Investigation of Separator Cyclone Performance Using CFD Simulation: A Case Study of Cement Industry Cyclone

M. Khaki Jameia[*1], M. Moradi Beshelia[2], Morteza Abbasi[3]

Abstract: This paper tends to investigate the performance of a separator cyclone used in cement industries in Iran. This investigation was performed using CFD modeling, which is validated by other valid studies. Separator cyclone performance includes of two important parameters as pressure drop and collection ratio. In this presented study, the pressure drop has been considered to assess the performance of the cyclone. This assessment follows by changing geometrical parameters of the cyclone as its outlet diameter and height. Beside these geometrical parameters, two fluid parameters are investigated as the inlet velocity and outlet pressure. At the end 4 specified models and the CFD model of main cyclone are compared by considering fluid flow parameters, circulation, and finally cyclones pressure drop. This paper suggests the best parameters to be changed which make the cyclone more efficient. Results show that from investigating parameters, increasing the inlet velocity to 25 m/s has the most effective impacts on cyclone efficiency.

1. Introduction

The geometrical form of separator cyclones changes translational kinetic energy of fluid into circular and centrifugal flow, which leads to separation of solid particles from the fluid flow. There are different kinds of industries, use separator cyclones because of cyclones simplicity of construction and wide range of operational conditions [1]. Cyclones construction simplicity leads to low investment and maintenance costs. During 20th century, separator cyclones are widely used as the major part of particle elimination from gas flow, in different kinds of air pollution control systems [2]. In many powdered production process of valuable materials, separator cyclones are preferred [3]. In operating conditions with high temperature or pressure, high particle materials load, and high corrosive components in fluid flow, the use of separator cyclones has many advantages [4, 5]. In fact, separator cyclones are simple controllers which use centrifugal forces to remove

*. Corresponding Author: Department of Mechanical Engineering, Sari Branch, Islamic Azad University, Sari, Iran khaki@iausari.ac.ir

particle materials in the fluid flow [2]. The circular geometry of separator cyclones, forces the fluid in a spinning motion and makes high vortex flow which the flow vortex leads in centrifugal forces [2]. These centrifugal forces have magnitudes depending on particles' sizes and their mass [6]. This effect leads in higher forces acting on solid particles inside the fluid flow because of their higher mass than gaseous particles which drift them toward the cyclone wall and fall them to the cyclone lower end. Several studies dealing with the analysis of the geometric [7, 8] and operational [9, 10] effects on the performance variables of cyclones are reported in the literature. In addition, devices have been proposed to improve the performance using the same geometry and operational conditions [11, 12]. In general, these studies have focused on the effect of changes in a single parameter on the cyclone performance [1].

There are two major parameters in separator cyclones design: collection efficiency and pressure drop. Two kinds of approaches could be used to predict the cyclone performance as single-phase and double-phase approaches.

In the single-phase approach the flow is modeled without considering the particles in the flow. Using this approach, the fluid flow is modeled more accurate and pressure drop, velocity, and vorticity fields are involved to optimize the cyclone fluid flow. Pressure drop in separator cyclones leads in energy loss and efforts in reducing this parameter could cause energy saving even if it leads in reducing collection efficiency [13]. The cyclone separator pressure drop depends on the cyclone geometry and the operating conditions [14].

Double-phase approach includes the methods of discrete phase models (DPM) which considers a discrete phase of particles in the fluid flow. In this approach the collection efficiency of particles is the major goal of study. By the DPM double-phase approach the modeling accuracy of fluid flow would be decreased. In the other hand collection efficiency and particles movement will be obtained.

The considered cyclone is using in cement crinkle process and need to be optimized. In order to decrease optimization costs and to increase the insurance of optimizing, numerical simulations were used. In recent researches it has declared that Ansys Fluent modeling could have acceptable agreement comparing to experimental data obtained from different cyclones [06, 08]. It shows that Ansys Fluent have the capability to simulate flow in cyclones. In this study, Ansys Fluent 17 software package was used to model the fluid flow and fluid-solid flow in a cyclone.

The properties of fluid flow compared using different models with different dimensions of the end cross section of considered cyclone separator. Optimum size and form of the lower level cross section is proposed in the pressure drop, velocity, and vorticity fields point of view.

2. BACKGROUND RESEARCHES

Cyclones have numerous applications in science studies and industries and flow in the cyclones are interested in the world. Using studies about flow in cyclones, this kind of systems are recently more optimized. Nowadays many of researchers focus on cyclones and its flow realization and optimization. Recent studies show that CFD

modeling of flow in cyclones could confirm experimental observations.

Sedrez et al. [1] in 2017 prepared an article about their experimental study of erosion and CFD modeling for fluid-solid flow in a cyclone. The cyclone they considered operates with Fluid Catalytic Cracking particles as a solid phase and air at room temperature as fluid phase. In this research experimental studies show that erosion increases by increasing the inlet gas velocity and this increase of erosion tends to maximum in 30 and 35 m/s of gas inlet velocity. They used two models to simulate erosion by three steps of flow modeling, particle tracking, and erosion calculation. The results of CFD models have acceptable agreement with experimental observations.

Sedrez et al. [1] have used Euler-Euler method for modeling fluid-solid particles flow while in another research in 2017, Balestrin et al. [2] have used Euler-Lagrangian approach for it. They investigated the effects of a reduction in the cross section of the vortex finder outlet duct together with a stretched cylindrical body on the flow pattern and performance of a conventional cyclone. These two researches have used Reynolds stress turbulence model. Balestrin et al. [2] have provided a new design of cyclone with higher collection efficiency than the cyclone they studied. They concluded that the fine particles means particles with a diameter of lower than 5 μm increases the collection efficiency while the cyclone geometry could affect it.

Safikhani [13] in 2016, has claimed that the efficiency of cyclone separators could be increased by increasing the vortex length in the cyclone. He investigated the performance

optimization of new cyclones using Computational Fluid Dynamics, Artificial Neural Networks, and Non-Dominated Sorting Genetic Algorithm. In this research these three models are combined and used to increase the collection efficiency of new cyclones and to decrease the pressure drop in them. The design variables of this research like many researches about cyclones were the geometry parameters of cyclone.

In the way to optimize and to increase the cyclones efficiency, Wasilewski [15] in 2017 analyses the effect of counter-cone position in a cyclone using experimental and numerical studies. In this research 15 different geometries are investigated. The investigated modifications involved the installation of a counter-cone with different geometrical parameters inside the cyclone these were proposed based on an analysis of the results of other studies on the subject. The CFD analysis in this research was performed using Ansys Fluent software. The analysis concerned the value of pressure drop and separation efficiency. The location of the counter-cone was found to be the key geometrical parameter. The results show that the counter-cone should be located above the lower outlet of the cyclone. Obviously the application of a counter-cone in each of the studied geometrical variants led to an increase in solid particle separation.

Wasilewski and Brar [16] in another research in 2017, has studied about the geometry optimization of cyclone separators used in clinker burning process. In this research he used Ansys Fluent software again to simulate the fluid-solid flow in the cyclone and used Reynolds stress model to involve the turbulence effects. This study considers

three cyclones with different diameters and estimates the pressure drop of them using CFD method. At the end the optimum diameter has introduced.

In 2016, Haig et al. [17] focused on small scale cyclones used in vacuum cleaners. They used CFD to simulate the performance of small cyclones and numerical results have compared to empirical observations. This study has clearly demonstrated the capability of CFD to accurately predict cyclone performance indicators of pressure drop and separation efficiency.

3. METHODS

In this research, methods are selected using previous related researches which contain studies about different kinds of separator cyclones. Using Ansys Fluent 17, pressure based simulations have been performed which uses the finite volume method of computational fluid dynamics approach. In different simulations, RANS turbulent model, named $k - \varepsilon$ is applied which is described in 4.1.4 part of this paper in more details. The fluid considered as continuous air and as an ideal gas.

In this research the fluid flow and the pressure drop caused by the flow is concerned. To obtain research goals, the inlet velocity, outlet pressure, outlet diameter, and cyclone height were changed and modeled.

4. MATHEMATICAL MODEL

This model has two parts, continues fluid flow and discrete phase. In fluid mechanics two methods are specified to calculate flow parameters, Eulerian and Lagrangian approaches. Most of researchers and CFD softwares have used Eulerian method to calculate the phase of fluid parameters [18]. But for prediction of fluid flow with a discrete phase, there are two approaches as Euler-Euler and Euler-Lagrange. Most of studies about cyclone separators have used Euler-Lagrange approach for discrete phase model [1, 2, 13, 15, 16, 17, and 19]. In this study Euler-Lagrange approach has used to model the fluid flow in cyclone and its discrete phase.

Among various techniques, computational fluid dynamics (CFD) is a widely accepted and promising tool to model the complex flows prevailing inside cyclone separators. Several studies have demonstrated the potential of CFD to predict the functionality and description of flow inside cyclones and to optimize performance parameters [16].

4.1 The governing equations of fluid flow

The CFD solvers use transport equations of motion to predict the flow parameters. These transport equations are Mass, Momentum, and Energy equations (sometimes state equation joins them). Then these equations should be discreet to be solved numerically.

4.1.1 The Mass Conservation Equation

The law of mass conservation is a general statement of kinematic nature, i.e. independent of the nature of the fluid or of the forces acting on it. There is no diffusive flux for the mass transport, which means that mass can only be transported through convection. With the convective flux, the general differential form of mass conservation equation following [18]:

$$\tag{1}$$

4.1.2 The Momentum Equation
The momentum conservation equation, expressed by Hirsch follows [18]:

$$\frac{\partial \rho \vec{v}}{\partial t} + \vec{\nabla}.\left(\rho \vec{v} \otimes \vec{v} + p\overline{\overline{I}} - \overline{\overline{\tau}}\right) = \rho \vec{f_e} \qquad (2)$$

Where ρ denotes the fluid density, $\vec{v}$ is the fluid velocity vector, p is pressure, $\overline{\overline{I}}$ is the unit tensor, $\overline{\overline{\tau}}$ is the viscous shear stress tensor, and $\rho \vec{f_e}$ is the external volume forces per unit volume.

An equivalent non-conservative form is obtained after subtracting from the left hand side the continuity equation multiplied by $\vec{v}$:

$$\rho \frac{\partial \vec{v}}{\partial t} + \rho\left(\vec{v}.\vec{\nabla}\right)\vec{v} = -\vec{\nabla}p + \vec{\nabla}.\overline{\overline{\tau}} + \rho\vec{f_e} \qquad (3)$$

Where $\overline{\overline{\tau}}$ equals to:

$$\tau_{ij} = \mu[\left(\frac{\partial v_j}{\partial x_i} + \frac{\partial v_i}{\partial x_j}\right) - \frac{2}{3}(\vec{\nabla}.\vec{v})\delta_{ij} \qquad (4)$$

4.1.3 The Energy Equation
In compressible flows the energy equation should be involved and considered because of energy variation that occurs through the flow. The differential form of the energy conservation equation has provided by Hirsch [18] as follows:

$$(5)$$

in which E indicates the total energy per unit mass, T is the absolute temperature, k is the thermal conductivity coefficient, $\overline{\overline{\sigma}}.\vec{v}$ is the surface sources which are the result of the work done on the fluid by the internal shear stresses acting on the surface of the volume, W_f is the work of external volume forces, and q_H denotes the heat sources other than conduction.

In cyclone flows external heat sources q_H and external volume forces W_f are equal to zero. By clarifying the term $\vec{\nabla}.\left(\overline{\overline{\sigma}}.\vec{v}\right)$ and introducing the enthalpy of the fluid H, the governed energy equation in cyclone flow becomes:

$$H = (E + p) \qquad (6)$$

$$(7)$$

4.1.4 Turbulence modeling
One of the most applied models for considering turbulence in flows is Reynolds Averaged Navier-Stokes equations (RANS). The most widely applied approximation for industrial applications of CFD is the approximation whereby the turbulent equations are averaged out, in time, over the whole spectrum of turbulent fluctuations [18].

There are some sub-models for RANS model and the most applied model in cyclone flow simulation is the two-equation model named $K - \varepsilon$ model [1, 2, 13, 15, 16, 17, and 19].

In this paper the $K - \varepsilon$ model was selected as the model basis for turbulent modeling. This model contains of two equations, one for turbulent kinetic energy k and the other for turbulent kinetic energy dissipation ε as follow [13]:

$$\frac{\partial}{\partial t}R_{ij} + \overline{u_k}\frac{\partial}{\partial x_k}R_{ij} = \frac{\partial}{\partial x_k}\left(\frac{\vartheta_t}{\sigma^k}\frac{\partial}{\partial x_k}R_{ij}\right) - \left[R_{ik}\frac{\partial\overline{u_j}}{\partial x_k} + R_{jk}\frac{\partial\overline{u_i}}{\partial x_k}\right] - C_1\frac{\varepsilon}{K}\left[R_{ij} - \frac{2}{3}\delta_{ij}K\right] - C_2\left[P_{ij} - \frac{2}{3}\delta_{ij}P\right]$$
$$- \frac{2}{3}\delta_{ij}\varepsilon \tag{8}$$

in which P_{ij} equals to:

$$P_{ij} = -\left[R_{ik}\frac{\partial\overline{u_j}}{\partial x_k} + R_{jk}\frac{\partial\overline{u_i}}{\partial x_k}\right] \tag{9}$$

$$\tag{10}$$

Where $R_{ij} = \overline{u_i'u_j'}$ is the Reynolds stress tensor and $u_i' = u_i - \overline{u_i}$ is the *i*th fluctuating velocity component. Additionally $\overline{u_i}$ is the mean velocity, x_i is the position, $\overline{P}$ is the mean pressure, P is the fluctuating kinetic energy production, ϑ_t is the turbulent viscosity and $\sigma^k = 1$, $C_1 = 1.8$, and $C_2 = 0.6$ are empirical constants [20].

4.2 The Governing Equations on Particle Motion

As previously stated the Euler-Lagrange method is the best approach to model fluid flow with discrete phase which uses Eulerian formulation to simulate the fluid flow and Lagrangian formulation to model Particle track motion. Then motion of small particles including the effect of nonlinear drag and gravitational forces is given by [13]:

$$\tag{11}$$

$$\tag{12}$$

Here, u_i^p is the velocity of the particle and x_i is its position, d is the particle diameter, S is the ratio of particle density to fluid density, and g_i is the acceleration of gravity. In this equation C_D is the drag force coefficient of particle which is given as [13 and 16]:

$$C_D = \frac{24}{Re_p} \qquad for \quad Re_p < 1 \tag{13}$$

$$C_D = \frac{24}{Re_p}\left(1 + \frac{1}{6}Re_p^{\frac{2}{3}}\right) \qquad for\ 1 < Re_p < \tag{14}$$
$$400$$

where Re_p is the particle Reynolds number defined as [13 and 16]:

$$Re_p = \frac{|u_j - u_j^p|d}{\vartheta} \tag{15}$$

5. Geometry and Grid Independency

A simple cyclone separator used in cement industry has been considered. In Figure.1 a schematic image of considered cyclone have shown and in Figure.2a x-y section of cyclone drawn in Solidworks software is provided. Parameters defined to introduce cyclone dimensions have been shown in Figure.1 and related values of these parameters are listed in Table.1.
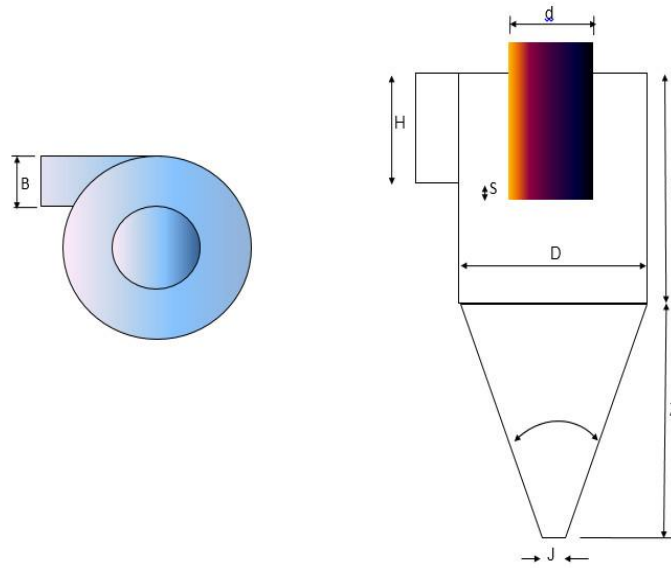
Figure 1 A schematic view of considered cyclone and its dimensions parameters

Table 1.
Dimensions length of cyclone

Parameters	Length (mm)
D	5206
d	2741
H	2468
L	2968
Z	7202
J	2750
B	1301

Figure.2b shows the grid mesh of cyclone in the view of x-y cross section plane. The cyclone meshed using 1 mm size of cells with quadrilateral shape. Quality checking shows that the skewness of generated mesh is lower than 0.46 which shows the acceptable accuracy of cells shape. The aspect ratio of generated mesh is about 1 to 4 and therefore acceptable.
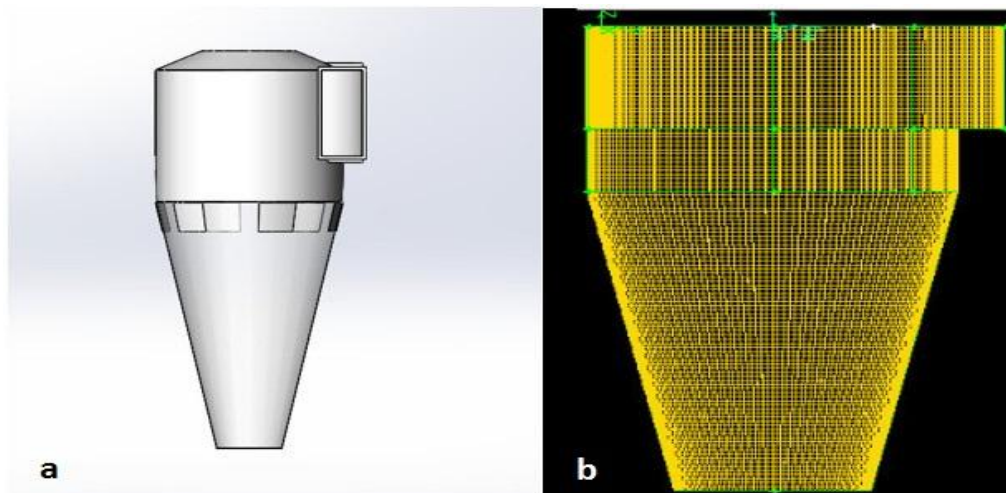
Figure 1 a) Cross section of x-y plane in Solidworks software b) the grid mesh of cyclone

But because of important effects of grid, checking quality parameters like skewness and aspect ratio couldn't be enough. It should be proved that CFD results are independent of grid's size and quality. Two kinds of errors generating in CFD simulations are truncation and round-off error, which by decreasing the grid cells size, truncation error will decrease but round-off error will increase [Anderson]. This implies that an optimum grid size should be selected, not too great and not too small. In order of checking the results independency of grid size, four different grid sizes are generated. In table.2 the grids information has been provided which contains of cell numbers and cell size range.

In Figure.3 axial velocity curves of the first 3 grid sizes on a line through the cyclone diameter and y axis, in the distance of 0.25 m lower than cyclone head (output) have shown. Grid number 4 had unusual results and because of that the first 3 grids are provided in one figure and variations of 4 grids have shown in Figure.4 which shows the non-acceptable errors.

Table 2.
Grids information for testing results independency of grid

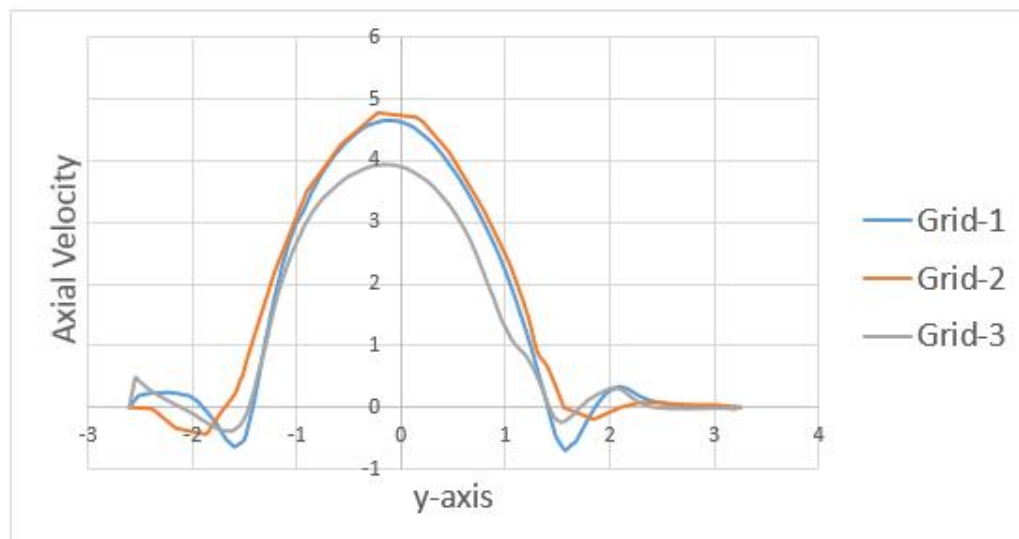| Grid Number	1	2	3	4
Elements Number	781740	31143	1839860	2228534
Nodes Number	312299	15021	717580	846147
Skewness Average	0.17951	0.20216	0.1787	0.18234
Skewness Standard Deviation	0.11479	0.11972	0.1138	0.11422
Aspect Ratio Average	1.7627	1.8755	1.7639	1.7495
Aspect Ratio Standard Deviation	0.48514	0.58797	0.4916	0.48044

Figure 2 Axial velocity in the grid 1, 2, and 3 through cyclone diameter of y-axis in 0.25 m lower than cyclone outlet
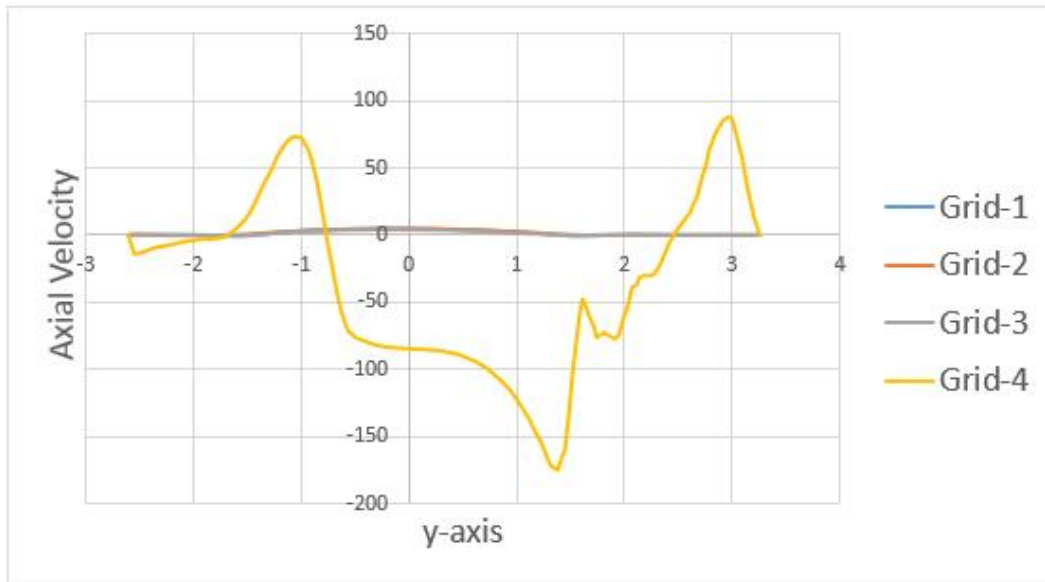
Figure 3 Increase of error in Grid-4

From Figure.3 and 4, and using comparing by experimental and CFD base studies and considering inlet velocity of inlet gas is about 12 m/s, it could be concluded that Grid-3 has the best results [13 and 21].

6.Validation

It's time to compare this research results to other experimental and CFD base researches about cyclone separator. As mentioned in this research $k - \varepsilon$ model is used as turbulence model and fluid has a velocity of 12 m/s at inlet of cyclone. In Figure.5 the results of axial velocity in Hamdy et al. [21] are presented, in which the cyclone diameter is 290 mm. Figure.6 shows the results of axial velocity obtained in this research for 1 m lower than cyclone outlet.
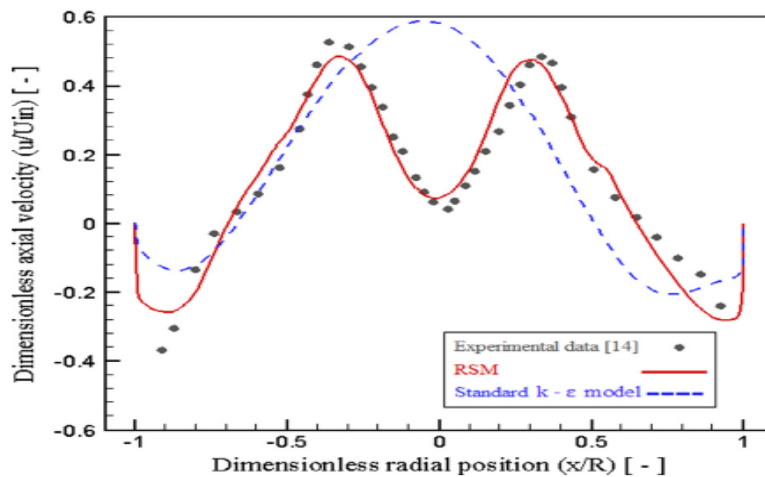
Figure 4 Axial velocity presented by Hamdy et al. [21]

It could be clear by comparing Hamdy et al. [21] results by standard $k - \varepsilon$ model to Figure.6, using this model of turbulence, axial velocity pattern is obtained correctly and its magnitude is so close. It should be mentioned that the cyclone modeled in Hamdy et al. [21] is so smaller than this research cyclone. Their cyclone diameter was 290 mm and diameter of this research's cyclone is 3808 mm (Table.1).

In Figure .7 the contour of static pressure in x-z plane is presented which follows by Figure.8 that shows the static pressure in x-z plane obtained in this research.
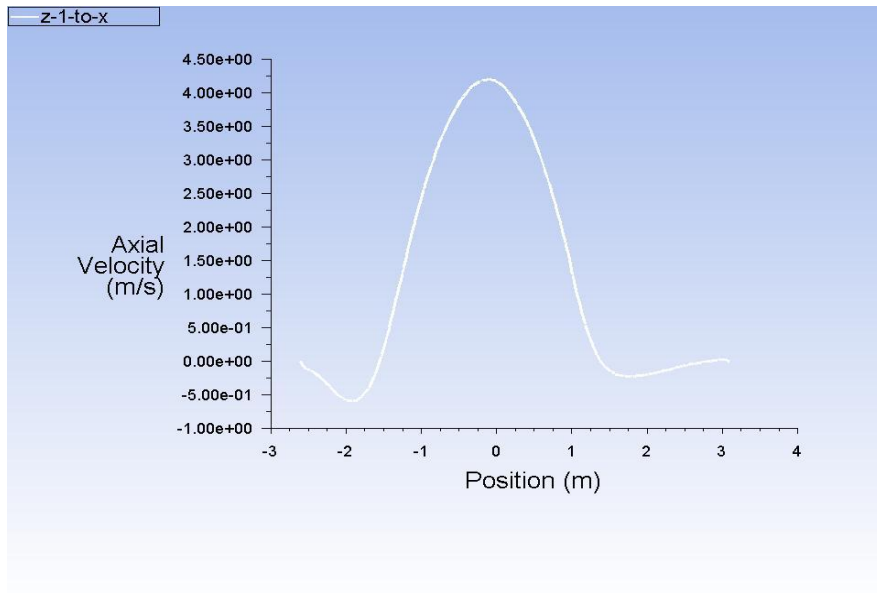
Figure 5 Axial velocity obtained in this research for 1 m lower than cyclone outlet
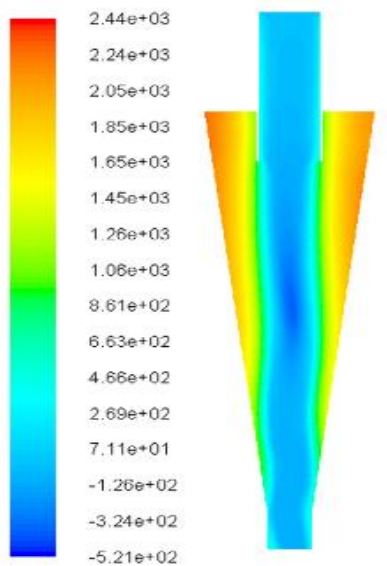
Figure 6 Static pressure in a x-z plane presented in Hamdy et al. [21]
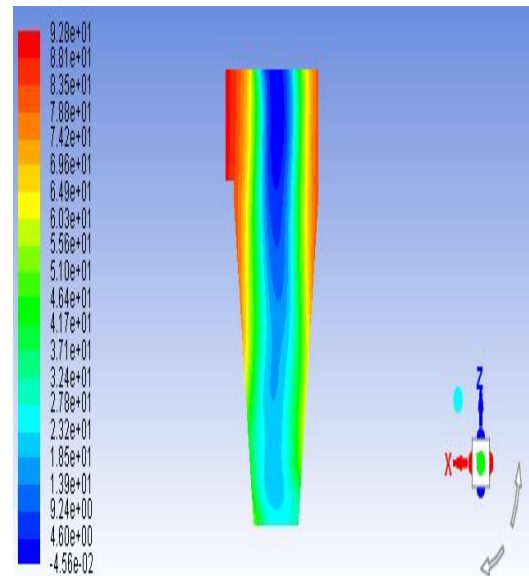
Figure 7 Static pressure in a x-z plane from results obtained in this research

7.Results

This research aimed to optimize the cyclone separator and to realize fluid flow behavior inside of it. To obtain these major objectives, some changes considered and modeled to determine the effects of cyclone geometry variations on its efficiency and pressure drop. Results show that this cyclone could be more efficient as it is. At first step the results of fluid flow in the main cyclone geometry will be provided.

7.1 The main cyclone model

In Figure.9 contour of static pressure in x-z plane of cyclone has shown. It follows by Figure.10 which shows the contour of static pressure in x-y plane of cyclone at top section of cyclone. In this model there is a 200 Pa pressure drop between inlet and outlet of cyclone. While the fluid flows in to the cyclone where it has 18 m/s velocity. The main cyclone geometry is listed in Table.1.

The pressure drop, obtained from this modeling was about 185 Pa. In this simulation the RSM model have used to model the turbulent impacts of flow.
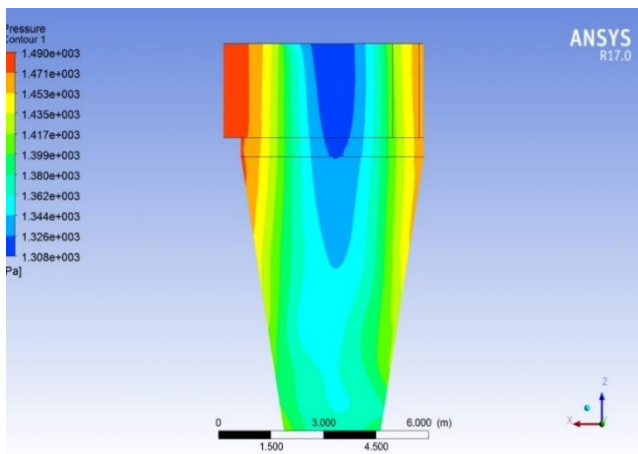

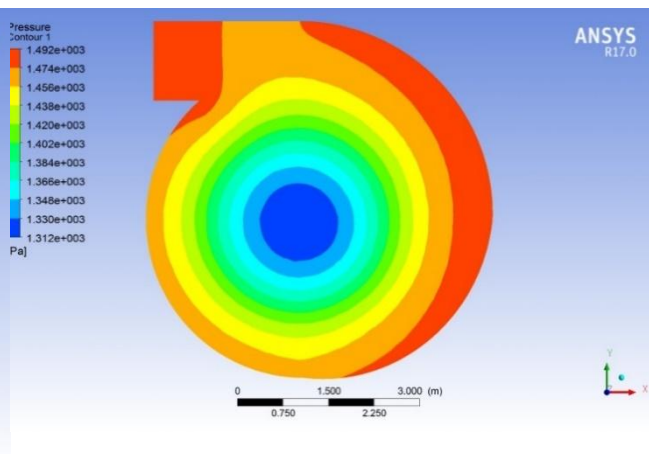*Figure 8 The static pressure distribution at x-z plane in y=0*

Figure 9 The static pressure distribution at x-y plain in z=-1 m

To investigate the circulation flow in the cyclone in Figure.11 the stream lines is shown in a 3D view. Like above this figure is followed by Figure.12 which shows the stream lines in the x-y plane of cyclone in a section where is 1 m lower than the upper face of cyclone. Figure.11 demonstrates that the flow couldn't reach the lower face of cyclone easily and the flow is concentrated at upper half of cyclone, means in a distance about 2 m above the lower section of cyclone.

While in Figure.12 it's obvious that the flow is distributed equally in the x-y section of cyclone. But remember that the section shown in Figure.12 is included in upper half of the cyclone which has more flow concentration. The low concentration in lower half of the main cyclone is a negative character in its design and could be corrected and optimized by a little more consideration.

Figure.13 shows the velocity vectors of fluid flow in the outlet section of main

cyclone. These vectors could prove the circulation generated in the cyclone outlet

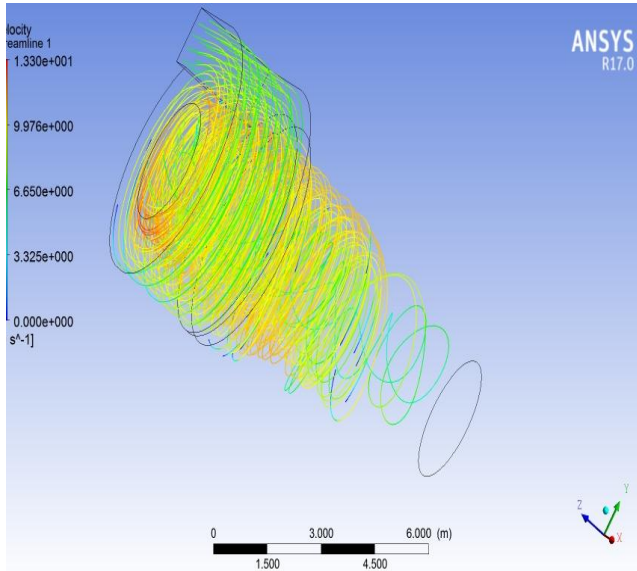which follows the circulation created inside the cyclone.

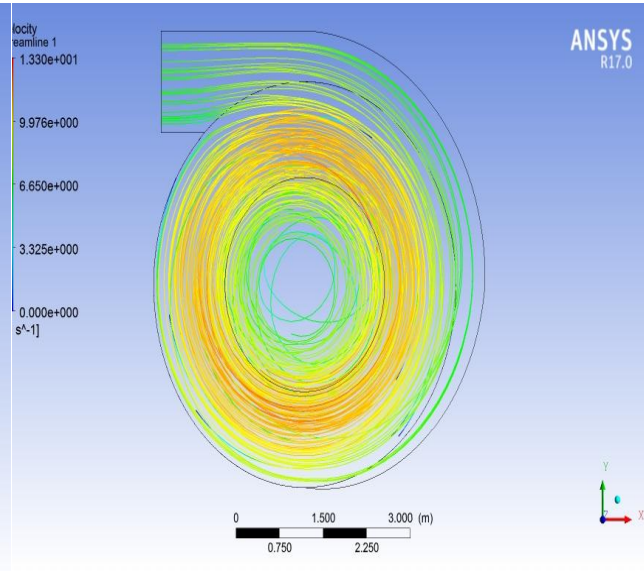***Figure 10 Flow stream lines in a 3D view of cyclone***

Figure 11 Flow stream lines at x-y plane in 1 m lower than upper face of cyclone
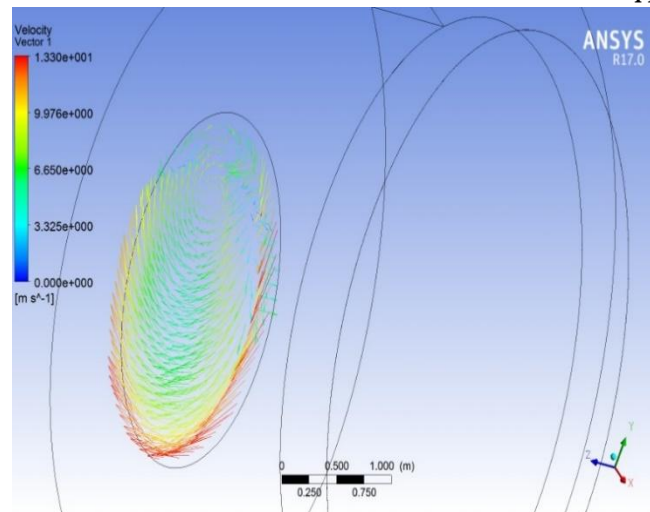

Figure 12 The velocity vectors in the cyclone outlet plain

In Figure.15 the axial velocity distribution in radial diameters along y axis in $z = -0.1$, -0.25, -0.5, -1, and -3 m. it's obvious from Figures.14 that the axial velocity distributions in the upper half of cyclone are similar, but in height of $z=-3$ m it starts to be different.

The tangential velocity distribution of radial diameter of cyclone in $z=-0.25$, -0.5, -1, and -3 m is shown in Figure.15. This figure, the same as Figure.14 shows that in -3 m height, the tangential velocity distribution as axial velocity starts to change.
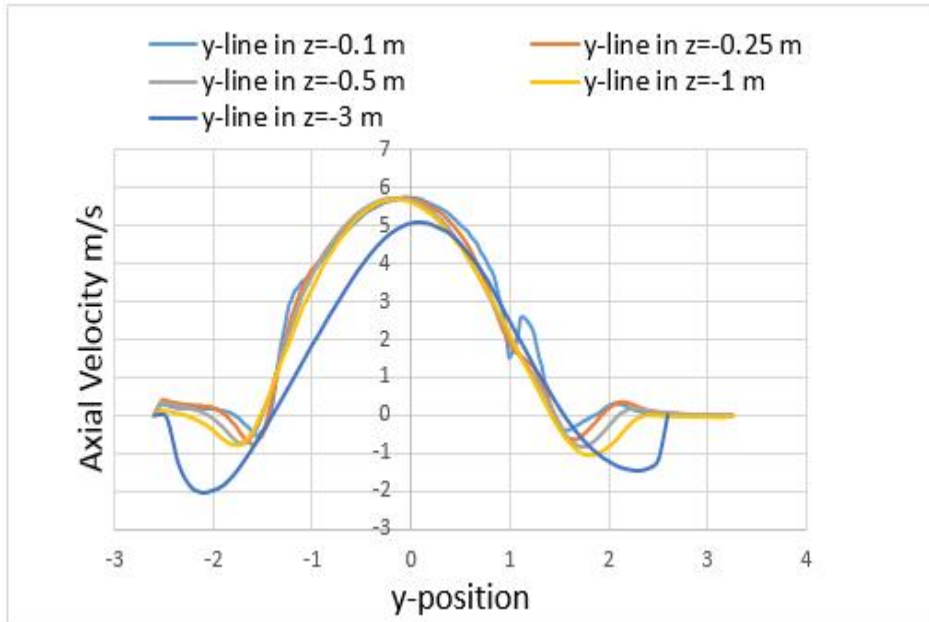
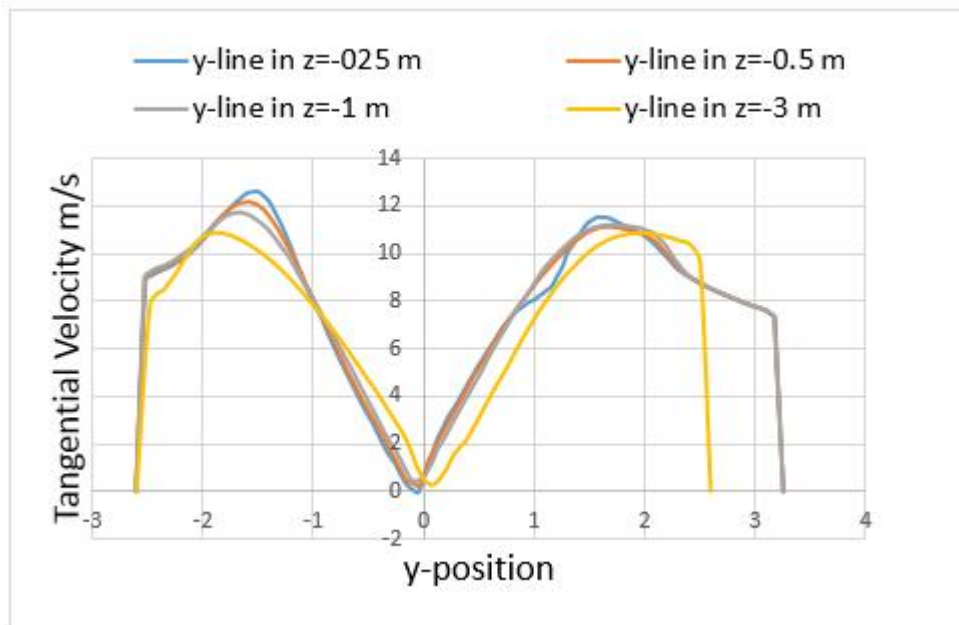*Figure 13 The axial velocity distribution in 5 different heights of cyclone*

Figure 14 The tangential velocity distribution in 4 different heights of cyclone

7.2 The Cyclone Optimization Models

To test the cyclone optimization 4 changes are performed in the cyclone geometry and flow parameters which were modelled using Ansys Fluent 17. These changes include of 2 geometrical and 2 fluid flow parametrical. Geometrical changes contain of a greater outlet and a taller cyclone. In the greater outlet model the outlet face diameter was changed from 2741 to 4000 mm and in taller cyclone model the L height parameter shown in Figure.1 is increased from 2968 to 3978 mm and the total height from 10.170 to 11.18 m. Flow parametrical changes include of

increase of fluid velocity at inlet face from 18 to 25 m/s, and its pressure decrease at inlet from 1510 to zero Pa.

In Figure.16 the stream lines obtained from all simulations has shown which contains of a: higher inlet velocity (25m/s), b: taller cyclone (11.18 m), c: lower outlet pressure (zero Pa), d: greater outlet (4 m), and e: the main cyclone model. There are 50 lines of streams in all 5 images of Figure.16.

Figure.16 demonstrates that in the main model (16e) the fluid flow concentrates are higher in higher heights of cyclone. This fact implies to the model with greater outlet face (16d). But in models with higher inlet velocity (16a), taller cyclone (16b), and lower pressure in outlet (16c) obviously the fluid stream concentration in lower heights of cyclone has increased.

Figure.17 contains of static pressure in z=-0.25 for 5 different models as a: higher inlet velocity (25m/s), b: taller cyclone (11.18 m),

c: lower outlet pressure (zero Pa), d: greater outlet (4 m), and e: the main cyclone model. These charts in Figure.17 implies that the lower pressure drop is obtained by increasing the fluid flow velocity at inlet to 25 m/s. What could be realized from Figure.17 is the fact that in the model with taller cyclone (17b), the higher pressure drop occurs around the center of cyclone means in lower area of cyclone cross section. But in the model with outlet pressure of zero (17c) this area of maximum pressure drop increases. This increase is followed by the model with greater outlet in which the area of maximum pressure drop has covered highest area.

Another fact that should be mentioned about charts of Figure.17 is that the charts related to models with higher inlet velocity and the main model of cyclone have a similar pressure drop and their charts are placed on each other.
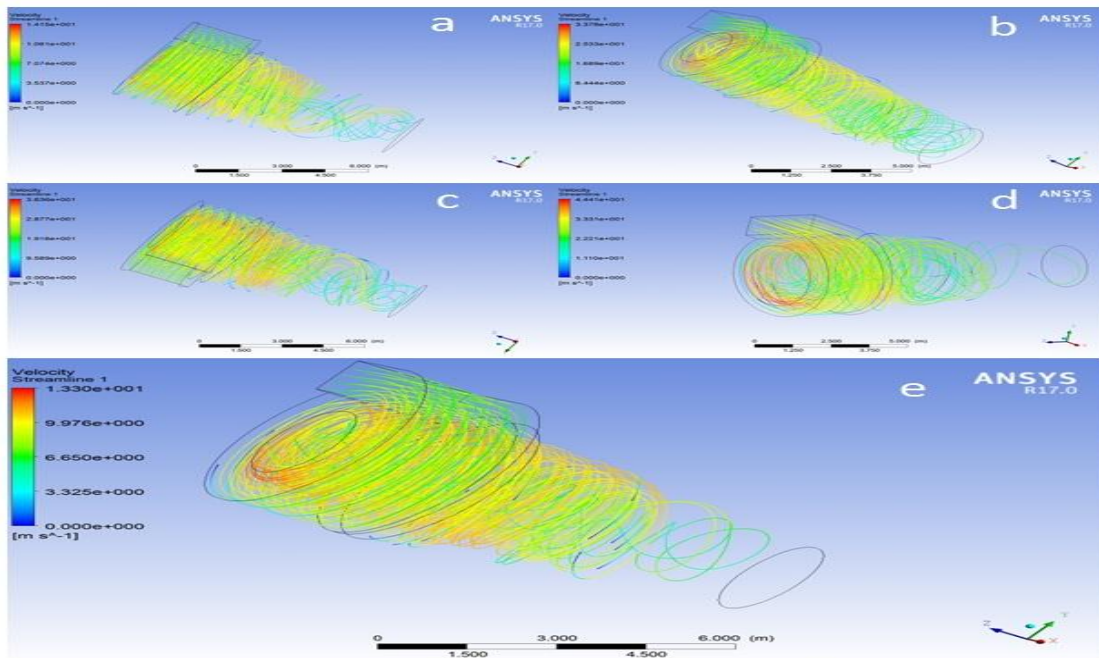

Figure 15 Stream lines in different models- a: higher inlet velocity (25m/s), b: taller cyclone (11.18 m), c: lower outlet pressure (zero Pa), d: greater outlet (4 m), and e: the main cyclone model

Figure 16 Static pressure charts for different models in z=-0.25 m- a: higher inlet velocity (25m/s), b: taller cyclone (11.18 m), c: lower outlet pressure (zero Pa), d: greater outlet (4 m), and e: the main cyclone model
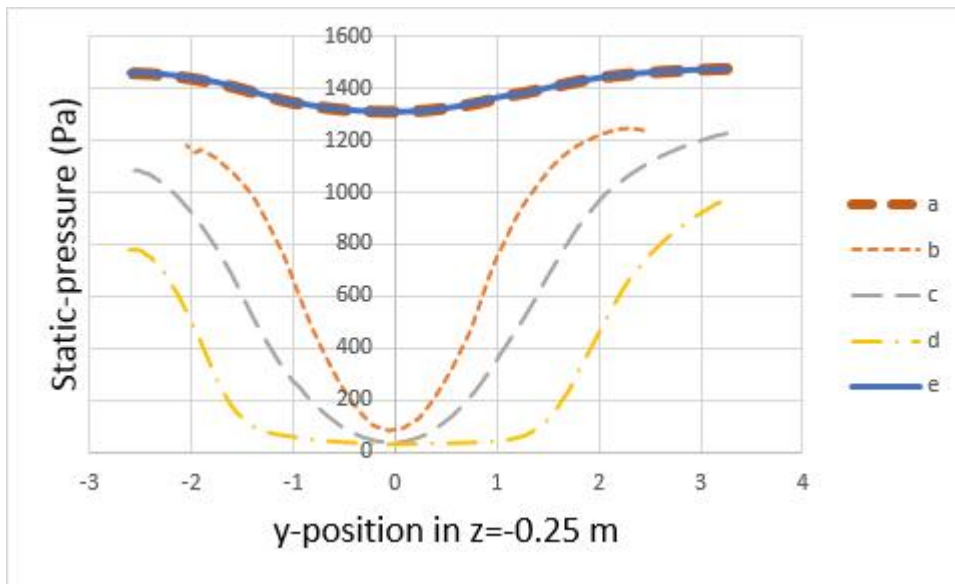
In Figure.18 the static pressure distribution in different heights, -0.25, -0.5, -1, and -3 m, for all models is provided. From this figure, it's obvious that for models with higher inlet velocity (18a), lower outlet pressure (18c), greater outlet (18d), and the main cyclone (18e), pressure distribution in these four heights are similar for each model and in each model an inconsiderable pressure drop at the center of cyclone in -3 m height. But in model with taller cyclone (18b) the pressure distribution of different heights is not similar to 4 other models. In this model (18b) by increasing the height from upper face of cyclone (from upper face of cyclone to lower one), the pressure drop decreases while keeping its symmetrical form.

In Figure.19 the distribution of axial velocity in z=-0.25 m for all 5 models is provided. All the models axial velocity curve is symmetric except model with greater outlet face (19d). Three models of higher inlet velocity (19a), lower outlet pressure (19c), and the main model of cyclone (19e) have

close axial velocity maximum which is about 5 m/s. But in model which has a taller cyclone, the axial velocity maximum gets higher value than all other 4 models about 25 m/s. Figure.19 for a once again proved that a cyclone with greater outlet (19d) causes lowest efficiency. The curve related to the greater outlet model (19d) is not symmetric and has three local pics.
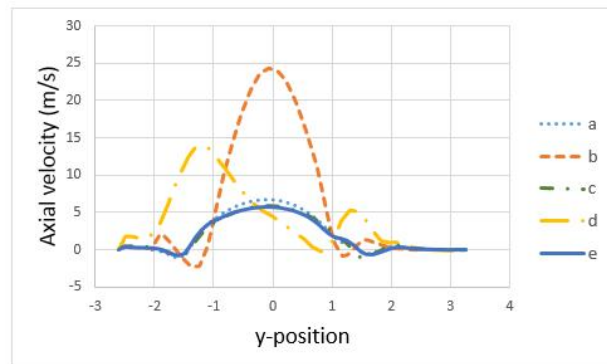

Figure 17 The axial velocity distribution in height of z=-0.25 m: : a: higher inlet velocity (25m/s), b: taller cyclone (11.18 m), c: lower outlet pressure (zero Pa), d: greater outlet (4 m), and e: the main cyclone model
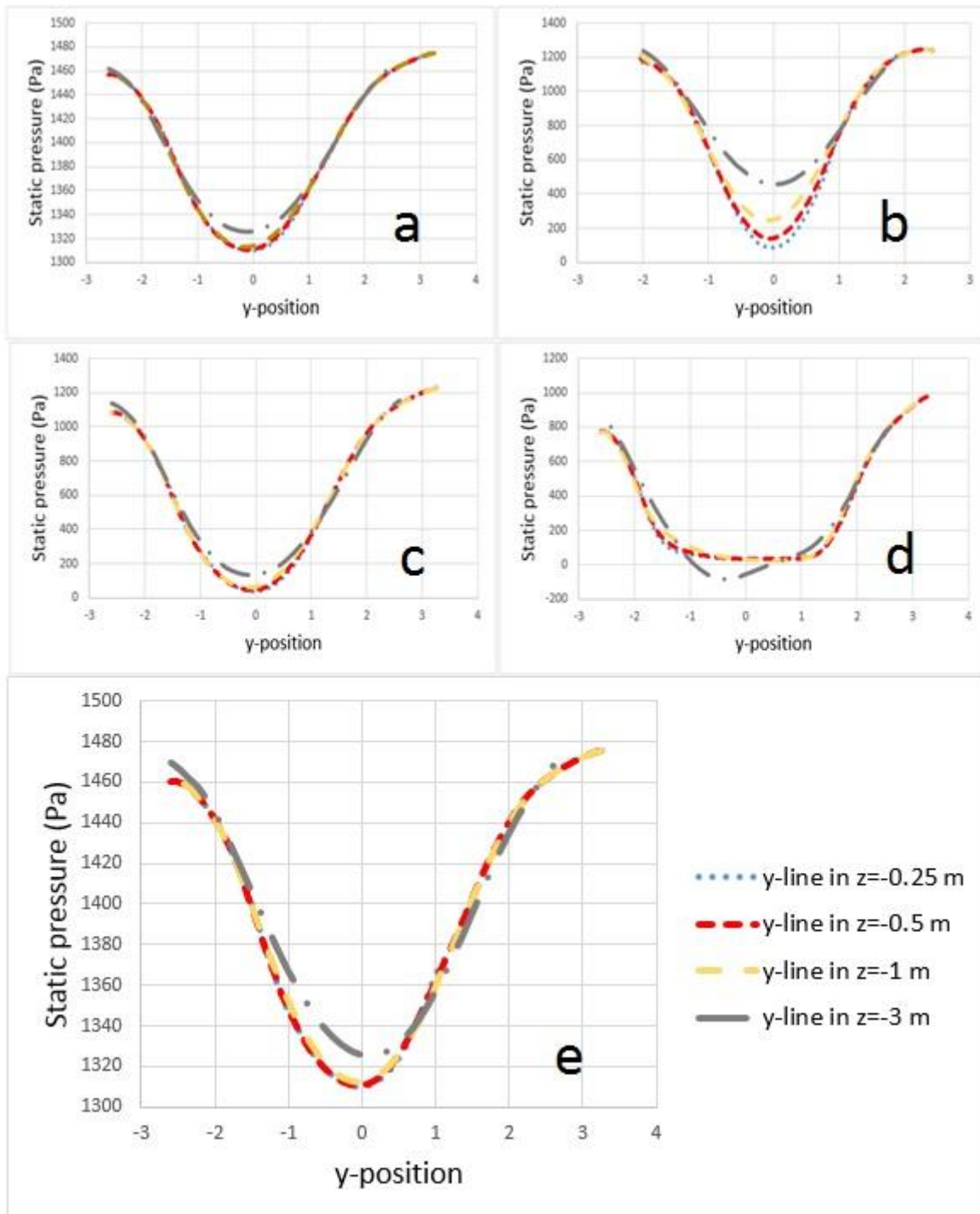
Figure 18 The static pressure distribution at different heights for all models: a: higher inlet velocity (25m/s), b: taller cyclone (11.18 m), c: lower outlet pressure (zero Pa), d: greater outlet (4 m), and e: the main cyclone model

8.Conclusion

As mentioned in primary parts of this paper, cyclones are mechanical systems with high applicability in different industries. In presented paper a separator cyclone from cement industry is investigated. Although cyclones are not complicated systems, the fluid flow inside has the most importance in their efficiency. In presented research the identified cyclone was modeled by Ansys Fluent 17 and then using analytical investigations, 4 different changes in geometrical and flow parameters was considered and modelled using Ansys Fluent 17, which their results provided above.

Results show that all geometrical and flow parameters could have significant effects on cyclone efficiency. After studies it was obvious that for this kind of cyclone, increasing outlet diameter would cause negative effects and lower efficiency. In fact, the fluid flow in the model with greater outlet face (d) has more and very high energy loss. The pressure drop in such model covers a greater area of cyclone and even near the walls pressure drop is more significant. While by increasing the cyclone vertical height (b) the story is completely different. The taller cyclone (b) has a symmetric fluid flow even at lower heights near the cyclone end. In this model, pressure drop occurs just near the cyclone center, which decreases from up to end of cyclone height. It has a higher axial velocity comparing to other 4 models at the center of cyclone (about 5 times higher), which causes a regular flow to outlet.

In other two models, the effects of inlet and outlet flow parameters were tested. It shows that by increasing the fluid velocity at the cyclone inlet, stream distribution was more equally than the real cyclone. In another one, the impacts of outlet pressure decreasing were tested. Results show that increasing inlet velocity has better impacts rather than decreasing outlet pressure. In higher velocity the stream concentration distributed more equally than lower outlet pressure and the main model of cyclone. Both flow parameters which are higher inlet velocity and lower outlet pressure make a symmetric flow inside the cyclone. But these flow parameters changes couldn't change the axial velocity significantly.

In fact, 3 of 4 parameters could increase the cyclone flow efficiency while the model with greater outlet face was not usable and applicable. But results show that this cyclone is not optimum and need important optimizations. Two models of taller cyclone and higher inlet velocity are more applicable and efficient, while the first one is the most efficient model.

At the end, some important results and conclusions are listed below:

1. Both geometrical and flow parameters have significant impacts on cyclone efficiency
2. Increasing the cyclone outlet diameter in this kind of cyclone decreases the flow efficiency
3. Increase of this cyclone height will increase its efficiency
4. Increase of inlet velocity for this cyclone increases the flow efficiency
5. Lower pressure at the cyclone outlet could increase the flow efficiency if be possible
6. For this cyclone, increase of cyclone height is the most effective

Using results obtained in this research we suggest, to have a more efficient cyclone:

- Increase the cyclone height if possible
- If the increase of cyclone height is not possible, the inlet velocity should be increased
- The cyclone outlet pressure shouldn't be increased, and could be decreased as more as possible
- The cyclone outlet diameter be as it is in the main cyclone (no change)

We suggest for future studies to:

- To study about decreasing cyclone end face diameter
- To study about the erosion inside the cyclone

References

[1] T. A. Sedrez, R. K. Decker, M. K. da Silva, D. Noriler, H. F. Meier, Experiments and CFD-based erosion modeling for gas-solids flow in cyclones, Powder Technology. 311 (2017) 120–131.

[2] E. Balestrin, R.K. Decker, D. Noriler, J.C.S.C. Bastos, H.F. Meier, An alternative for the collection of small particles in cyclones: experimental analysis and CFD modeling, Sep. and Purif. Technology. (2017).

[3] K. Elsayed, Design of a novel gas cyclone vortex finder using the adjoint method, Sep. and Purif. Technology. 142 (2015) 274-286.

[4] E. Balestrin, R.D. Luciano, D. Noriler, R.K. Decker, H.F. Meier, Sensibility Study of the Reynolds Stress Model Parameters for Swirling Flows in Cyclones, Chemical Engineering Transactions. 43 (2015).

[5] D. Noriler , A.A. Vegini, C. Soares , A.A.C. Barros, H.F. Meier, M. Mori, A new role for reduction in pressure drop in cyclones using computational fluid dynamics techniques, Brazilian J. Chemical Engineering. 21 (2004) 93-101.

[6] K. Elsayed, C. Lacor, The effect of cyclone vortex finder dimensions on the flow pattern and performance using LES, Computers & Fluids. 71 (2013) 224-239.

[7] M. Parsi, M. Agrawal, V. Srinivasan, R.E. Vieira, C.F. Torres, B.S. McLaury, S.A. Shirazi, CFD simulation of sand particle erosion in gas-dominant multiphase flow, J. Nat. Gas Sci. Engineering. (2015).

[8] C.A. Youngdahl, Nondestructive monitoring of erosive wear in transfer lines and cyclones at synuels pilot plants, Paper Presented at the Corrosion/84, International Corrosion Forum Devoted Exclusively to the Protection and Performance of Materials, 1984.

[9] R. Utikar, N. Darmawan, M. Tade, Q. Li, G. Evans, M. Glenny, V. Pareek, Hydrodynamic Simulation of Cyclone Separators, 2010 (Chapter 11).

[10] R.E. Vieira, S. Sajeev, S.A. Shirazi, B.S. McLaury, G. Kouba, Experiments and modelling of sand erosion in gas-liquid cylindrical cyclone separators under gas production and low-liquid loading conditions, 17th International Conference on Multiphase Production Technology, BHR Group, 2015.

[11] C.A.R. Duarte, F.J. de Souza, V.F. dos Santos, Numerical investigation of

mass loading effects on elbow erosion, Powder Technology. 283 (2015) 593–606.

[12] R.E. Vieira, A. Mansouri, B.S. McLaury, S.A. Shirazi, Experimental and computational study of erosion in elbows due to sand particles in air flow, Powder Technology. 288 (2016) 339–353.

[13] H. Safikhani, Modeling and multi-objective Pareto optimization of new cyclone separators using CFD, ANNs and NSGA II algorithm, Advanced Powder Technology. (2016).

[14] K. Elsayed, C. Lacor, The effect of cyclone inlet dimensions on the flow pattern and performance, Appl. Math. Model. 35 (2011) 1952–1968.

[15] M. Wasilewski, Analysis of the effect of counter-cone location on cyclone separator efficiency, Sep. and Purif. Technology 179 (2017) 236–247.

[16] M. Wasilewski, L. S. Brar, Optimization of the geometry of cyclone separators used in clinker burning process: A Case Study, Powder Technology. (2017).

[17] C.W. Haig, A. Hursthouse, D. Sykes, S. Mcilwain, The rapid development of small scale cyclones-numerical modelling versus empirical models, Appl. Math. Model. (2015).

[18] C. Hirsch, Numerical Computation of Internal and External Flows: The Fundamentals of Computational Fluid Dynamics, second ed., John Wiley & Sons, Burlington, 2007.

[19] A-N. Huang, N. Maeda, S. Sunada, T. Fukasawa, H. Yoshida, H-P. Kuo, K. Fukui, Effect of cold air stream injection on cyclone performance at high temperature, Sep. and Purif. Technology (2017).

[20] W.D. Griffiths, F. Boysan, Computational fluid dynamics (CFD) and empirical modeling of the performance of a number of cyclone samplers, J. Aerosol Sci. 27 (1996) 281–304.

[21] O. Hamdy, M. A. Bassily, H. M. El-Batsh, T. A. Mekhail, Numerical study of the effect of changing the cyclone cone length on the gas flow field, Appl. Math. Model. 46 (2017) 81–97.