Impact of geometric parameters and inlet boundary conditions on the behavior of a cyclone separator

Mazen Wahed[1], Abouelmagd Abdelsamie[2], Wael Abbas[3],Nehad Zidan[1]???

[1]Physics and Engineering Mathematics Department, Faculty of Engineering- Elmattaria, Helwan University, Cairo, Egypt.
[2]Laboratory of Fluid Mechanics, Mechanical Power Engineering Department, Faculty of Engineering-Elmattaria, Helwan University, Cairo, Egypt.
[3]Basic and Applied Sciences Department, College of Engineering and Technology, Arab Academy for Science and Technology, Cairo, Egypt.

Abstract

This work aims to investigate, numerically, the impact of different geometrical parameters on pressure drop and tangential velocity in a cyclone separator. These parameters are the size of the rectangular cross-sectional inlet area, shape of the inlet-cross-sectional area, and mass flow rate. Different shapes of inlet cross-sectional area were chosen to be either rectangle (standard), circle, or ellipse. In this work, the impact of different turbulent models is tested first. These models are shear stress transport model (SST) k-ω and k-ε which are compared with a laminar case (without any turbulence model). The cases with and without turbulence model are validated with available reference data. It has been found that the k-ε model agree well with the reference data. The larger inlet rectangular cross-sectional area leads to higher ratio of static pressure between the center of the cyclone and edges. On the other side, the tested range of mass flow rate (0.04, 0.08 and 0.16 Kg/s) to known effect this parameter in the static pressure and tangential velocity inside the cyclone. Furthermore, the static pressure and the tangential velocity have been observed in the case with ellipse cross-sectional inlet compared with that of rectangular- and circular-cross sectional area.

.

Keywords: Cyclone separator; Computational fluid dynamics; Ellipse Cyclone.

1. Introduction

Cyclone separators and dust cyclones are separation devices which sorting the different density at composite material that use the principle of inertia moment of body to remove particulate of any material from impure gases. Cyclone separators are one of air pollution control devices known as pre-cleaners, which work much like a centrifuge, but with a continuous feed of dirty air. There are several applications use this cyclone in the industrial, food, medical, water treatment station and the cement factories [1]. A lot of researches investigated the cyclone design, these researches have two points of view. One of them based on the geometry of the cyclone, and the other is based on the employed numerical models.

Derksen et al., 2006 [2] discussed the effect of particle mass-loading on the gas flow and solid particle motion in a cyclone separator (Stairmand design). They also examined the effect of the turbulence and swirl on the pressure inside the cyclone, and the collection efficiency. They found that, the collection efficiency in turbulence case is better than the swirl one.

Elsayed and Lacor 2010 [3] compared the geometry of Stairmand design and a novel design they suggested. It has been found that the new cyclone design results in nearly one-half the pressure drop obtained by the old Stairmand design at the same volume flow rate. Elsayed and Lacor, 2011 [4] illustrated the effect of different dimensions of the inlet of the cyclone used the Reynold stress model for turbulence, they obtained the maximum tangential velocity which decreases at increasing the height and width of inlet cyclone. They also found that the pressure drop decreases with increasing the inlet dimensions. Pirker et al., 2012 [5] explained the phenomenon of cyclone short-cut flow by using a hybrid turbulence model depend on the lattice Boltzmann method or Navier-stokes flow to resolve unsteady flow features in the top annulus region of a counter current gas cyclone. Parvaz et al., 2020 [6] examined the effect of different outlet shapes of a cyclone, such as cylindrical, inverted cone, conical, diamond and outlet without dip leg at constant inlet velocity. They concluded that the case with cylindrical dip leg has the highest tangential velocity, whereas the case with conical shape has a local high tangential velocity in dust zone. The minimum and maximum pressure drop observed to be in diamond and cylindrical dip leg arrangements. Elsayed and Lacor, 2011 [7] investigated, numerically using large eddy simulation (LES), the effect of the cone tip-diameter on the flow and performance of cyclone separator. They found the cone tip decreases with increasing the tangential velocity and increasing the pressure drop. Elsayed and Lacor, 2013 [8] illustrated the effect of the vortex finder dimensions on the performance on the different cyclone separators by using the large eddy simulation (LES), they found that decreasing the diameter vortex finder increasing pressure drop and increasing the tangential velocity. Karagoz et al., 2013 [9] examined a new design for cyclone. They found that the dust collected in the dust zone easier than the old design, which increases the collection efficiency. Brar and Elsayed, 2017 [10] improved the performance of the gas cyclones by adding multi-inlet ports relying on large eddy simulation (LES) method. Sun et al., 2017 [11] studied different geometries of cyclone, they found that decreasing the inlet height, inlet width and the vortex finder diameter led to increase the collection efficiency. Parvaz et al., 2018 [12] improved efficiency of the cyclone by increasing the inner cone of the cyclone which decreasing the pressure drop and the collection efficiency. Brar and Elsayed, 2018 [13] illustrated the movement effect of the position of the vortex finder around the global axis of the cyclone on the cut off diameter of cyclone and the pressure drop. Wasilewski and Brar, 2018 [14] studied the effect of the angle of the inlet duct bend on the separation efficiency and pressure drop in cyclone separators. They found increasing the separation efficiency with an increase in the angle. Wasilewski et al., 2020 [15] analyzed the effect of varying the vortex finder diameter and its insertion length on the flow field and overall performance in a square cyclone. They found that decreasing the diameter of vortex finder improves the separation efficiency by changing the shape of corner to be square but this change increasing the pressure drop. Izadi et al., 2020 [16] illustrated the different types of the hydro cyclones to obtain the maximum efficiency. They increasing the separation efficiency by increasing the inlet flow of cyclone.

Babaoğlu et al., 2021 [17] studied the changing shapes of the inlet cross-section cyclone effect on the static pressure, axial velocity and tangential velocity. They found that the maximum tangential velocity in the rectangle inlets shape. Dziubak and Bakała, 2021 [18] presented the possibilities of modifying the structure of axial flow cyclone in the direction of increasing separation efficiency. Li et al., 2022 [19] illustrated the effect of different geometry and SCO_2 content on the performance of SCFs-solid cyclone which change the static pressure, tangential velocity and axial velocity, then found the highest tangential velocity in the double inlet cyclone.

From the reviewed literature, it is very clear that the most important improvements of the cyclone are the geometrical development and the initial/boundary conditions. Therefore, in the current work, the shape of the cross-sectional area will be investigated by examining three different shapes: rectangular (standard), circular, and ellipse. The impact of changing the mass flow rate on the cyclone behavior will be studied as well.

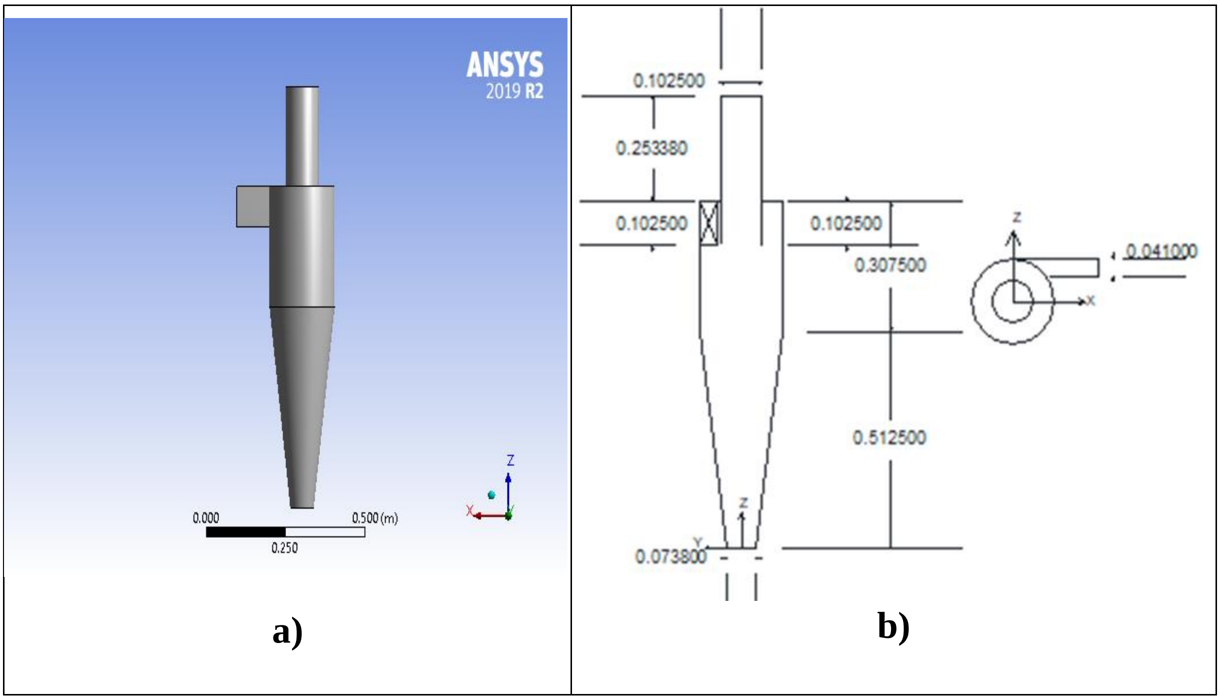
2. Methodology

In this section, the geometry, mesh, and the numerical method will be discussed in details

2.1 Cyclone Geometry

In this work the considered geometry is presented in

Figure 1, which is similar to that introduced by Stairmand-design [3] design but increase the length of vortex finder to twice the standard length to avoid the backflow. Presenting one of the most popular design guidelines (Stairmand-design) which states that the cylinder height and the exit tube length should be, respectively,1.5 and 0.5 times of the cyclone body diameter as shown in



a) **b)**

Figure 1 a) Schematic diagram of 3D-cyclone, b) sketch of the cyclone shows the dimension in meters.

2.2 Simulation the cyclone with particle and without particle.

The simulation of the cyclone was done by using the model k-ε twice, one simulation without particles and the one with the particles. It's found that adding the particles don't affect on the simulation results. This is because the volume loading of the particle is very small, and one-way coupling is only activated.

2.3 The Mesh Topology

The numerical results were obtained at 154215 mesh elements. The simulation with this mesh is compared with the results from Ref. [3]. It has been observed that the difference between the average static pressure in the current work and that of Ref. [3] is about 29.75 %. This error is considered as accepted error in the current work, because the limited available resources.

2.4 Numerical models

Before investigating the geometrical effect and the mass flow rate, the impact of turublence model will be tesed first. Three different case will be examined for that. A case without any turbulence, a case with (SST) k-ω and another one with k-ε [20-21-22].

3. Results and Discussion

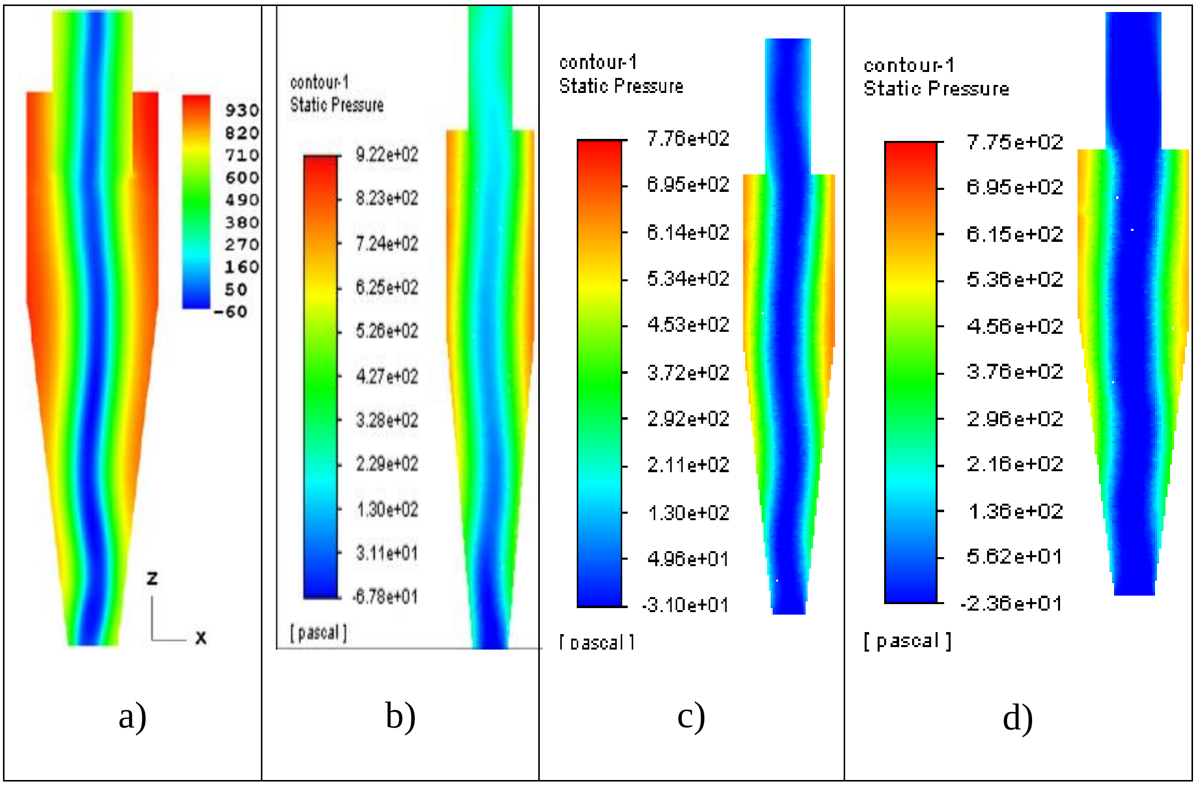
In this section, all simulation were built on the top of the Stairmand geometry design and the design introduced by Elsayed et al. (hereinafter named as design of Ref. [3]) The tested examined in the current work is summarized in Table

Table 1. The parameters change for cyclone

No.1	Case Study (inlet mass flow rate)	Shape inlet cyclone
Case-1	k-ε model (0.08 Kg/s)	Original rectangular shape with dimension of Length = 20.5 cm; Width= 8.2 cm
Case-2	(SST) k-ω model (0.08 Kg/s)	The original shape
Case-3	Laminar model (0.08 Kg/s)	The original shape
Case-4	Mass flow rate (0.04 Kg/s)	The original shape
Case-5	Mass flow rate (0.16 Kg/s)	The original shape
Case-6	Half dimension of inlet cyclone	Rectangle shape with dimension of Length = 5.125 cm; Width= 2.05 cm
Case-7	Double dimension of inlet cyclone	Rectangle shape with dimension of Length = 20.5 cm Width= 8.2 cm
Case-8	Circle shape inlet cyclone	Circle shape with dimension of Diameter = 5.86 cm
Case-9	Ellipse shape inlet cyclone	Ellipse shape with dimension of Major axis = 10.25 cm; Minor axis =19.55 mm

3.1 Impact of models and validation

Looking at the pressure and tangential velocity contours in Fig. 2 and Fig. 3, respectively, it can be observed that the simulation with k-ε model (Case-1) predicts the distribution of pressure and tangential velocity better than the other two cases (Case-2 and Case-3). Figures 4 and 5 show the pressure and the tangential velocity at line-cut at coordinates of (0.1025, -0.1025,0.7) and (0.1025, -0.1025,0.7), named as "line cut-1". As it can be observed the k-ε model (Case-1) can predicts the maximum value of pressure in better way that that of the other cases (Case-2 and Case-3) at this line-cut. Therefore, the k-ε model for turbulence will be employed in the rest of the work.



a) b) c) d)

Figure 2 Static pressure contour at center plane. a) Ref. [3], b) Case-1, c) Case-2, d) Case-3.

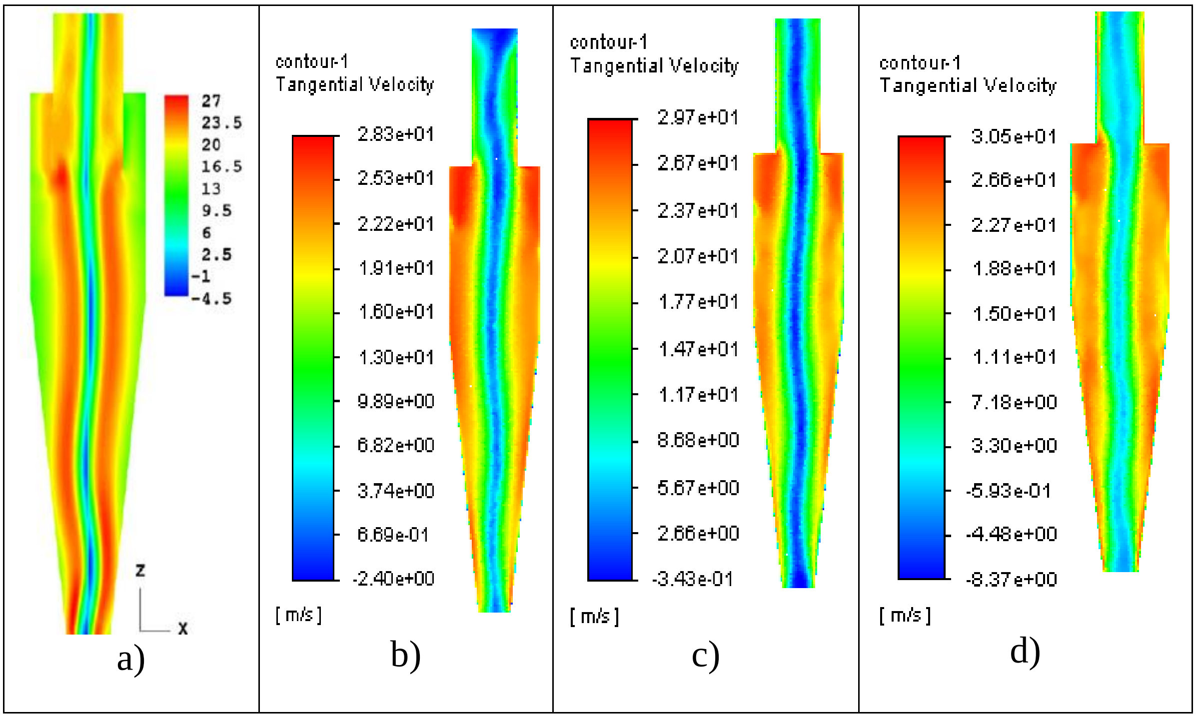


Figure 3 Tangential velocity contour at center plane. a) Ref. [3], b) Case-1, c) Case-2, and d) Case-3.

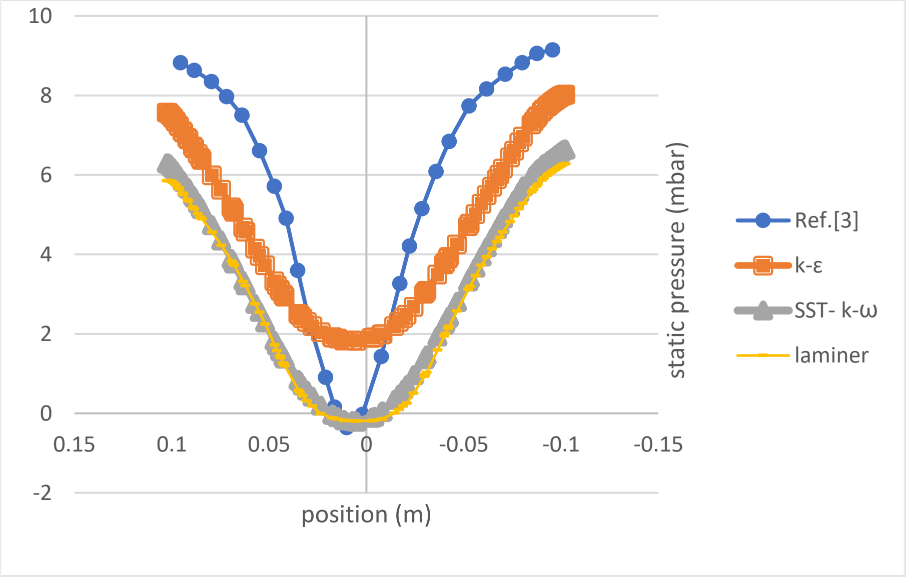


Figure 4 Line-cut (line cut-1) for static pressure.

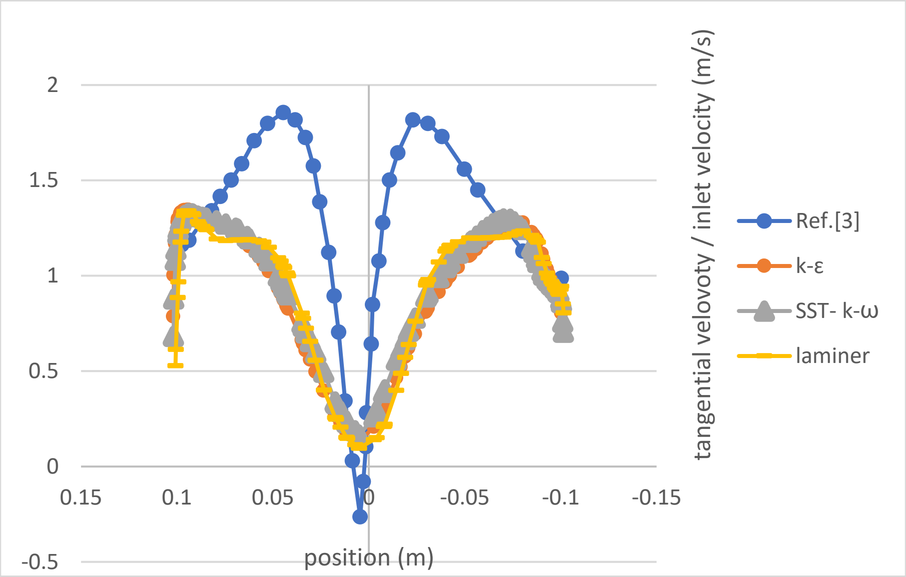


Figure 5 Line-cut (line cut-1) for tangential velocity.

3.2 Impact of mass flow rate

The effect of different mass flow rates (0.04, 0.08 and 0.16 Kg/s) is examined in this section. From Figures 6 - 9, it can be concluded that this range of mass flow rate has no significant impact on the behavior of the cyclone separator.

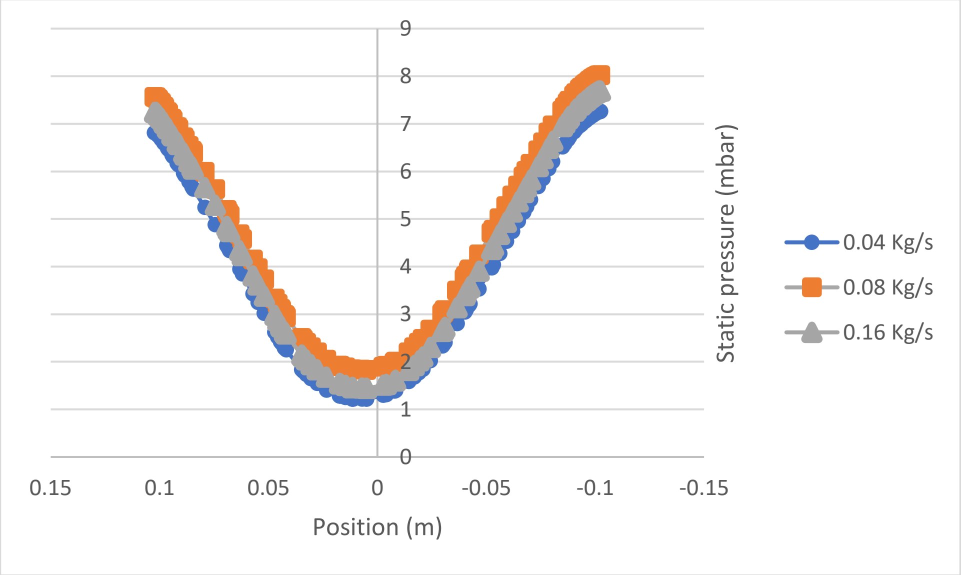


Figure 6 Line-cut (line cut-1) shows the impact of mass flow rate on the static pressure

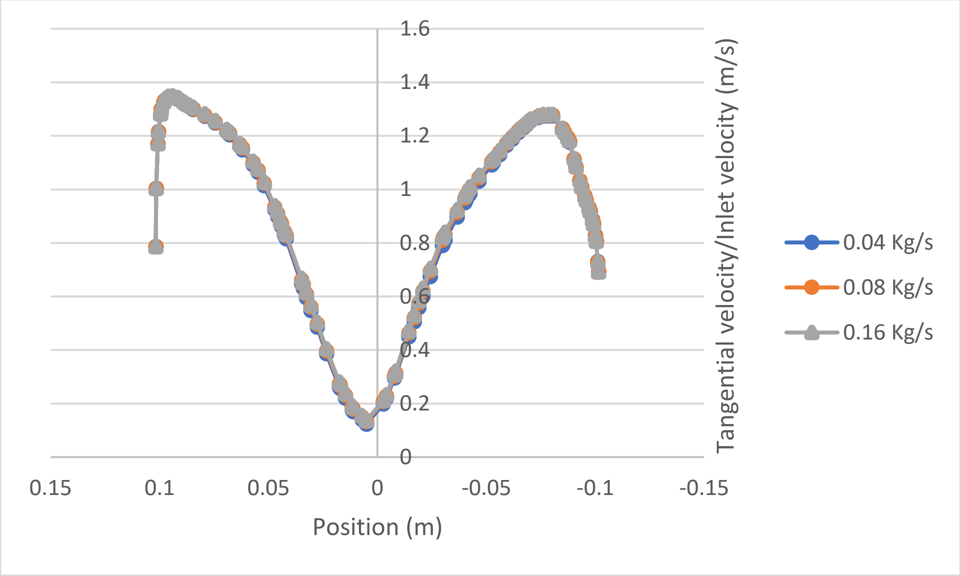


Figure 7 Line-cut (line cut-1) shows the impact of mass flow rate on tangential velocity

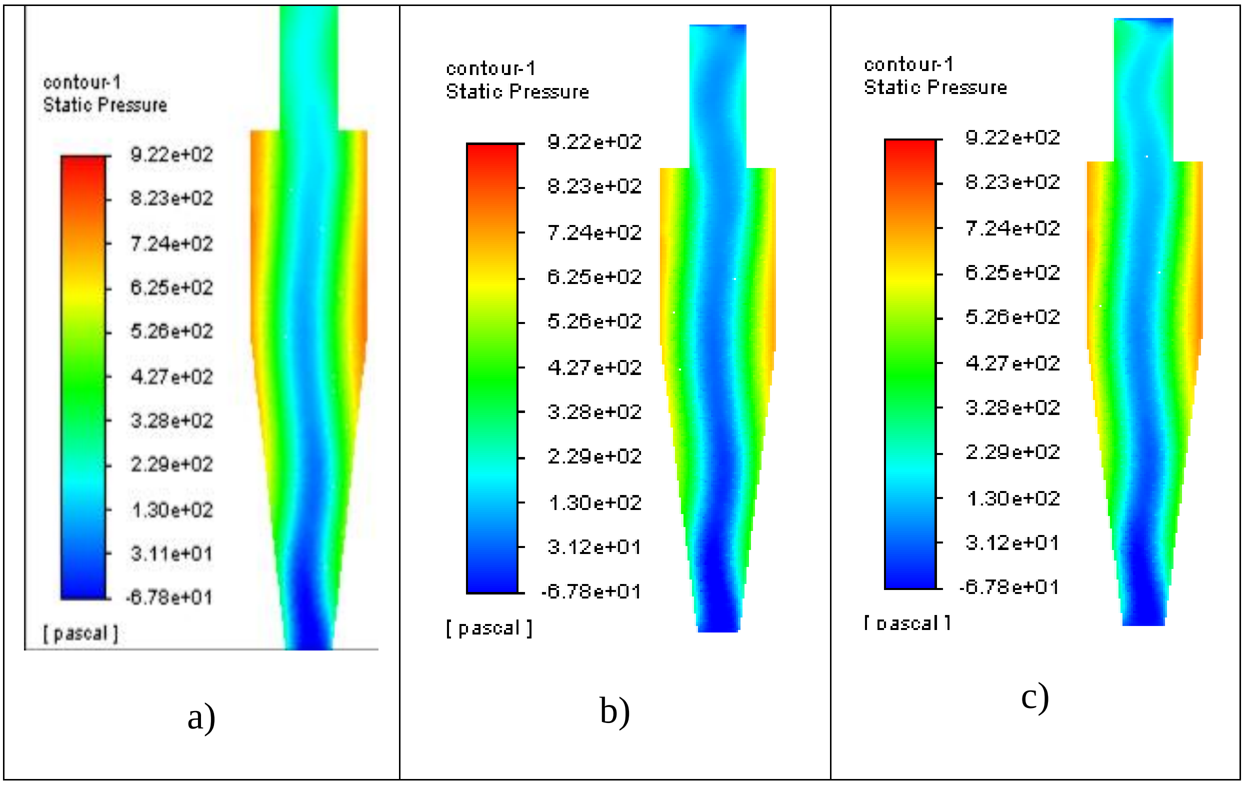
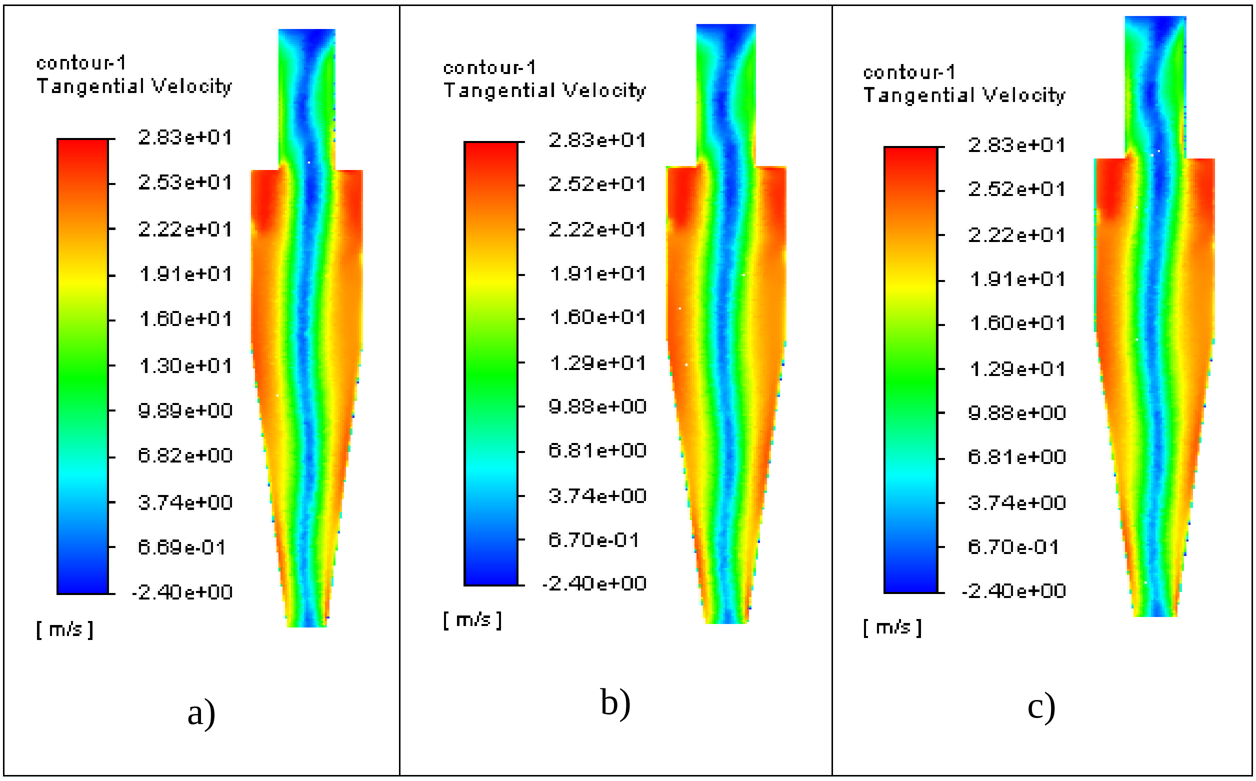


Figure 8 Impacts of mass flow rate on static pressure a) Case-1, b) Case-4, c) Case-5



a) b) c)

Figure 9 Impacts of mass flow rate on tangential velocity a) Case-1, b) Case-4, c) Case-5

3.3 Impact of the inlet size

In this section, the effect of changing the size of inlet rectangled-cyclone is investigated by changing the standard size original rectangular shape with dimension of length = 20.5 cm and width= 8.2 cm (Case-1) to half the size (Case-6) and double the size (Case-7) as it in Table 1. From 10-13 it is very clear that the increasing the inlet size of the rectangular cross-sectional areal lead to high variation in the pressure and velocity between the center of the cyclone and the edges. This is a recommended behavior to increase the separation the particles. Therefore, increasing the inlet size would lead to more separation.

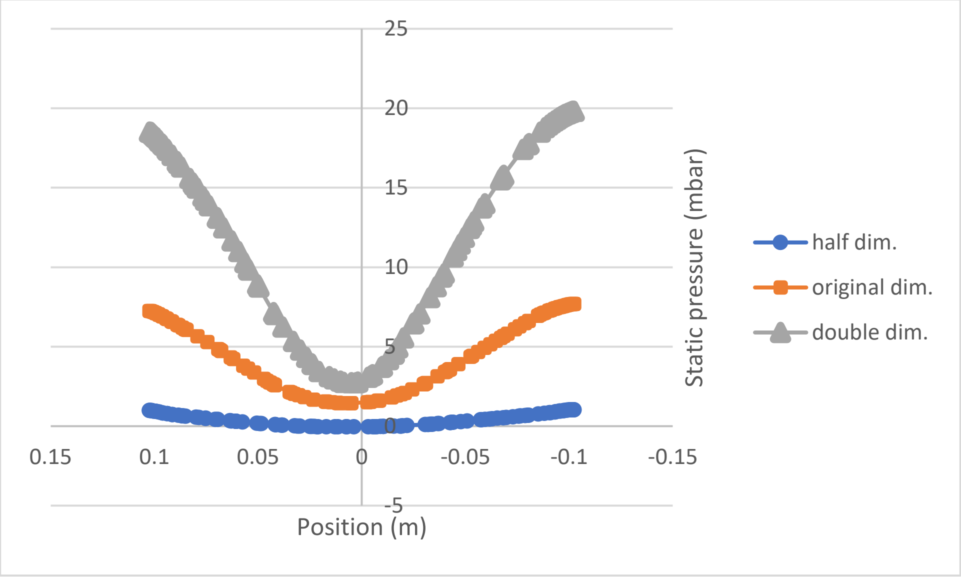


Figure 10 Line-cut (line cut-1) shows the impact of size of the rectangular inlet cross-sectional area on the static pressure

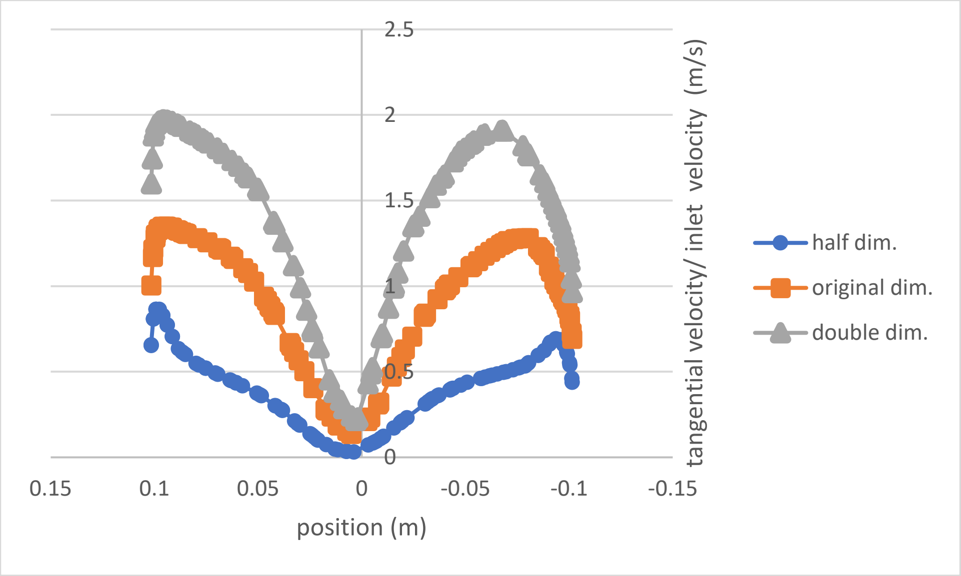
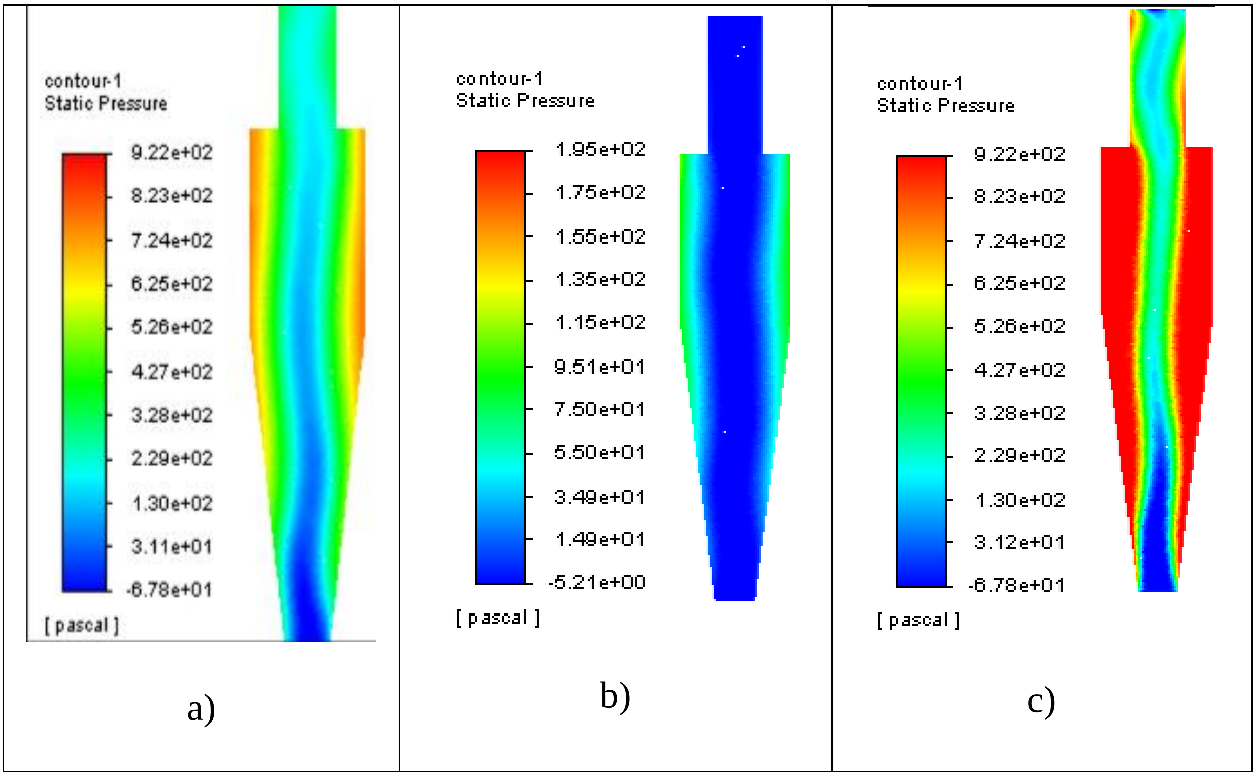


Figure 11 Line-cut (line cut-1) shows the impact of size of the rectangular inlet cross-sectional area on tangential velocity.

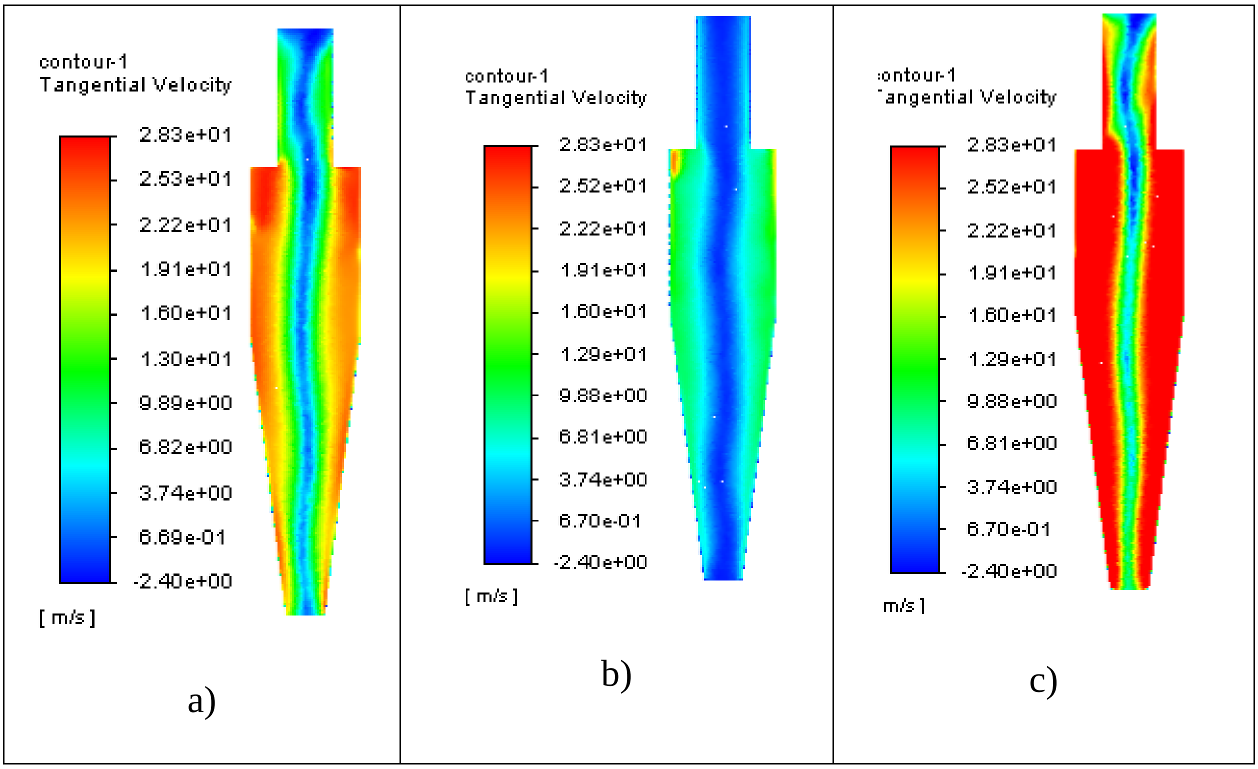


a)

b)

c)

Figure 12 Impacts of size of the rectangular inlet cross-sectional area on static pressure a) Case-1, b) Case-6, c) Case-7.



a)

b)

c)

Figure 13 Impacts of size of the rectangular inlet cross-sectional area on tangential velocity a) Case-1, b) Case-6, c) Case-7.

3.4 Impact of the shape of the inlet

In this section, the geometry of the inlet shape is changed to study the impact of the inlet-shape on the pressure and the tangential velocity. From Figs. 14-17 it can be observed that both pressure and tangential velocity are minimum in case of ellipse-shape are maximum for rectangle -shape.

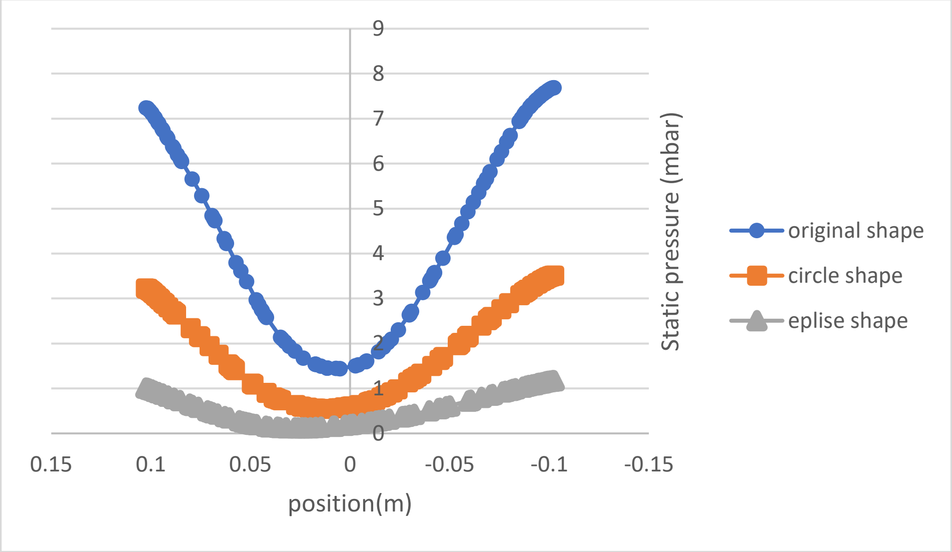


Figure 14 Line-cut (line cut-1) shows the impact of the shape of the inlet cross-sectional area on the static pressure

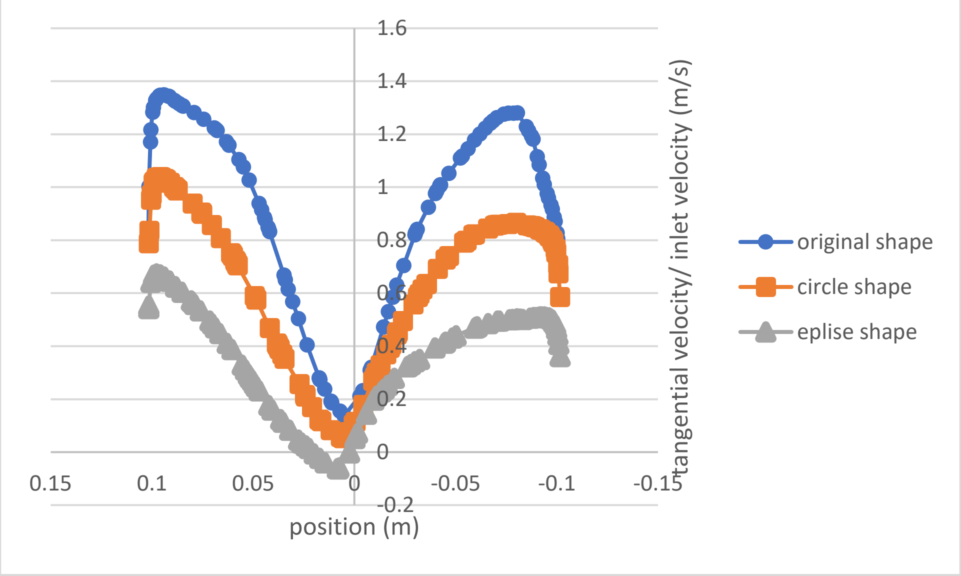


Figure 15 Line-cut (line cut-1) shows the impact of the shape of the inlet cross-sectional area on the tangential velocity

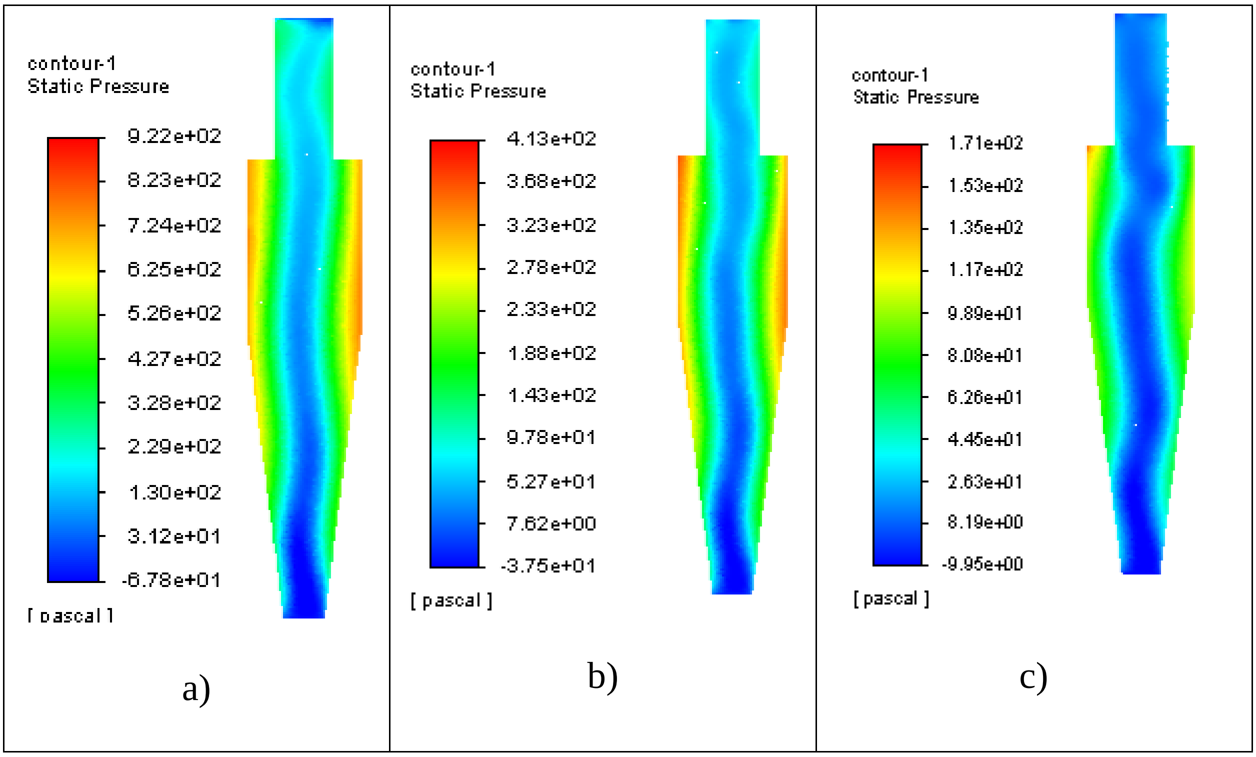


Figure 16 Impact of the shape of the inlet cross-sectional area on static pressure a) Case-5, b) Case-8, c) Case-9

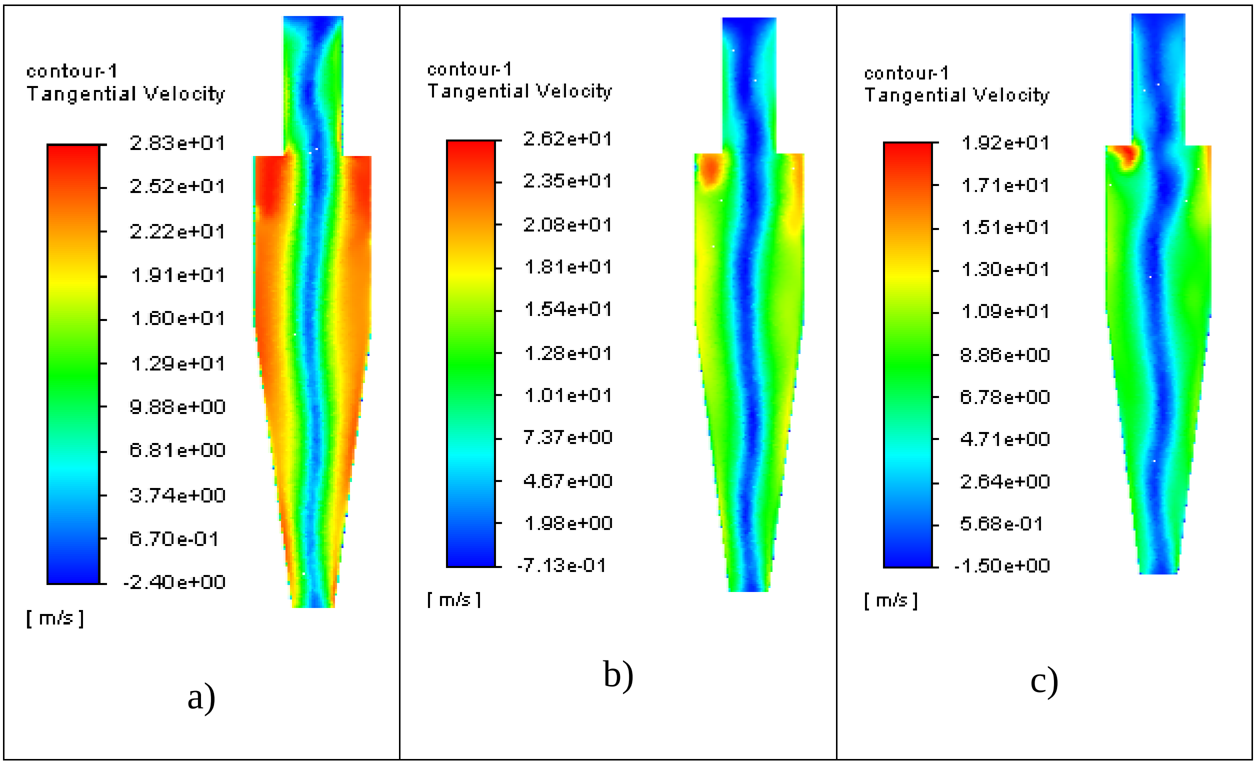


Figure 17 Impact of the shape of the inlet cross-sectional area on tangential velocity a) Case-5, b) Case-8, c) Case-9

4. Conclusion

This work studied numerically the impact of different parameters on the behavior of a cyclone separator using ANSYS-Fluent 19-R2 package. In this work, different turbulence models have been tested first: (1) Shear stress transport model (SST- k-ω), (2) k-epsilon turbulence model (k-ε) and (3) laminar model. It has been found that the k -ε model agrees more with the reference date Ref. [3]. It has been observed that changing the mass flow rate of inlet cyclone to be half (0.04 Kg/s) and double (0.16 Kg/s) of the original mass flow rate (0.08 Kg/s) has no significant effect on the static pressure and tangential velocity. Also changing the size of rectangular-cross sectional inlet to half size and double size of the original standard leads to high variation in the pressure and velocity between the center of the cyclone and the edges, hence more gas-particle separation is expected. Finally, investigating the effect of changing the shape of inlet cyclone by changing the shape to be rectangular shape, ellipse shape, or circle shape. The minimum static pressure and minimum tangential velocity have been observed in the case with ellipse cross-sectional inlet compared with that of rectangular- and circular-cross sectional area. In the next publication, more parameters will be studied at larger number of mesh element.

Reference

[1] H. Patel, A. Darji, A desgin and analysis of cyclne separator for high efficiency by using CFD analysis, Indian journal of research,volume (7),2250-1991, 2018.

[2] J.J. Derksen, S. Sundaresan, H.E.A. van den Akker, Siumation of mass loading effects in gas-solid cyclone separator, Powder Technology, 163, 59-68, 2006

[3] K. Elsayed, C. Lacor, Optimization of the cyclone separator geometry for minimum pressure drop using mathematical models and CFD simulations, Chemical Engineering Science, 65, 6048-6058, 2010

[4] K. Elsayed, C. Lacor, the effect of cyclone inlet dimensions on the flow pattern and performance, Applied Mathematical Modelling,35,1952-1968, 2011.

[5] S. Pirker, C. Goniva, C. Kloss, S. Puttinger, J. Houben, S. Schneiderbauer, Application of a hybrid Lattice Boltzmann–Finite Volume turbulence model to cyclone short-cut flow, Powder Technology 235, 572–580, 2012

[6] F. Parvaz, S.Hosseini, K. Elsayed, G. Ahmadi, Influence of the dipleg shape on the performance of gas cyclones, Separation and Purification Technology, 233, 116000, 2020

[7] K. Elsayed, C. Lacor, Numerical modeling of the flow field and performance in cyclones of different cone-tip diameters, Computers & Fluids,51,48-59, 2011

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

[9] I.Karagoz, A.Avci, A. Surmen, O.Sendogan, Design and performance evaluation of a new cyclone separator, Journal of Aerosol Science,59,57-64, 2013

[10] L. Brar, K. Elsayed, Analysis and optimization of multi-inlet gas cyclones using large eddy simulation and artificial neural network,Powder Techology,5910,30121-3, 2017.

[11] X. Sun, S. Kim, S. Yang, H. Kim, J. Yoon, Multi-objective optimization of a Stairmand cyclone separator using response surface methodology and computational fluid dynamics,Powder Techology,5910,30533-8,2017.

[12] F. Parvaz, S. Hosseini, K. Elsayed, G. Ahmadi, Numerical investigation of effects of inner cone on flow field, performance and erosion rate of cyclone separators, Separation and Purification Technology,201,223-237,2018.

[13] L. Brar, K. Elsayed, Analysis and optimization of cyclone separators with eccentric vortex finders using large eddy simulation and artificial neural network, Separation and Purification Technology,5866,30293-3,2018.

[14] M. Wasilewski, L. Brar, Effect of the inlet duct angle on the performance of cyclone separators, Separation and Purification Technology,5866,33354-9,2018.

[15] M. Wasilewski, L. Brar, G. Ligus, Experimental and numerical investigation on the performance of square cyclones with different vortex finder configurations, Separation and Purification Technology, 239, 116588, 2020

[16] M. Izadi , A. Makvand , E. Assareh ,F. Parvaz, Optimizing the design and performance of solid-liquid separators, thermofluids,volume5-6,100033 2020.

[17] N. Babaoğlu, F. Parvaz, S. Hosseini, K. Elsayed,G. Ahmadi, Influence of the inlet cross-sectional shape on the performance of a multi-inlet gas cyclone,Powder Techology,384,82-99,2021.

[18] T. Dziubak and L. Ba,kała, Computational and experimental analysis of axial flow cyclone used for intake air filtration in internal combustion engines, Energies,14,2285,2021.

[19] Z. Li, Z. Tong, A. Yu, H. Miao, K. Chu, H. Zhang,G. Guo, J. Chen, Numerical investigation of separation efficiency of the cyclone with super critical fluid–solid flow, Particuology, 62,36-46,2022.

[20] F.Menter, "Zonal Two Equation k-ω Turbulence Models for Aerodynamic Flows", AIAA Paper 93-2906,1993.

[21] F.Menter, "Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications", AIAA Journal, vol. 32, no 8. pp. 1598-1605,1994.

[22] W.Jones and B.Launder, " The Prediction of Laminarization with a Two-Equation Model of Turbulence", International Journal of Heat and Mass Transfer, vol. 15, pp. 301-314,1972.