding optimized mesh design of networking ،first a mesh with 6 ، 12 and 24 divisions was made which smaller mesh size causes change in separating performance. With more smaller mesh size up to 48 parts it has been noted that performance results is equal to the results of 24 parts mesh size which means more smaller mesh would make any difference on solution results. Also numerical solution is independent from mesh design of networking. The one of important parameters in separating systems design ،is the pressure drop through the length of reservoir. In figure 5 pressure difference between reservoir inlet and outlet with respect to the different numbers of mesh has been drawn .as it is seen ،pressure drop value for meshes with 6 and 12 mesh numbers have a lot of differences from 24 mesh numbers have . However mesh (n=24) has pressure near to pressure in mesh (n=48) ،a straight linear pressure drop curve confirms this statement .So it could result that for 24 mesh numbers ،problem solution is independent of applied mesh type.

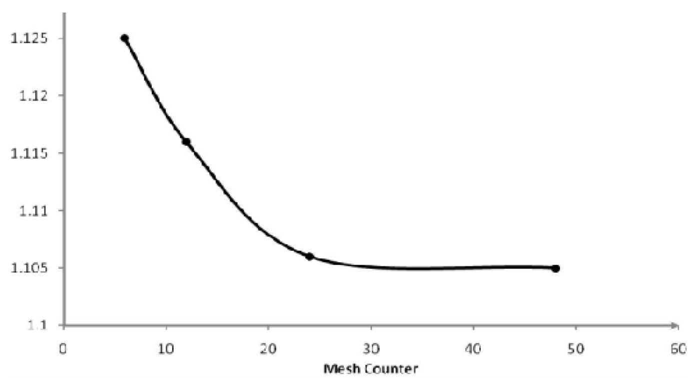


Fig.5. Inlet pressure change according to the mesh numbers.

As what described before in this step of problem solution ، a 24 numbers mesh ، as in the figure 6 ، sit basis of solution and due to problem physics it is changed slightly .i.e. smaller mesh has been placed in inlet to increase thrown sustained particles numbers. Also in addition due to importance of two blades ،mesh design (network) in this area has been more smaller (meshes with 0.25mm length) In addition because reservoir lengths much longer than blades length ،mesh in reservoir length has been done with ratio1.03 from two sides. (It means that length of each mesh is 1.03 times of its previous mesh).

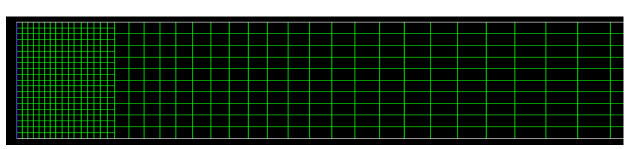


Fig.6. inlet mesh generation

Governing equations

Here, It is assumed that there is two dilute enough phase ،gas and liquid ،which mutual particles effects on each other and particle volume element effect on gas phase is negligible . Operationally ،This discussion confirms that discrete phase should be presented in a volume element that it is small enough usually less than 10-12 % notice that discrete phase mass loading may becomes more than 10 until 12%.

-Discrete phase equations:

Here particle path curve (drop or bubble) is obtained from discrete phase using integration particle force balance that in lagrangian forms. In this force balance ، particle inertia becomes equal with forces applied to particle and it could be written as follow: (For x direction in coordinate system) [7]:

$$\frac{du_p}{dt} = F_D(u - u_p) + \frac{g_x(p_p - p)}{p_p} + F_x \quad (1)$$

In which F_x is force acceleration acting on particle mass unit and $F_{(D)}(u - u_{(p)})$ was drag force for each particle mass unit and we have:

$$F_D = \frac{18_\mu}{P_p d_p^2} \frac{C_D R_e}{24} \quad (2)$$

u is fluid phase speed ، u_p is particle speed، μ is viscose fluid molecular speed ، ρ is fluid density ، ρ_p is particle density and d_p is particle diameter .Re is corresponding Renols number ، that is determined from following relation :

$$Re \equiv \frac{pd_p|u_p - u|}{\mu} \quad (3)$$

In the above relationship:

Drag factor C_d could be taken from each of two following equations:

$$C_D = a_1 + \frac{a_2}{Re} + \frac{a_3}{Re^2} \quad (4)$$

In which a_1،a_2،a_3 are constant and for calculating them we can use following relation presented by Aleksander and cooperators [8].

$$C_D = \frac{24}{Re_{sph}}\left(1 + b_1 Re_{sph^{b_2}}\right) + \frac{b_3 Re_{sph}}{b_4 + Re_{sph}} \quad (5)$$

In which (6)

$$b_1 = \exp(2.3288 - 6.4581\emptyset + 2.4486\emptyset^2)$$
$$b_2 = 0.0964 + 0.5565\emptyset$$
$$b_3 = \exp(4.905 - 13.8944\emptyset + 18.4222\emptyset^2 - 10.2599\emptyset^3)$$
$$b_4 = \exp(1.4681 + 12.2584\emptyset - 20.7322\emptyset^2 + 15.8855\emptyset^3)$$

That is derived from Hider and Lonspail study، shape factor (∅) is determined by the following equation:

$$\emptyset = \frac{s}{S} \quad (7)$$

In which s is a sphere area with a particle equivalent volume and S is a particle real area with a particle real area. Reynolds number Re_{spn} is calculated by a sphere diameter with equivalent volume .For particle smaller than micron ، there is Stocks drag rule. In case F_D is derived from following equation:

$$F_D = \frac{18_\mu}{d_{p^2} P_p C_c} \quad (8)$$

Factor C_c is Kaningham correction for stocks drag rule that can be calculate from

$$C_c = 1 + \frac{2\lambda}{d_p}\left(1.257 + 0.4\, e^{-\left(\frac{1.1 d_p}{2\lambda}\right)}\right) \quad (9)$$

In which λ, is molecular average free path.

For fluid spray, the most appropriate drop size distribution expression is Rozen - Ramler expression which is a complete limit from sizes divides into the enough separate distance numbers that each one of them is shown with an average diameter and path curve calculation is done. On the basis of them if sizes distribution is Rozin - Ramler type from drop mass element that their diameters are bigger than d ، is calculated from relation10:

$$Y_d = e^{-(d/d)^n} \quad (10)$$

In which d is size constant and n is size distribution parameter.

-Turbulent Model:

One of the phenomena that have an important effect in sustained particles separating is internal rotational flow at the adjacent available corners in blades geometry .angular area in geometry causes increase in flow turbulence as a result of rotational flow in this area. This rotation causes increasing probability of sustained particles contact to walls and consequently more particle numbers are trapped. For fluid flow turbulent modeling ، turbulent equations are used. These equations specify flow lobar motion in high turbulence. In some conditions where there is turbulence in flow ، It is necessary that turbulent equations have been solved as well as momentum pressure equations .A lot of models are presented for turbulent modeling k-ε is one of the most reliable presented models is considered for equations Collection for turbulent parameters [9 , 10].

$$\mu_{eff} = \mu + \mu_T \qquad \mu_T = pC_\mu \frac{k^2}{\varepsilon} \tag{11}$$

$$\frac{\partial(\rho k)}{\partial t} + div(pkU) = div\left[\frac{\mu_{eff}}{\sigma_l} grad.k\right] + G - \rho\varepsilon \tag{12}$$

$$\frac{\partial(p\varepsilon)}{\partial t} + div(\rho\varepsilon U) = div\left[\frac{\mu_{eff}}{\sigma_l} grad.\varepsilon\right] + G\frac{\varepsilon}{k} 2\mu_{eff} E_{ij}.E_{ij} - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \tag{13}$$

Which: C_μ , $C1_\varepsilon$, and $C2_\varepsilon$ are k-ε model factors. μ_{eff} , μ_t and μ are effective viscosity ، turbulent and laminar،E_{ij} is line shape change speed ، k is turbulent kinetic energy and σ , εσ , σk are k-ε model factors ، therefore using appropriate turbulent model has a great importance in problem analysis and achieving acceptable results because turbulent different models ability in various problems analysis are different from each other (as an example all of them aren't able to predict flow rotational area ، ability of 3 current turbulent models- standard k-ε model، RNG k-ε model and Reynolds stresses model (RSM)are compared in predicting flow rotational area).As it can be seen just RMS model is able to show flow rotation in these areas .Therefore in this research Reynolds stresses model(RSM) in turbulent behavior determination is used .

Boundary conditions

Mass flow distribution in vessel inlet is uniform and equal with 99403.13 kg/hr and or 1712.76 m^3/hr. uniform mass flow makes it possible that in order to decrease the mesh numbers and as a result calculation time decreasing ،just space between two adjacent blades was studied (In this basis ، noting distance between two blades،flow speed at the inlet to space between two blades is given equal to 0.269 m/s).Also as blades are obstacles to present condensates flow ، blades height can be neglected and in the first step and analysis can be done in a two dimensional domain. In this way mesh volume decreases by considerable rate and calculations performing time decreases extremely. Noticing separator operational conditions، considered materials for methane fluid flow methane(CH_4)with constant density 58 kg/m^3 also pentane(C_5H_{12})، benzene(C_6H_6)،hexane(C_6H_{14}) ، heptanes (C_7H_{16}) ، octane(C_8H_{18}) ، diesel($C_{10}H_{22}$) ، gasoline($C_{16}H_{22}$) and fuel oil ($C_{19}H_{30}$) as existent sustained particles in fluid flow have been studied. In this case ، injection is done planar and from the inlet. Injected particles diameter is considered constant Between 30 to 110 microns. By notice that particles since particles have been influenced by inlet flow rapidly ، speed determination for injected particles hasn't any effect on solution procedures، so here pre-assumptive values have been used and in the table 2 is shown in sum.

Table 2: operational properties of inlet fluid

Materials	Density	Phase type
methane(CH_4)	58 Kg/m^3	continuous
pentane(C_5H_{12})	626 Kg/m^3	discrete
benzene(C_6H_6)	875 Kg/m^3	discrete
hexane(C_6H_{14})	660 Kg/m^3	discrete
Heptanes (C_7H_{16})	684 Kg/m^3	discrete
gasoline($C_{16}H_{22}$)	830 Kg/m^3	discrete
diesel($C_{10}H_{22}$)	730 Kg/m^3	discrete
octane(C_8H_{18})	720 Kg/m^3	discrete
fuel oil ($C_{19}H_{30}$)	960 Kg/m^3	discrete

Also considering real operational conditions in the refinery ، inlet fluid to separators analysis has been shown in table 3 , pressure in operational conditions is 69.6bar.

Table3: Inlet fluid conditions

Inlet flow analysis	Mol Percent		Inlet flow analysis	Analysis results	
O_2	Nil	Mol. %	H_2S	--	Ppmv
N_2	5.16	Mol. %	Gross Heat Value	9850.6	Kcal/sm^3
Co_2	0.41	Mol. %	Specific Gravity	0.6780	
CH_4	84.74	Mol. %	Net Heat Value	8913.9	Kcal/sm^3
C_2H_6	4.39	Mol. %	Water Dew Point	--	^oC
C_3H_8	2.24	Mol. %	Hydrocarbon Dew Point	--	^oC
$i-C_4H_{10}$	0.63	Mol. %	Line Temperature	-10	^oC
$n-C_4H_{10}$	0.93	Mol. %	Line Pressure	69	barg
$i-C_5H_{12}$	0.39	Mol. %	Molecular Weight	19.64	Gr/mol
$n-C_5H_{12}$	0.30	Mol. %			
C_6^+	0.81	Mol. %			

Here methane fluid at the inlet to separators and has been considered without any rotation and speed and temperature values in the inlet and outlet are shown in table 4.Of course according to the existent PFD drawings، pressure in the outlet is equal to 69.66bar which considering operational pressure (69.6bar)، relative pressure is zero. Marginal conditions for a section that specifies fluid environment in flow has been selected from symmetric type. The reason for this selection is to prevent marginal layer formation (on the other hand ،to become zero speed on the marginal).It should be noted that in this marginal condition setting trapping of sustained drop in fluid flow must be detected and considered.

Table4: Boundary condition

Boundary conditions	Pressure(bar)	Speed (Mesc)	Temperature(k)
Inlet	-	0.269	311
Outlet	69.6	-	311
Blades	-	0.0	0.0

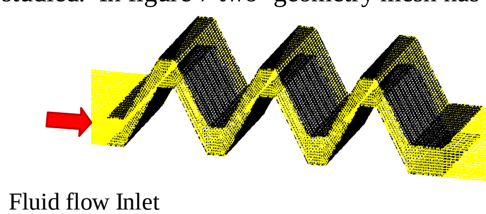
RESULTS

In this section given results from problem numerical solution are presented and different ingredient effect such as ، blades tip shape ، angle and blades ، distance z sustained particles diameter in fluid flow ، drops kind and etc on separator behavior and its performance have been studied . So ، first original geometry due to mesh network high volume ، has become simple and new geometry is studied and then geometry optimization and also fluid behavior in different operational conditions is calculated.

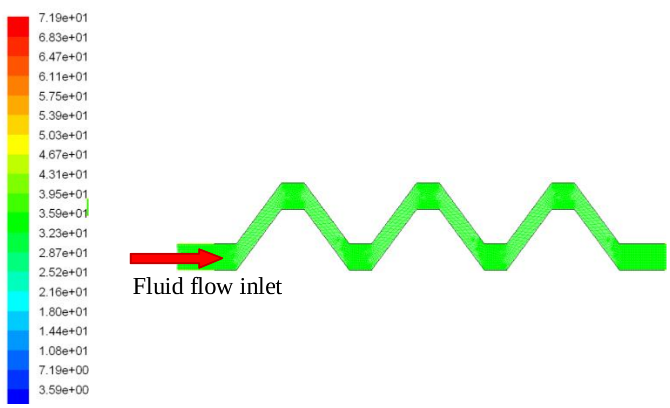
-Comparing separator overall behavior in two and three dimensional

In this study sustained particles made from pentane and with 50micron diameter ، have been considered.

As geometry high dimension leads to high mesh volume formation ، so separator analysis with real geometry contained high time consuming time that operationally is not possible. Therefore we are following real geometry simplification ، to perform problem numerical analysis. A distance length between blades in two and three dimensional has been modeled and studied. In figure 7 two -geometry mesh has been shown.



a) 3- dimensional geometry



b) 2- Dimensional geometry

Fig.7. plates geometry

Existence of blades in the flow path causes pressure drop through the reservoir length .So one of the parameters that can be considered as a test to determine simplified geometry results truth and accordance of them with main geometry results in pressure drop through a blade path length. As it is understood ، pressure drop through a separating length in 3 dimensional states is equal to 36.91Pa; that shows a little difference. It should be noted that low pressure drop through the reservoir length portend appropriate design. Other than what was described، blade setting against the fluid flow way cause to change in flow lines (such as flow jump and or rotation) and also to change in particles path that it is particles separating basis in this separating system. Low velocity locales formation and also boundary layer adjacent to the walls are specified in velocity contours as well (Blue color locales). Separator performance which is trapped sustained particles ratio to total throw particles ، in three and two dimensional forms regularly are 50% and 30% respectively. So results from main geometry and simplified two - dimensional geometry can be used due to acceptable accuracy and simplicity as main geometry indicator.

- Effect of blade tip geometry on separator performance

To determine the effect of blade tip geometry on separator behavior, 3 different blade tip types-round sharp, and flat (main geometry) – have been studied. In all these cases sustained particles in fluid flow are from pentane type with 50 micron diameter pressure contours in these geometry are shown in Figure 8.

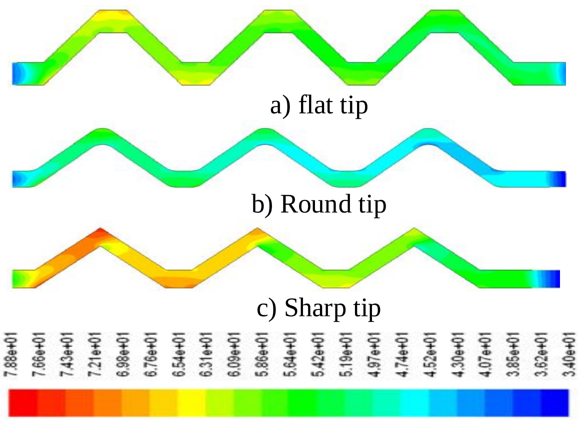


Fig.8. Pressure contours(Pa) : a) flat tip b) Round tip c) Sharp tip

As it is shown in Figure 8, which bigger blade tip angle ، more high pressure locales form and also much picture drop is done at the two ends of blades due to velocity decreasing in these locales. Another effect those blades tip geometry has on separator operation is change in flow velocity contours. Velocity contours and velocity vector have been shown in figures 9 and 10.

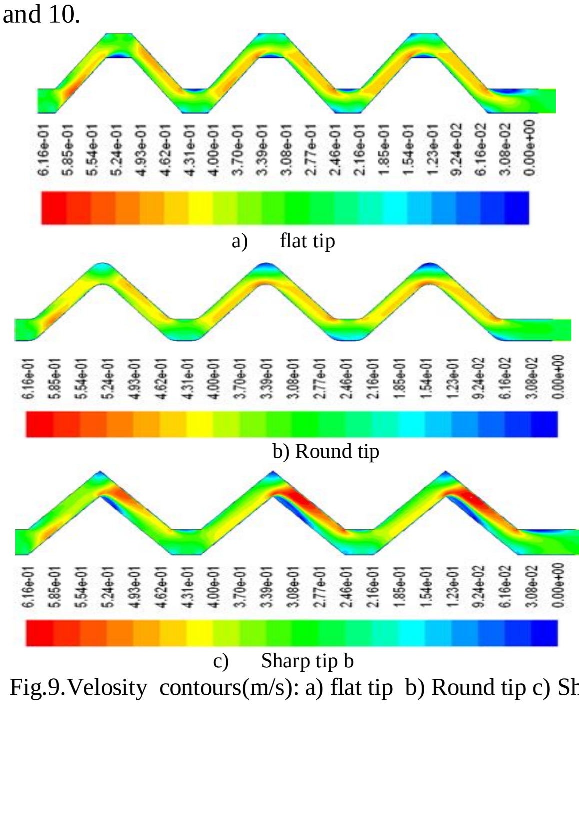
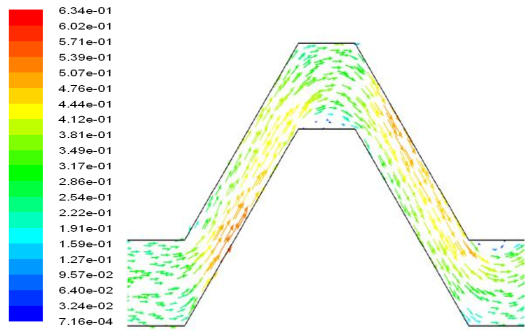
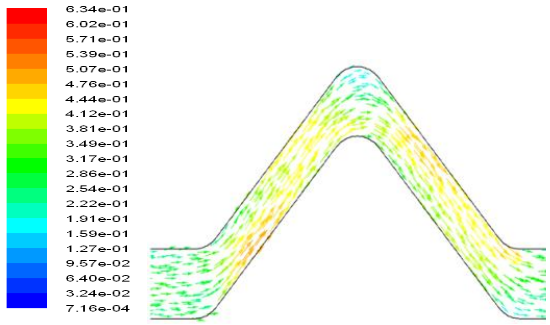


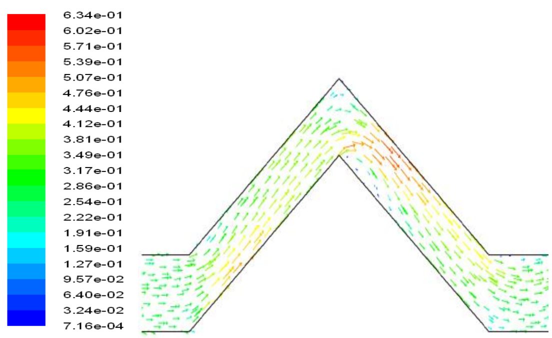
Fig.9.Velosity contours(m/s): a) flat tip b) Round tip c) Sharp tip



a) flat tip



b)Round tip blade



c)Sharp tip blade

Fig.10. Velocity vector(m/s): a) flat tip b) Round tip c) Sharp tip

In figure 10 produced rotations in fluid flow lines is obvious in corners .As it can be seen particles throw in sharp tip blade is more and there is much rotation in fluid flow. It is noted that whatever blade tip angle becomes less , particles take the shape of blade path ، better، as it is seen in sharp tip blade ، blade tip angle causes to particle throw toward partitions. Accordingly separator performance for blades with different tips is calculated table 4.Also in this table pressure drop in each geometry has been brought too.

Table 4: Result of simulation

Blade tip type	Efficiency%	Pressure drop (Pa)
Round	20	20.05
Flat	30	30.91
Sharp	22.5	37.64

Separator performance and pressure drop for blade with different tip for pentane sustained particles with 50 micron diameter is shown table 4.As for figure 4 it can be seen that flat tip (main geometry) has more performance and has lower pressure drop too in comparison other geometries.

-Study of effect of distance between blades on separator performances

Other important and effective parameter on separator performance and pressure drop is distance between blades. Whatever this distance becomes smaller ،droplet contact to partitions possibility increases and as a result they are trapped more and performance improves .Contrary decreasing the distance between blades also causes the pressure drop among the separator. Also optional distance selection is very important for designers. figure 11 is shown performance changes graph for different blades distances.

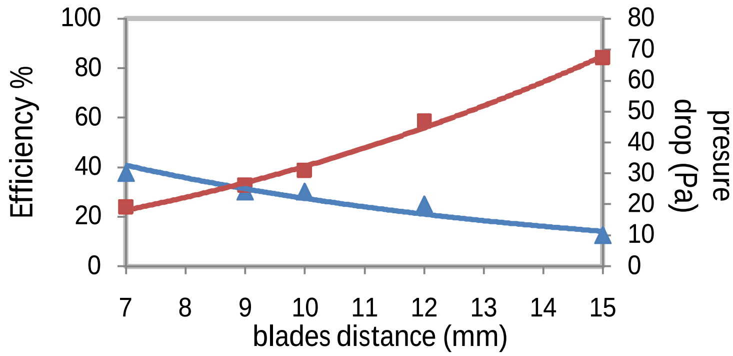


Fig.11. Separator pressure drop vs. performance according to the blades distance

As it can be seen from figure 11, with blades distance increasing separator performance decrease. The reason, as it is shown in last graphs is that shows the particles path ، Whatever the distance between blades becomes smaller, partitions performs much rule as a prevention against particles flow and so more particles get into the trap and separator performance increases .By the same way، with increasing blades distance ، Blade length increases that causes pressure drop in this path.

b) Velocity vectors in separator down corner a) Velocity vectors in separator up corner

Fig.12. internal rotation flow near the corners.

By notice that pentane was considered as discrete fluid ،outlet and inlet gas analysis results in B,C units is described in the table 5.

Table 5: Experimental data

Separator		i-C_5H_{12} Mol%	n-C_5H_{12} Mol%	Total pentane pentane	efficiency%
Unit B separator	inlet	0.33	0.26	0.59	40
	outlet	0.20	0.15	0.35	
Unit C separator	inlet	0.39	0.30	0.69	56
	outlet	0.17	0.12	0.29	

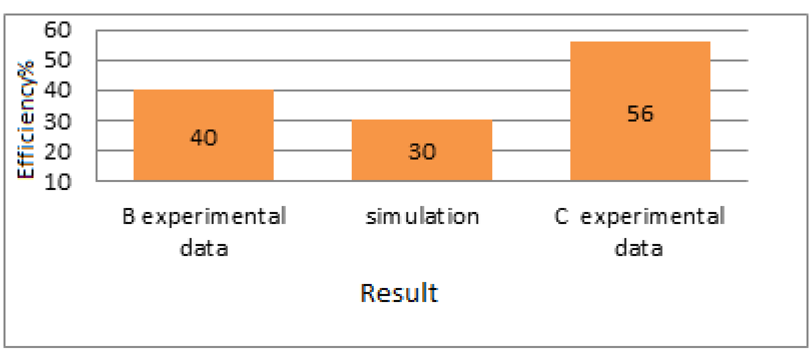


Fig.13. experimental data and simulation results

The difference between pentane gas outlet and inlet is the pentane values that has been separated in separating operations in B and C separators separates and are transferred as liquids from into phase1.In Figure 13 is shown that result of simulation and B train experimental data have 90% compatibility.

Conclusions

The result has shown that the RSM model has a good predication for the types of gas-liquid separating stream. The effect of circulation of flow at the corner of closed-by plates is the main reason for applying this model, Due to effect of pressure drop and growing of liquid droplet, the best geometry of wavy plate should be chosen. The result of this work has shown that the flat plate has a better efficiency duo to both of pressure drops and growing of liquid droplet. This is because of impacting of flow to wavy plate and the increasing of residence time of liquid in Flow that increasing the size of droplet.

NOTATION

U	Continuous fluid phase speed
Up	Discrete fluid particle speed
μ	Fluid molecular speed
P	Fluid density
P_p	Particle density
D_p	Particle diameter
Re	Reynold´s number
a_1, a_2, a_3	Constants
C_d	Drag factor
Φ	Shape factor
s	A sphere area with volume equal with a particle
S	Mass real area
$Resp_3$	Reynold´s number with spherical diameter Or equivalent volume
Cc	Gatingham corrective factor
λ	Molecular average free path
d	Size constant
n	Size distribution parameter

REFERENCES

[1] Ernest E. Ludwig, (1999) “Applied Process design for chemical and petrochemical plants”: -Volume 1. Third Edition-Emphasizes how to apply techniques of process design and interpret-Results into mechanical equipment details

[2] Christopher A. Chung (2003) “Simulation modeling handbook”: a practical approach /.

[3] William. L. Luyben (2004), “Process modeling, Simulation, and control for chemical engineers”: Process Modeling and Control Center-Department of Chemical Engineering Lehigh University.

[4] HAN Luchang, LIU Yuejin and LUO He’an), (2007) “Numerical Simulation of Gas Holdup Distribution in a Standard Rushton Stirred Tank Using Discrete Particle Method”*- - College of Chemical Engineering, Xiangtan University, Xiangtan 411105, China- Chin. J. Chem. Eng., 15(6) 808—813

[5] Ted Frankiewicz (SPE)/NATCO Group and Dr. Chang-Ming Lee/NATCO Group (2002). “Using Computational Fluid Dynamics (CFD) Simulation to Model Fluid Motion in Process Vessels on Fixed and Floating Platforms ”, SPE Annual Technical Conference and Exhibition held in San Antonio, Texas, 29 September–2 October.

[6] June (2006) ,“Computer Simulation of Multiphase Separators for Floating Platforms Helps Shrink Size by 50%”, journal article by fluent software users,

[7] Tital, Y. and Barena, D., (1997),inth ,“Simplified transient simulation of two- phase flow using quasi- equilibrium momentum balance”. Multi phase flow.

[8] Adelzadeh ،M.,.Two phases fluid flow principles ، first edit ، 1387 H.SH. ،

[9] Betonkar. Moghiman translation M, Numerical – computational calculation of fluids motion and heat transfer, 1377H.Sh. , first edition ، Ferdosi mashhad university publication

[10] Verestic ، malalaskera. shojayee fard translation , H, 1379H.SH. ، introduction to calculative fluid dynamic ،second edition ،Elm and sanat university publication

[11] due point setting unit separators in Sarkhoun and Qeshm gas refinery

[12] Fluent Soft Ware Manual. V. 6.3.26

[13] Gambit Soft Ware Manual. V. 2.1.6