Available online at www.scholarsresearchlibrary.com

Scholars Research Library

Der Pharmacia Lettre, 2017, 9 [6]: 163-175
[http://scholarsresearchlibrary.com/archive.html]

ISSN 0975-5071
USA CODEN: DPLEB4

Numerical Solution of Cyclone Separators to Optimize the Geometric Parameters and Increase the Efficiency

Reza Rostami Dehsarvi, Elham Ameri*

Department of Chemical Engineering, Shahreza Branch, Islamic Azad University, P.O. Box 311-86145, Shahreza, Iran

***Corresponding author:** Elham Ameri, Department of Chemical Engineering, Shahreza Branch, Islamic Azad University, P.O. Box 311-86145, shahreza, Iran, E-mail: shams2243@gmail.com*

ABSTRACT

The aim of this study is to design gas separator cyclones. Accordingly the design and simulation are performed by Gambit and Fluent software programs respectively. To achieve good conclusion the results are collected in terms of efficiency and compared with Lapple's analytical solution. As the results show there is a good approximation between the proposed Lapple's analytical results in this study. In this study several variables are examined one of which is the residence time variable in the cyclone. The results showed that the difference between heavy and light particles' residence time is only 2 or 3 seconds which indicates that the heavy particles play an important and determining role in the efficiency of cyclones because after the discharge of heavy particles the light particles' output path is a direct route that only takes a few minutes. The next variable is velocity magnitude. In this study the changes in two types of light and heavy particle are shown in a graph. Another variable is the position of particles per second. In this study, the position of light and heavy particles is shown x-direction of a graph. And in the end the total pressure in the cyclone is investigated. The results showed that in the middle of the cyclone the pressure is reduced due to the vortex motion and it is increased near the walls.

Keywords: Cyclone, Simulation, Numerical solution, Efficiency, Cyclone variables.

INTRODUCTION

Separation is of the most widely used processes in many industries. The purpose of separation is to remove contaminants, concentration of the solution and etc. [1]. Cyclones are one of the devices used to separate solid particles from the gas phase. Cyclone is a device that separates particles suspended in a fluid such as air, gas or liquid without the use of filtration. In this device the gas flow with solid particles enters a cylindrical or cone-shaped chamber. The principles of the separation of the particles from the gas flow is based on the fact that the inertia exerted on the solid particles in the cyclones is many times greater than the inertia applied on the gas phase; thus as a result of this force the heavy particles are led to the bottom of chamber on the cyclone's wall. In other words particles in the circulating or vortex flow has a lot of inertia and cannot follow the vortex flow. Therefore they hit the bottom of the cone-shaped cyclone and slip into the end of the cyclone where they can be collected. Cyclones can be classified from different points of view [2]. One of these cases is the cyclones' inlet. The inlet is usually rectangular or circular. In some cases, depending on the type of processing, the oval entries are also used. In the cone-shaped area the gas flow changes direction and goes upward and the outlet pipe. After contact with the wall of the cyclone the dust particles fall to the bottom of the cyclone and they are discharged by the outlet. The Figure 1 shows the geometry of a cyclone. The flow rate and the geometry of the cyclone determine the size of the particles that can be separated from the flow [3] (Figure 1).

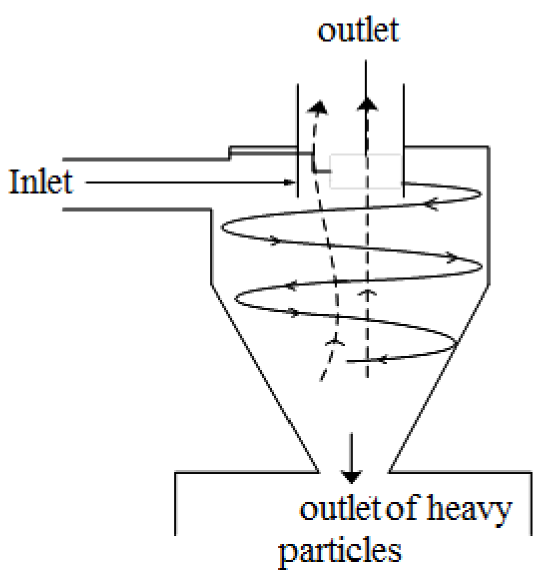


Figure 1: Cyclone schematics.

Review of the previous studies

Parvaz et al. analyzed the behavior and performance inside the cyclone and this flow is turbulence. They used the computational fluid dynamics to simulate the flow inside the cyclone. Finally, they compared the RNG and k-ε turbulence models and compared the result of both models with the experimental work. One of the advantages of their study was that by analyzing the RSM turbulence model they understood that the tangential velocity near the wall is more and the velocity function is more symmetrical and the results of this turbulence model have the same behavior of the experimental results [4].

Bagheri et al. studied the best methods for numerical simulation of gas fluid and solid particles' flow by cyclone modeling using computational fluid dynamics (CFD). They used Lagrange-Euler method to predict the behavior of solid particles and calculate particle efficiency and applied Navier-Stokes equations along with Reynolds stress model (RSM) to predict fluid flow. Results show that computational fluid dynamics can present the gas-solid phase behavior in the cyclone and used to predict the separation efficiency [5].

Jabaran et al. studied the effect of backflow on the performance of a dust collector system including cyclone and cylindrical container containing a thrower flow of the particles under experiment and CFD. The results show that increasing the backflow could have a significant impact on the separation efficiency of the system. The Eulerian - Lagrangian simulation perspective is used to simulate CFD. The dominant equations on the continuous phase are solved by finite volume method and the integration of the scattered fuzzy path equations is done by RungeKutta numerical method [2].

Haji Dolu et al. simulated two phase flow (oil and gas) in a two-phase gas and oil separator in National Iranian South Oil Company (Maroon 1) as three-dimensional. The aim of this study was to evaluate the impact of the cyclone input on the flow turbulence in the separator; thus simulations are done by simple and cyclonic deflector using the FLUENT software. The VOF1 multi-phase and e -k Realizable models are used for flow simulation and turbulence modeling. Eventually by drawing velocity profiles at different sections of the separator it is shown that using the cyclonic inlets instead of simple inlets it is possible to slow down the flow in the separator more rapidly or reduce turbulence in the separator inlet [6].

In 2013, Masihi et al. performed CFD simulation of cyclones with different diameters to separate the black powder and industrial wastes from natural gas. In this study the 3D geometry of gas - solid cyclone is plotted by Gambit software and the gas flow path along with solid particles is simulated inside the cyclone by FLUENT software. By analyzing the results of simulation, it became clear that the shape and geometry of the cyclone, the input gas flow rate and the size of solid particles have a significant impact on the separation efficiency and pressure drop in the cyclone [7].

Research theory

The centrifugal force that separates the particles from the circulating gas and pushes them towards the wall of the cyclone is a function of the gas velocity, particle mass and the distance between the center of rotation and particle.

$$F = \frac{m\omega^2}{r}$$

By applying this force where m presents the mass of the particles and r express radius of the cyclone, the particle with the velocity ω is faced with the opposing circulating gas flow, which is the same as the drag force. This force is subject to the particle velocity, particle diameter and carrier gas viscosity. Given the Stokes flow regime the drag force (DF_p) is obtained by below Equation where $d_p$resembles adiameter of the smallest particle v_t present sedimentation rate.

$$DFp = 3\pi\mu d_p V_t$$

At the inlet the centrifugal force is much more than the drag force because the particle initial radius velocity is zero. But as soon as this speed increases, (practically in a few hundredths of a second) the forces become equal and the particles head up the cyclone wall radially at a steady pace.

Where:

$$\frac{mw^2}{r} = 3\pi\mu d_p\omega$$

Since the mass of the particle is defined as follows:

$$m = \frac{\rho_p \pi d_p{}^3}{6}$$

It is notedρ_p indicated density of particle.

Assuming the spherical particles the centrifugal force is equal to:

$$F = \frac{\rho v_i{}^2 d_p{}^3}{r}$$

$$v_t = \frac{\rho d_p{}^2 v_i{}^2}{9\pi D}$$

Where ρ is equal with $\rho_{\text{paricles}} - \rho_{\text{air}}$, $v_i$tangential flow rate and D is chamber diameter.

The following equation is obtained by integrating the above relations [8]:

$$d_p = \left[\frac{9_\mu W}{\pi N v_i \rho_p - \rho_a}\right]^{1/2}$$

W: Entrance channel width

N: Number of effective rotation.

v_i: Tangential flow rate.

$(\rho_p - \rho_a)$: $\rho_{\text{paricles}} - \rho_{\text{air}}$

In fact, dp is the diameter of the smallest particle that is discharged through the output gas path at time t and reaches the cyclone wall and theoretically all particles larger than dp should be collected with 100% efficiency. According to the above equation the smallest particles collected with 100% efficiency is directly related to the gas viscosity and input channel width and is inversely associated with number of effective circulations, input gas velocity and density difference between particles and the gas. Of course this model is flawed because practically all particles larger than the intended diameter cannot be removed with an efficiency of 100% which is because all data are estimated.

Like the above equation, Lapple obtained the diameter of particle with an efficiency of 50% using quasi-experimental experiments [9].

$$d_p = \left[\frac{9_{\mu} W}{2\pi N v_i (\rho_p - \rho_a)} \right]^{1/2}$$

The only difference with the previous equation is the number 2 in the denominator.

In the Figure 2, the ratio of particle size of a standard cyclone to the diameter of the cylindrical part was shown. One of the features of the graph is that the efficiency reaches 100% asymmetrically which is against the previous theory that it reaches 100% with a steep slope to 100%. Although the Lapple's curve is based on experience, the numbers offered by some manufacturers present lower efficiency for dp to dp50 ratio than Lapple.

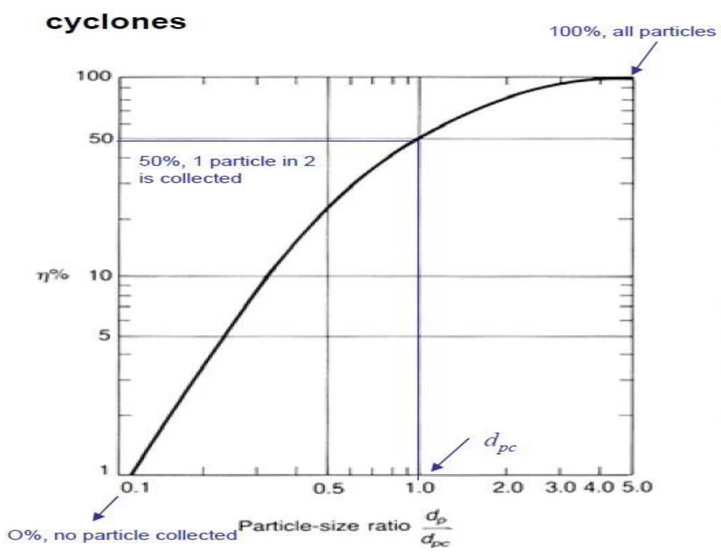


Figure 2: The ratio of standard cyclone particle size against the cylindrical section diameter [10].

Research method

In this study, the Fluent software was used for simulation. Figures 3 and 4 were displayed the dimensions and created cells.

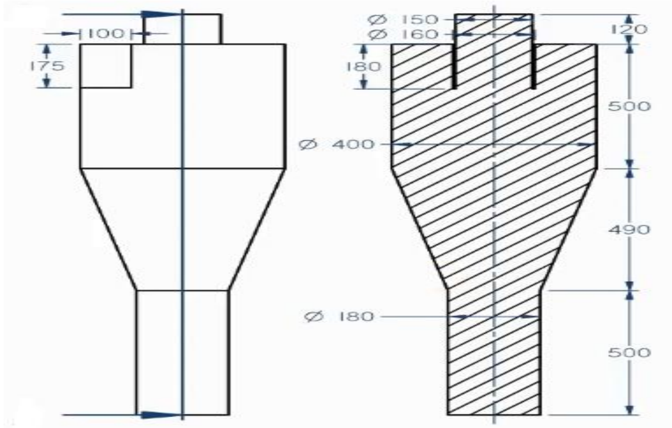


Figure 3: The primary simulation dimensions.

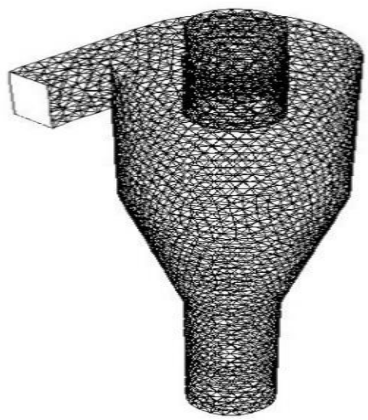


Figure 4: The definition of boundary conditions on the inlet wall.

For the discretization of the momentum and other equations by second order BOD POD is used. Also to create a high speed in calculations and due to the low number of equations (the absence of turbulence and energy equations) and their lack of complexity the SIMPLE coupled pressure and velocity solution is used. Since viscosity function in this problem is complex. The Reynolds Stress mode with 7 constants is used which is closer to reality than other models and also calculated the effect of the cyclone wall curvature on the fluid. The input velocity is 12.69 m/s. In the specification method, section the value of intensity and hydraulic diameter is 0.127 and the hydraulic diameter is obtained [11].

Also in the pressure outlet in specification method, the value of intensity and hydraulic diameter is 0.15. The hydraulic diameter is calculated based on the previous section with the difference that the section of the previous part is rectangle and it is circular here.

Table 1: Fluid simulation is performed by particles based on the table.

Fluid and particle specifications	
Specification	**Value**
Fluid velocity m/s	12.69
Particle velocity m/s	12.69
Particle density Kg/m^3	2770
Particle diameter µm	1, 1.2, 1.4, 2.3, 3.6, 4.9, 5.7, 6.7
Flowm^3/s	0.222

Since the particles are injected in the inlet along with the gas into the cyclone the surface of injection is set in the Injection section and defined for the input flow level. Particle distribution is defined by a uniform model.

The hydraulic diameters of the input and output wall are 0.127 and 0.15 and the particle diameter is 1 µm, 2 µm, 5 µm and 10 µm. In this problem the efficiency is calculated to several diameters according to the following equation and compared with Lapple experimental solution (1 µm, 2 µm, 5 µm and 10 µm). Also the particles' velocity is assumed as equal to the gas velocity of 12.69. Also the particle velocity is 2770 m/s.

RESULTS AND DISCUSSION

Calculation of efficiency in terms of particle diameter

In this section fluid simulation is conducted by particles based on the table in the last section.

The Figure 5 presents the particle residence time linearly. Each line represents the motion of a particle. As it can be observed the particles have the highest residence time in the cyclone inlet and as they move towards the outlet the lines get darker (less time). The Figure 6 also presents the vortex motion of particles in the cyclone. At the end of the clone due to the instability caused by the pressure difference, the light particles move towards the exit of the cyclone and the heavy particles are deposited towards the bottom outlet of the cyclone.

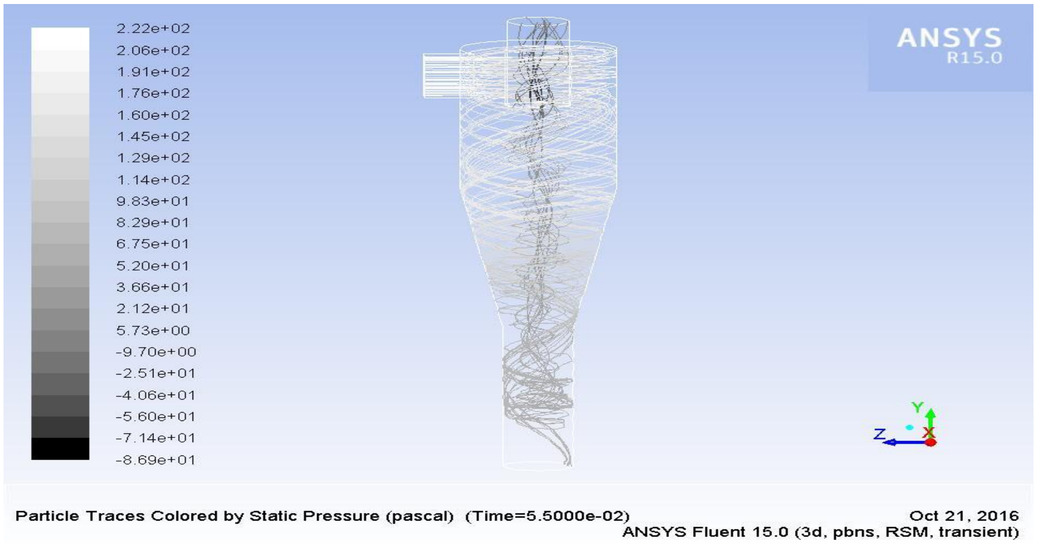


Figure 5: The path of the particles.

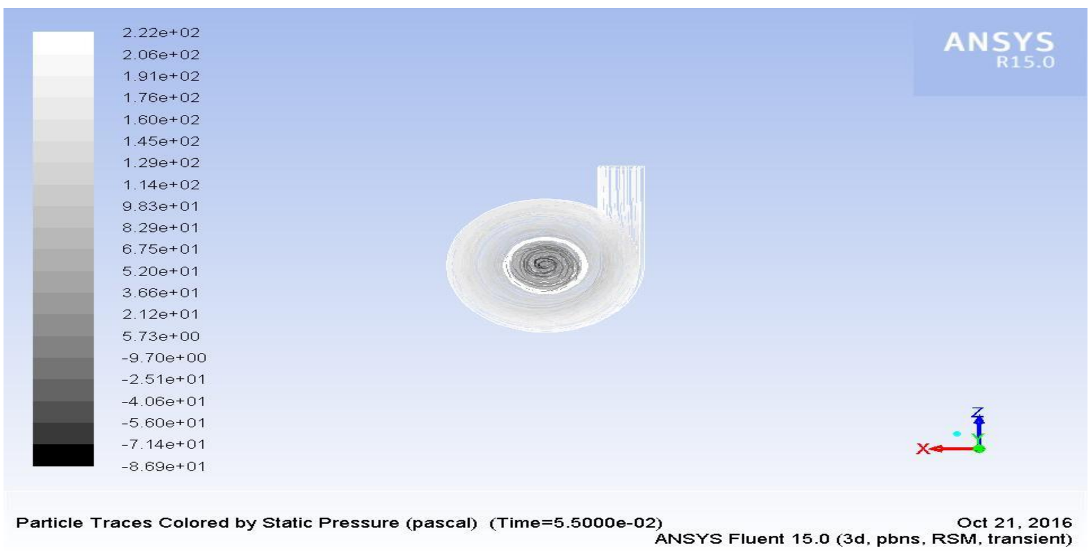


Figure 6: The movement of particles from above.

In the above figures, each of the lines represents one of the particles. As it can be observed some particles enter the device, some are released form the above and others are deposited at the bottom outlet. Based on the equation obtained in the last section:

$$\eta = \frac{\text{Trapped particles}}{\text{Incomplete particles-injected particles}}$$

Here, the above parameters are calculated by the settings described in the previous section for the diameter of 1 micrometer.

Calculation of the efficiency for other diameters will be similar to the above. It should be noted that in the simulation it is only required to change the diameters in the injection.

Verification

By comparing the results with Lappler esults (Figure 7).

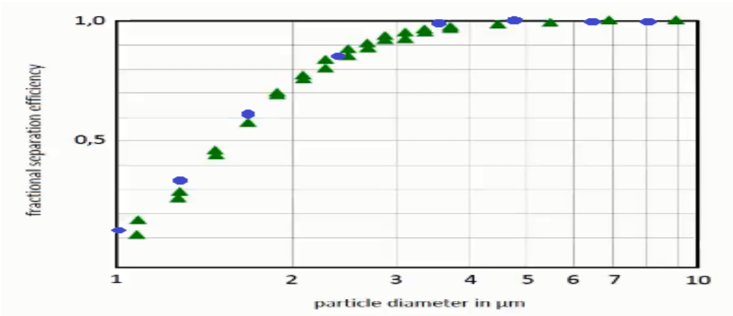
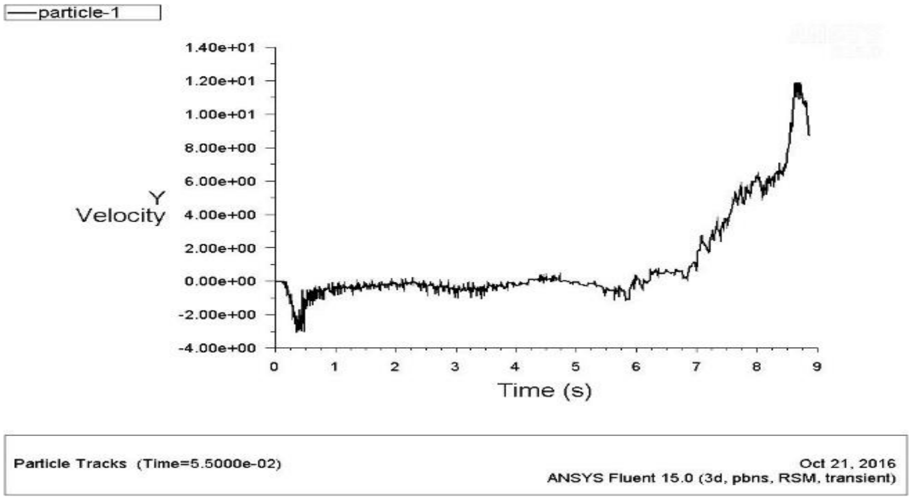


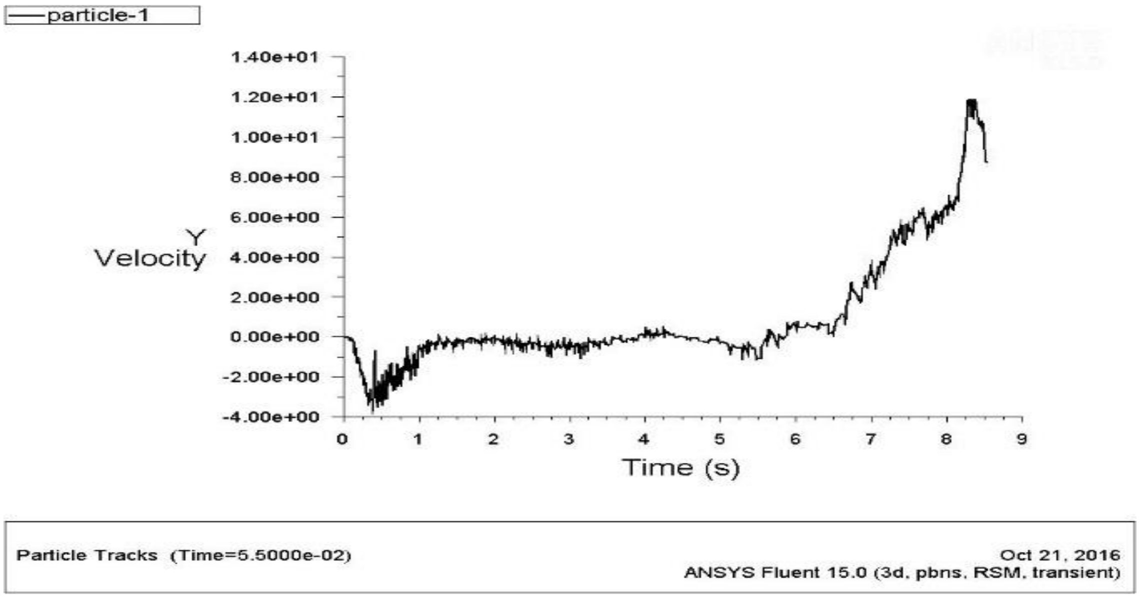
Figure 7: Comparison of simulation results with Lapple results.

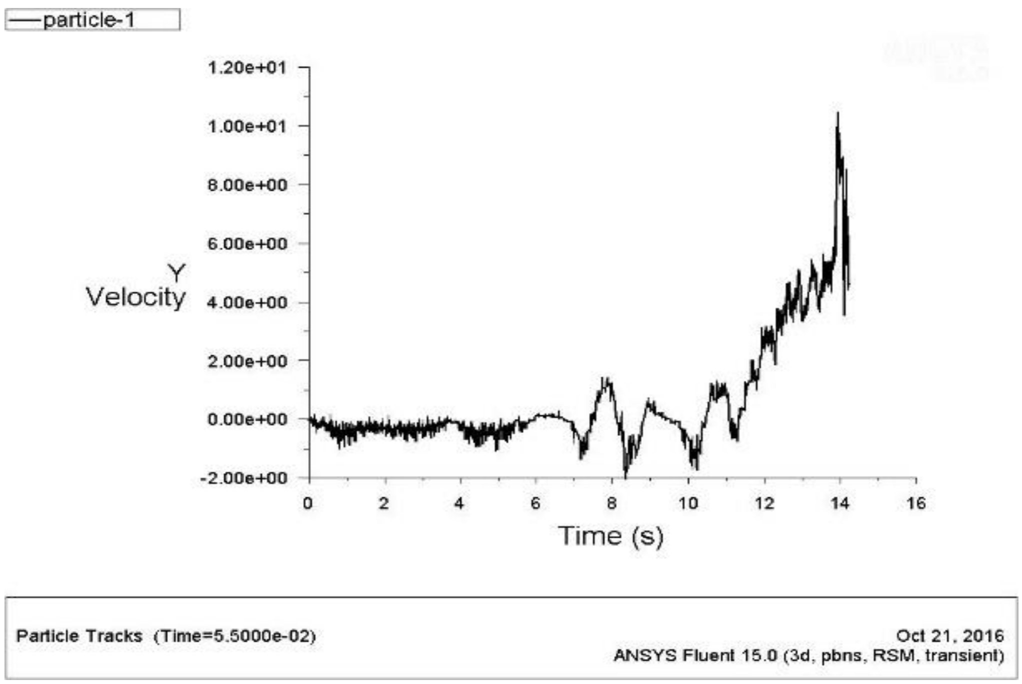
Triangles and circles present the experimental and simulation results respectively. As it can be observed, the results do match. Now that the accuracy of simulation is proved, the parameters of the cyclone are addressed.

Residence time

One of the important parameters is particle residence time. Using this variable, it is possible to study the deposition of heavy particles and the escape of light particles. Figures 8a -8f presents the particle residence time distribution.




particle-1
Y Velocity
1.40e+01
1.20e+01
1.00e+01
8.00e+00
6.00e+00
4.00e+00
2.00e+00
0.00e+00
-2.00e+00
-4.00e+00
0 1 2 3 4 5 6 7 8 9
Time (s)
Particle Tracks (Time=5.5000e-02)
Oct 21, 2016
ANSYS Fluent 15.0 (3d, pbns, RSM, transient)


particle-1
Y Velocity
1.20e+01
1.00e+01
8.00e+00
6.00e+00
4.00e+00
2.00e+00
0.00e+00
-2.00e+00
0 2 4 6 8 10 12 14 16
Time (s)
Particle Tracks (Time=5.5000e-02)
Oct 21, 2016
ANSYS Fluent 15.0 (3d, pbns, RSM, transient)

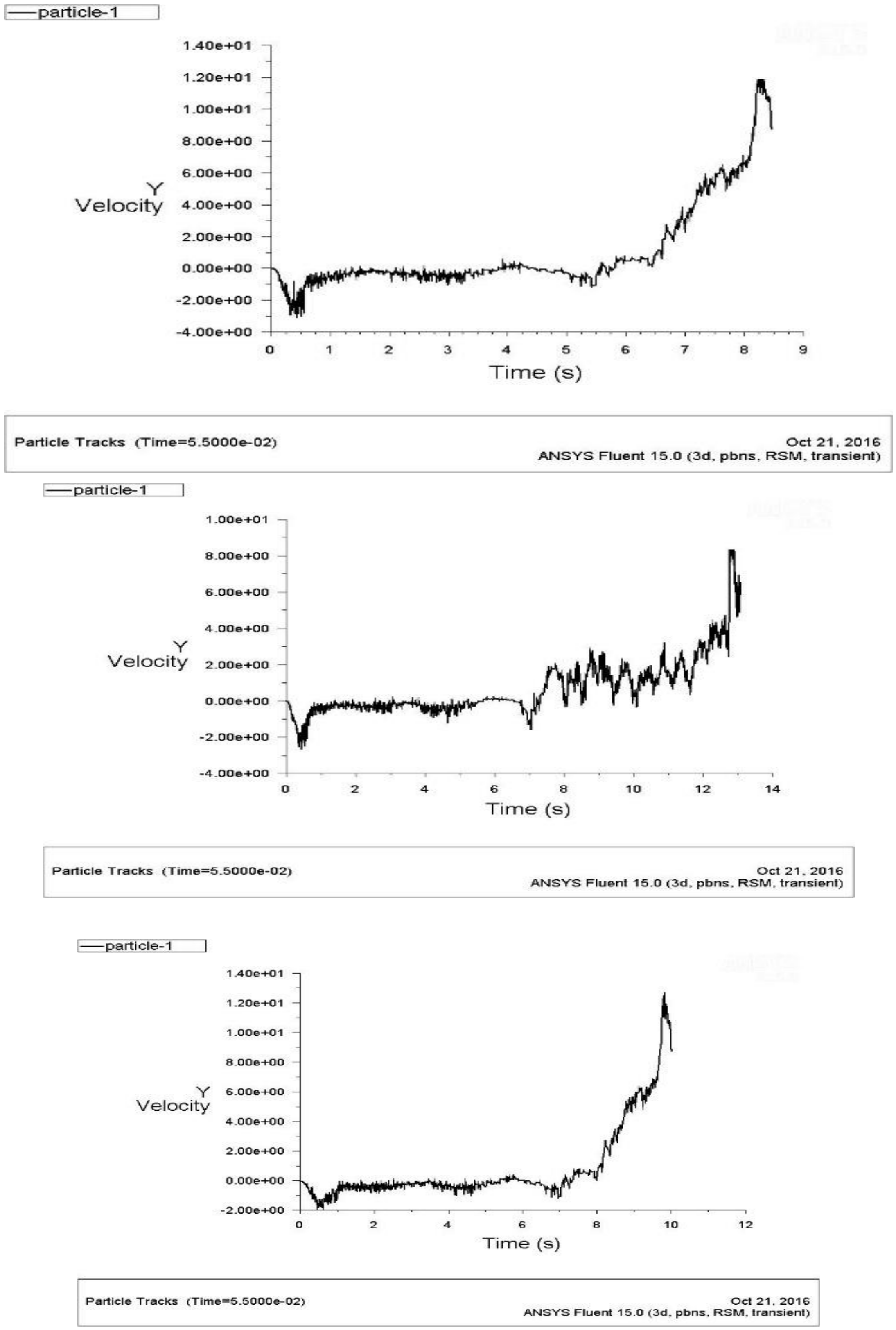


Figure 8: Residence time distribution for 6 cyclone particles.

The deposited particles have less residence time that the particles discharge from the top of the cyclone, which is indicated in the Figure 9. As the particle residence time is lower, the cyclone performance will be better. Residence time is directly associated with the input fluid velocity and device geometry. If the input velocity is high and the size is small, there will be a turbulent flow in the cyclone inlet which will cause the cyclone performance to face difficulty. Therefore setting these two parameters is one of the important design issues. According to the Figures 9 and 10 it can be observed that the parameters are set correctly and there is no turbulence. Therefore, the cyclones have a high performance at the height to cylinder diameter ratio of 4, the height to cone diameter ratio of 9 and the velocity of 12.69.

Now the velocity is analyzed for the deposited and light particles.

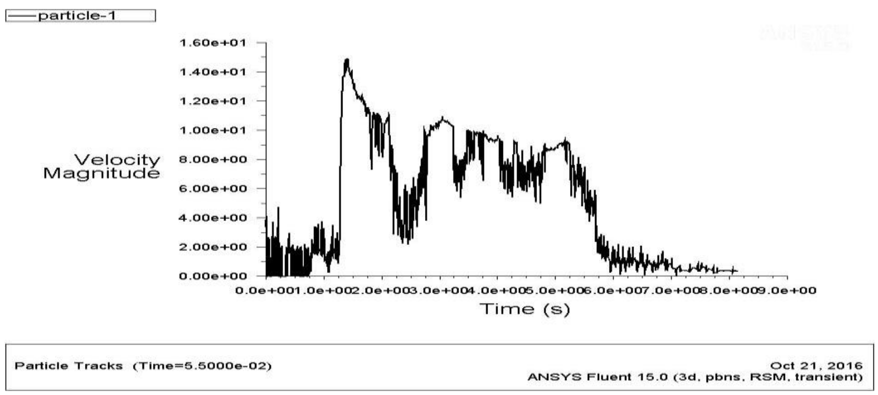


Figure 9: Velocity magnitude distribution for the deposited particle.

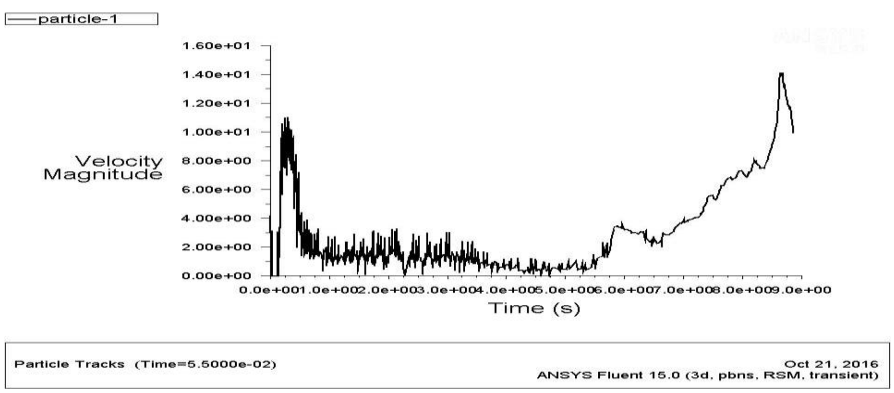


Figure 10: Velocity magnitude distribution for the volatile particle.

According to the above Figures, the deposited particles' velocity magnitude is zero after a while but the light particles have an increasing velocity.

Particles' position

The Figure 11 presents the particles' position in y direction. The deposited particles reach zero position after a period but the volatile particles have a parabolic graph (Figure 12). As a result, the particles position presents the type of particle. In fact the following graphs indicate that the light particles have a residence time more than 10 seconds in the cyclone while the heavy particles have a residence time less than 10 seconds in the devise. The difference between the two times based on the graph is about 3 sec. The 3-second difference is between heavy and light particles inside the devise. As a result, the time is also an important parameter in the design so that as the time difference is greater, the device will have better performance because it indicates that heavy particles will deposit sooner.

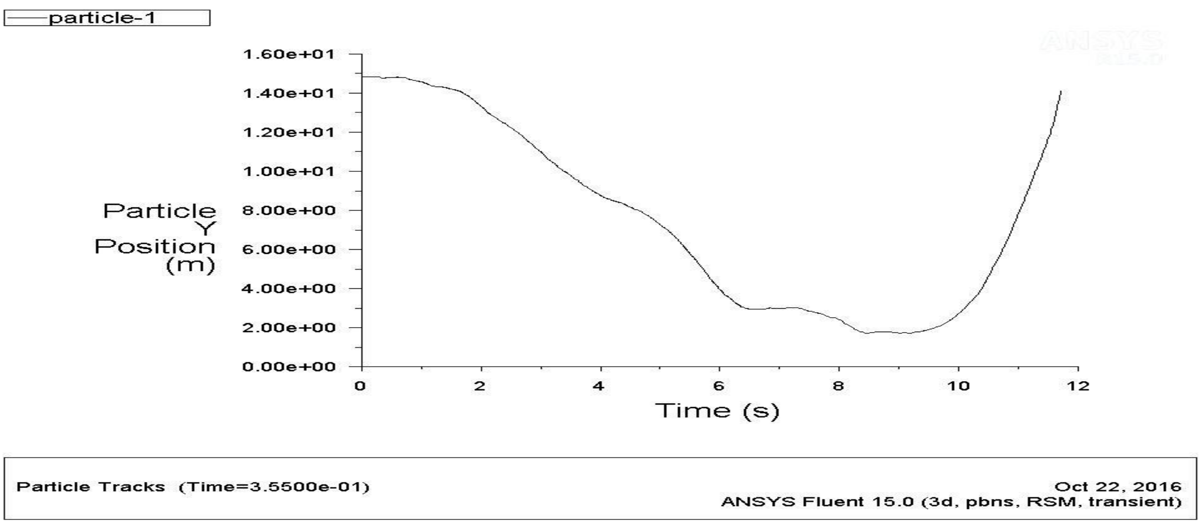


Figure 11: Volatile particle position in y direction.

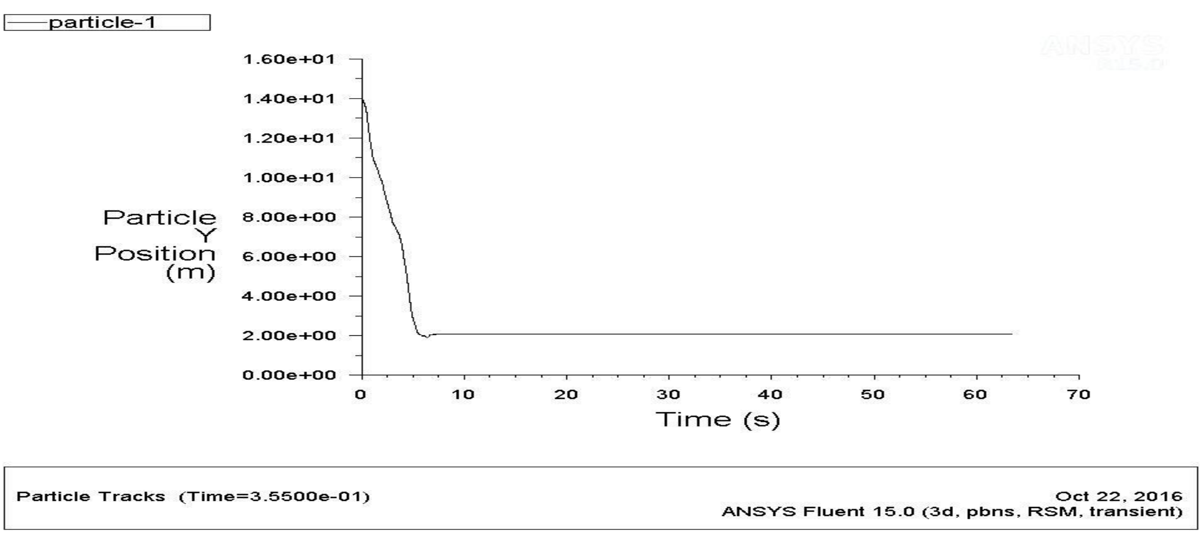


Figure 12: Deposited particle position in y direction.

Pressure distribution

Figures 13 and 14 shows the total pressure variations in the cyclones. Since the flow created in the cyclone will be in the form of vortex, it is expected that the pressure is reduced at the center and the pressure is increased by increasing the distance. As it can be observed by approaching the center of the cyclone, the pressure was decreased, the pressure was increased by getting away from it, and there was the highest pressure on the walls of the cyclone.

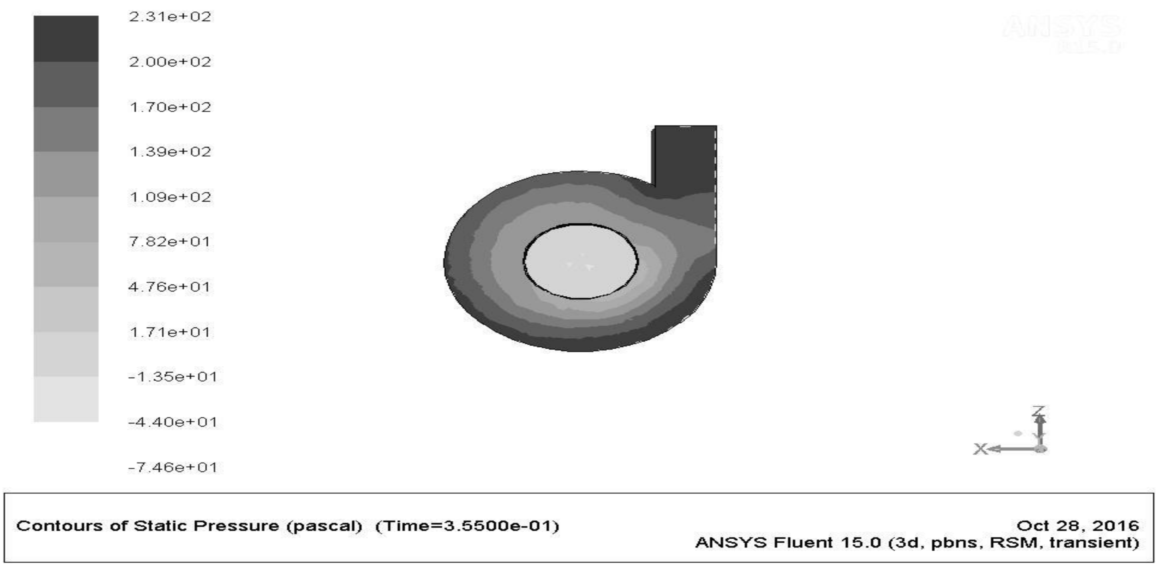


Figure 13: Velocity magnitude distribution for the volatile particle.

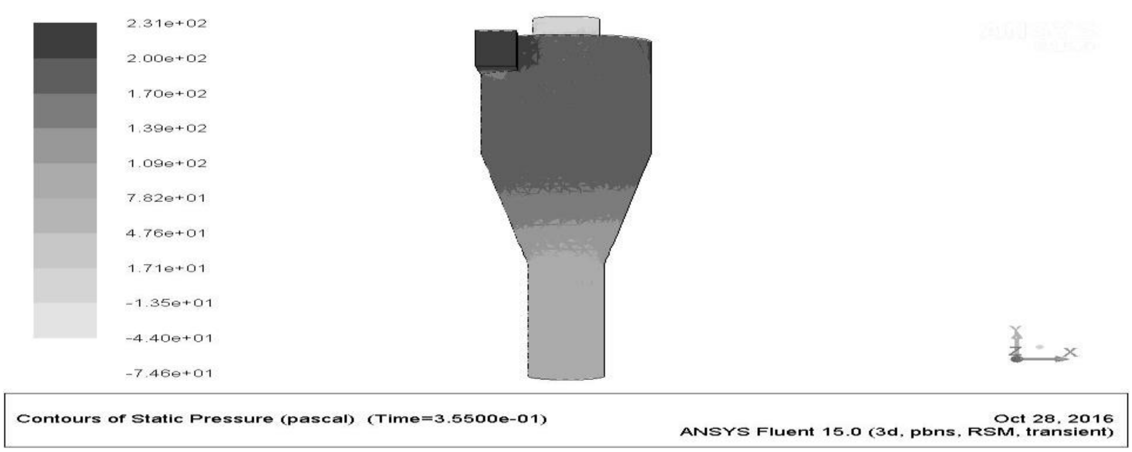


Figure 14: The total pressure distribution.

CONCLUSION

As the results, show there is a very good approximation between the proposed simulation and Lapple's analytical results. As a result, these geometric dimensions are appropriate to design and build cyclones and the specifications, features and type flow in the cyclone can be analyzed by the simulation and be assured about the accuracy of the analysis. Such studies are cost free and save time. Simulation analysis could help to develop knowledge and information about the cyclones. In this study, several important variables in the cyclone were examined. The first time variable was the residence time. This variable is one of the most effective variables in the design of the device. Using the results obtained in this study the residence time for each particle be calculated and observed. The results showed that the difference between heavy and light particles is only 2 or 3 seconds which indicates that that heavy particles play a determining role in the efficiency of cyclones because after the discharge of heavy particles the light particles' output path is a direct route that only takes a few minutes. The next variable was velocity. Using simulation the changes in velocity magnitude can be observed for all particles. In this study, the changes the two types of light and heavy particle were shown in a graph. Another variable was the position of particles per second. Using simulation the position of particles per second can be observed and calculated for all particles in all directions. In this study, the position of light and heavy particles is shown x-direction of a graph and in the end the total pressure in the cyclone was investigated. The results showed that in the middle of the cyclone, the pressure is reduced due to the vortex motion and it is increased near the walls.

REFERENCES

[1] Seader, JD., Ernest, J., Henley, D., Roper K. Principles of separation processes, **2013**. (3rd edn): 1.

[2] Khazaei, I., Moghiman, M., Numerical investigation of cyclone chamber diameter and segregation input on separation of particles in a cyclone separator. Eighteenth Annual International Conference on Mechanical Engineering, Iran, **2010.**

[3] Parvaz, F., Raghei, R., Compare the performance of different models to predict the turbulence in the gas flow within the airocyclones. Third national conference on oil and gas and related industries, Kerman, martyr Bahonar University, **2015.**

[4] Bagheri, N., Moradi, S., cyclone simulation by computational fluid dynamics. Process engineering Scientific Conference, Tehran, alchemy Energy Seminar, **2014.**

[5] Jabaran, A., Shirvani, M., Experimental and CFD backflow effect on the performance of a new dust collection system. The National Conference of Chemical Engineering, Branch, Islamic Azad University, **2009.**

[6] Delkhah, N., Haji, DolloE., NoghrehAbad, A.R., Simulation of fluid flow in horizontal separator of oil and gas phases. The first national conference on new technologies in the chemical and petrochemical, Tehran, enterprise seminar cycle of Science and Technology, **2014.**

[7] Masihi, M., Sharif, K., Abad H., Rahbar N. Study and numerical simulation of gas-solid cyclone to inhibition of pollution and waste and industrial waste in natural gas pipelines by using CFD, Fourth National Conference wastewater and solid waste management in the oil and energy industries, Tehran, Enterprise Seminar Alchemy of Energy, **2013.**

[8] Shepherd C.B., Lapple C.E. Flow pattern and pressure drop in cyclone dust collectors. Industrial and Engineering Chemistry. **1939.** 31(8): 972–984.

[9] Shepherd, C.B., Lapple, C.E., Flow pattern and pressure drop in cyclone dust collectors. Industrial and Engineering Chemistry **1940.** 32 (9): 1246–1248.

[10] Kegg, A., Numerical investigation of gas cyclone separation efficiency with comparison to experimental data and presentation of a computer-based cyclone design methodology.". LINK Electronic Thesis and Dissertations Center, Electronic thesis or dissertation. University of Akron, **2008.**

[11] Rahimi, R., Rahimi, MR.. overview of computational fluid dynamics in chemical engineering, Zahedan, Sistan and Baluchestan University, Department of Chemical Engineering, National Iranian Chemical Engineering Congress, **2004.**