An Analytic Study of the Effect of a Vane on the Hydraulic Field around a Cylinder

Rafi M. QASIM[1], Tahseen Ali JABBAR*,[1]

*Corresponding author
[1]Department of Fuel and Energy, Basra Engineering Technical College,
Southern Technical University, Iraq,
rafi.mohammed@stu.edu.iq, tahseen.ali@stu.edu.iq*

DOI: 10.13111/2066-8201.2021.13.3.11

Received: 28 November 2020/ Accepted: 29 June 2021/ Published: September 2021

Abstract: *The flow pattern around the cylinder body is a very serious problem and this problem may become more serious and sensitive, when we place a vane neighboring to the cylinder. The present paper deals with the vane impact on the flow pattern around the cylinder. To investigate this problem the ANSYS fluent software is employed in order to achieve the two dimensional numerical analysis. Here, Reynolds Average Navier Stokes model is adopted. The investigation comprises the following hydraulic variables, like eddy viscosity, turbulent intensity, turbulent kinetic energy, turbulent dissipation rate, flow velocity profile, static pressure and pressure coefficient. The constant flow velocity, cylinder diameter and vane dimension are adopted in this analysis, while the different certain distance between the vane and the cylinder is considered. The used vane has a rectangular shape. In this analysis, it is clear that the vane plays a sensitive vital role in the hydraulic behavior of the flow pattern around the cylinder. The study has taken up four distances between the vane and the cylinder, these distances is a function of the cylinder diameter, in addition to the direct touch that happens between the vane and the cylinder. The analysis also shows that when the cylinder has direct touch with the vane, the dramatic reduction will occur in hydraulic variables.*

Key Words: *Cylinder body, Flow field, Flow pattern, turbulent flow, Vane, Vortex*

1. INTRODUCTION

The field of the turbulent flow around a cylindrical body like circular bridge pier can be defined as an unsteady complex flow field, containing horseshoe vortex formation. At this point, the wake vortex shedding, the flow separation and the design engineers often encounter a challenge in dealing with the drag force which is generated in a cylinder body when the flow passes it; consequently, this process must be taken into consideration in order to avoid any problem that may arise during the serviceability life. In this respect, several researchers have dealt with this issue from the past period up to the present time. Now the researchers ideas are tackled as follows:

Roshko, A (1960) [1] referred to the fact of the drag coefficient that resulted in the different researches as much closed, while in the supercritical region the results are shown with fluctuation. In his experiments and researches, he carried out the corrections of the wall interference to guarantee the accurate results. Mital, R (1995) [2] performed an analysis which depends on a large eddy simulation turbulent model. It is an initiative that wanted to utilize just another analysis method, different from the traditional one, in this type of flow. He experienced some difficulties in the simulation of the three dimensions, clearly this state is reflected on the computing resources. Dey et al. (1995) [3] suggested a model which allows to recognize the scour hole around the cylindrical pile in clear water. Breuer, (2000) [4] carried out a study by using a large eddy simulation for the high subcritical Reynolds number, in which a feasible comparison was obtained from the experimental data. Salaheldin et al. (2004) [5] used different models of turbulence to simulate the flow field around the circular piers. Li (2005) et al. and Zhao et al. (2005) [6, 7] carried out some experimental works to investigate the influences of the cylinder diameter and the approaching velocity on the depth of scour. Wei and Aode (2006) [8] studied numerically the scouring mechanism and turbulent flow field around the circular pier by using the method of a large eddy simulation. The obtained results indicated that, at the pier downstream, as the scour depth increases the shear stress of the bed decreases and undisturbed shear stress will approach also, and a gradual increase in the turbulent intensity is shown. Also, the pressure fluctuation and the vertical pressure gradient increase gradually. Marakkos and Turner (2006) [9] studied the cylinder upstream zone, by adopting Particle Image Velocimetry in order to show the behavior of the flow for the circular cylinder on an end-wall. Wei and Aode, (2007) [10] concentrated on the influence of the free surface on the structure of the flow. The work involved two step methods that link the two dimensional equations of the compressible ideal gas and the model of a large eddy simulation in order to calculate the three dimensional flow field together with the free surface around the pier. Rajani et al. (2009) [11] concentrated on the two dimensional and the three dimensional analysis of the flow that crosses the circular cylinder, considering various regimes of laminar flow. The study included the verification found between the computational results and the measured data. Gao et al. (2010) [12] performed a numerical simulation to investigate the impact of the turbulence around a vertical cylinder. Using the numerical analysis, Tchawe et al (2015) [13] investigated the flow around the emerged circular vertical cylinder in two dimensional open channel. The results showed, the increases in pressure and decreases in the velocity which occurs with Reynolds number, increasing at the upstream region from the cylinder. The results also illustrated the significant role of the cylinder diameter to the shortages of scour hole which occurs around the cylinder and the downstream wake forces from the cylinder. Azizi et al. (2016) [14] investigated the flow pattern by using numerical simulation around a pier that is circumfluent by the submerged vane. The three dimensions of the whole system were simulated by the employment of fluent software. Also, the results indicated the acceptable agreement between the numerical simulation and the experimental data. Al-saffar (2018) [15] carried out an experimental study to investigate the flow field around the circular cylinder, which was fixed normally to the bed of an open channel. In this experiment, various flow conditions were applied, altering from the subcritical to the supercritical conditions. The obtained results referred to the presence of two various relationships among the planes around the circular cylinder. Zaid et al. (2019) [16] investigated the flow around the bridge piers, mounted on a fixed flatbed by using three dimensional numerical model. The model depends on the Reynolds Average Navier Stokes Equation approach. The main target of this work points to the numerical model validation and depends upon the prior experimental data. Abdulhussein et al. (2019) [17] carried out an experimental work to study the impact of utilizing the strip guide flow panel device in order to diminish scour depth impact around the pier of the bridge. The study used three different sizes for the cylindrical pier and strip guide flow panel devices. The results showed a reasonable and noticeable efficiency of the scour reduction, and a high hydraulic factor of safety obtained and compared with scour around pier without using the device. The study inferred that the device works as an energy dissipater which diminishes the turbulent energy and shortages of the scour hazard around the pier. In this work, the vane which has a position at a certain distance, specified at the upstream of the pier, has been used to decrease the turbulent intensity effect. The vane is a device of the altering turbulent flow reduction measure which was used suitably to avoid and/ or deflect the turbulent flow mechanisms to decrease the turbulent flow intensity which is adjacent to the pier. This work investigated the impact of the vane device in diminishing the turbulent intensity zone that generated around the pier. It depended on several numerical software runs that used a fluent program.

2. PROBLEM MODELLING

Fluent ANSYS software is used to model the entire system which consists of four components, and these components are circular cylinder, vane, water and channel, that are based on the finite volume method as shown in figure 1 below. The formulation and modeling of the current problem has been handled. Different types of elements with different properties have been used to describe the fluid –structure interaction problem. The physical assumption**s** which **were** adopted in the simulation and analysis of the present problem are; water which is considered as the flow material along the channel, water is also considered as incompressible liquid and has constant physical properties. Two dimensional flow analysis. Table 1 shows the physical properties of water.

Table 1 - Water physical properties

Density (Kg/m^3)	Viscosity (kg/m.s)
998.2	0.001003

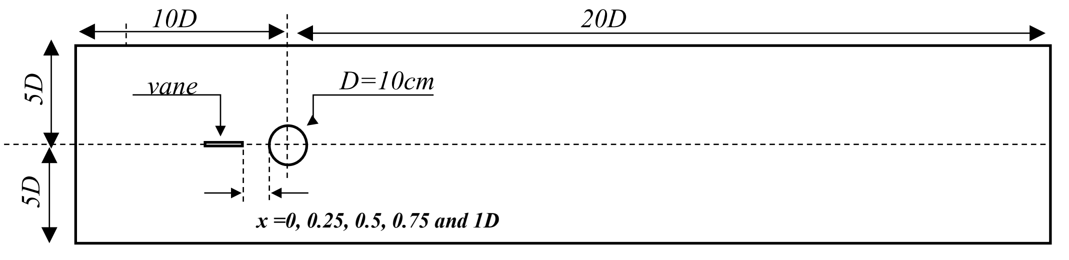


Fig. 1 - Problem description

The fluid size domain is specified at a certain distance, measured from the cylinder to the inlet and the outlet, it is equal to ten times the diameter of the cylinder in the direction of water flow, measured from the cylinder to the inlet. Also it is equal to twenty times the diameter of the cylinder in the direction of water flow measured from the cylinder to the outlet, and it is equal to five diameters in the other directions. The hydraulic system boundary condition is shown in table (2). The channel bed is flat, fixed and horizontal, while the sides of the channel are vertical.

Table 2 - Boundary conditions of hydraulic system

Inlet	Velocity at inlet
Outlet	Pressure at outlet
Cylinder	No slip-Wall
Vane	No slip-Wall
Channel bed	No slip-Wall
Channel sides	No slip-Wall
Top surface of flow	Atmospheric Pressure

The cylinder with the size equal to (10cm) is used to pose the present problem, also, the configuration of the submerged vane which is utilized in the present study has a rectangular shape. Figure 1 illustrates the shape of the submerged vane and the circular cylinder with its dimensions. The water flow velocity which is tackled in the present problem is equal to 0.04 m/sec and the Reynolds number is 3980. In the present simulation, different distances are considered, and these distances are: (0.25D, 0.5D, 0.75D, and D). The status of the direct touch found between the vane and the cylinder is also considered. The goal of the present numerical simulation refers to the investigation of a certain distance impact noticed between the vane and the cylinder on the flow field around the cylinder. The vane is located at the upstream region before the cylinder. So, the flow must pass the vane before it passes the cylinder; therefore, the turbulent flow intensity mechanism will suffer from the turbulence reduction, when the water flow encounters the vane. This effect will be reflected on the flow pattern in the zone around the cylinder, taking into consideration the distance effect. Here, it is realized that the path of the flow will be more sensitive to the vane presence as compared with the case of not using it. The numerical study includes the investigation of turbulent flow kinematic energy, intensity, velocity distribution in the direction of flow, velocity component which is perpendicular to the flow velocity and the tangential flow velocity, static pressure, pressure coefficient and eddy viscosity around the cylinder body. The study continues to investigate the water flow pattern around the cylinder at three different sections where the first section is located at the cylinder upstream and the second section is located in middle; the third section is located at the cylinder downstream. It is clear that the middle section location is between the upstream and the downstream of the cylinder. The performed analysis is unsteady for 8 seconds. The vane length is equal to the cylinder diameter.

3. GOVERNING EQUATIONS

Mass conservation:

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho\vec{v}) = 0 \tag{1}$$

Momentum equation:

$$\frac{\partial}{\partial t}(\rho\vec{v}) + \nabla.(\rho\vec{v}\vec{v}) = -\nabla P + \nabla.\bar{\bar{\tau}} + \rho\vec{g} + \vec{F} \tag{2}$$

The turbulence kinetic energy, k, and its dissipation rate, ε, are calculated from:

$$\frac{Dk}{Dt} = \frac{\partial}{\partial X_i}\left\{\left((\nu\delta_{jk} + c_s\frac{k}{\varepsilon}\bar{u}_k\bar{u}_j)\frac{\partial k}{\partial X_k}\right\} - \bar{u}_k\bar{u}_j\frac{\partial u_i}{\partial X_k} - \varepsilon\right. \tag{3}$$

$$\frac{D\varepsilon}{Dt} = \frac{\partial}{\partial X_i}\left\{\left((\nu\delta_{jk} + c_1\frac{k}{\varepsilon}\bar{u}_k\bar{u}_j)\frac{\partial \varepsilon}{\partial X_k}\right\} - \frac{\varepsilon}{k}c_2\bar{u}_k\bar{u}_j\frac{\partial u_i}{\partial X_k} - c_3\varepsilon\right. \tag{4}$$

Model constants: c_s, c_1, c_2, c_3 are 0.22, 0.18, 1.44 and 1.92, respectively [18].

4. FLUENT VERIFICATION

One of the significant subjects required in using any software is to make a verification to the software before using it in solving the current problem. It is very necessary to verify and make inspection to the fluent software before it is adopted to do the analysis of the current problem. Therefore, the fluent is employed to perform the analysis and comparison of the case study in reference [16]. Details of the case study can be found in reference [16]. It has been noticed that there are three dimensional analysis of the flow field around a square pier with basic conditions, shown in table 3, while table (4) refers to the obtained value of the drag coefficient from the study in reference [16] and the drag coefficient value from the present study.

Table 3 - Details of case study in reference [16]

D (m)	V (m/s)	H (m)	$\rho\ (kg/cu.m)$	Re (no unit)
1	0.292	2	1.225	20000

Table 4 - details of the present and previous study

Reference studies	Reynolds Number	Setup	C_d
Zaid et al. (16)	20000	Numerical	2.1951
Present study	20000	Numerical	2.15

5. INDEPENDENT MESH

In this part of the research, the effect of the number of elements on the most important properties of flow, which is the velocity of flow, is studied. Figure 2 shows, when the number of elements is greater than 30000, the velocity magnitude at three different sections will be constant. From that, in all the different cases of study, the number of elements greater than 30,000 were taken up.

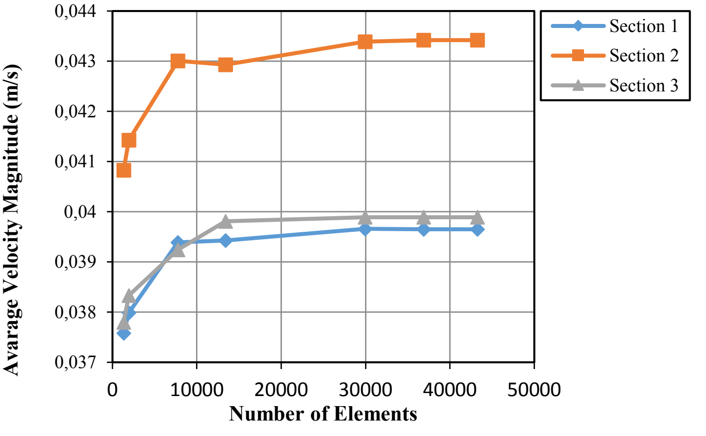
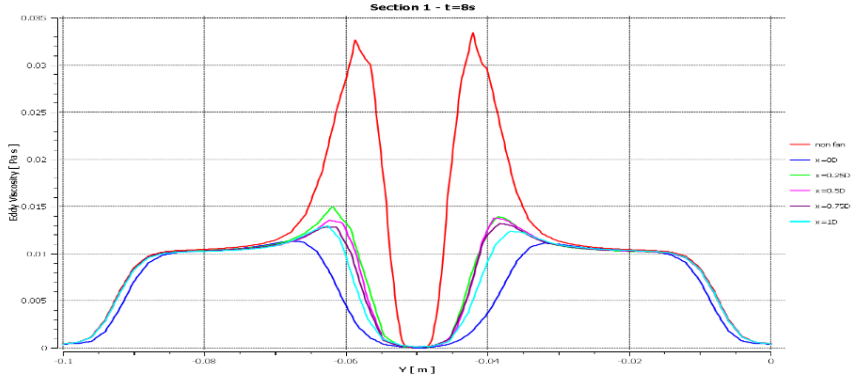


Fig. 2 - Independent mesh

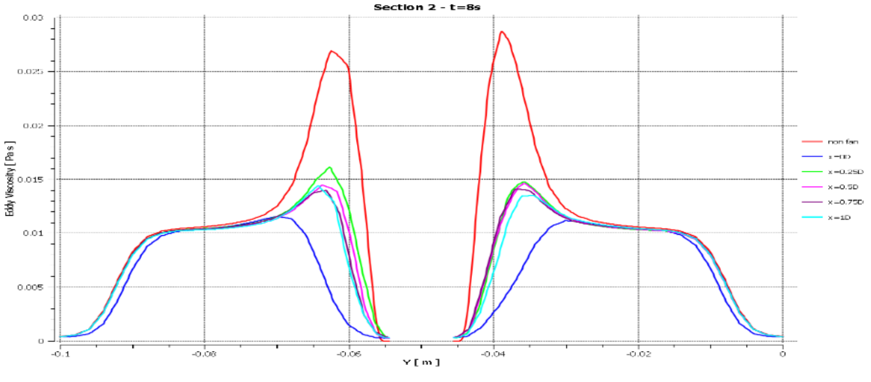
6. RESULTS AND DISCUSSIONS

The numerical analysis of the present problem has been carried out by using the volume method. It is shown in the following part of the paper. It implies:

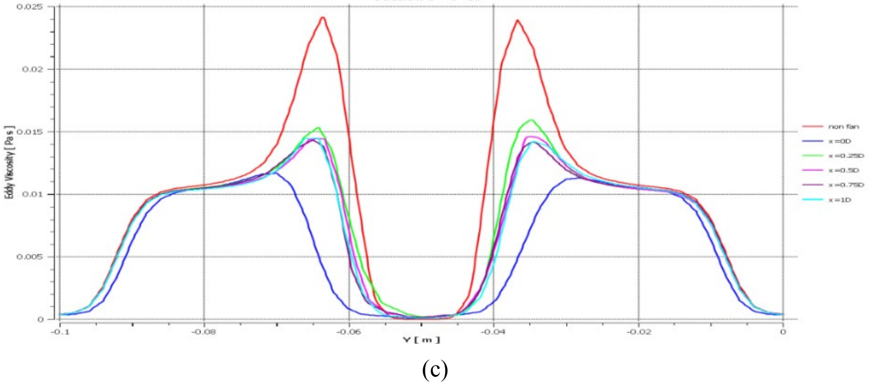
Eddy Impact: figure 3 illustrated the relationship between the eddy viscosity and the transverse distance for various sections (sec. 1, sec. 2 and sec. 3). The eddy viscosity is higher when the cylinder is alone as compared with the presence of vane that is in touch or neighboring to the cylinder. For the case when the vane is in direct touch with the cylinder, the reduction in the eddy viscosity is completely obvious. It is clear that when the vane moves at a distance from the cylinder, the eddy viscosity grows and develops but it always remains less than the case of when the cylinder is alone. The clarification of this fluctuation in the behavior occurs due to the flow separation and to the distance between the cylinder and the vane. The leading and trailing sharp corner of a rectangular vane works as a natural point of separation. Also, the attachment points along the vane vary with time and this will be reflected on the distribution of eddy viscosity. Whenever the distance between the vane and cylinder increase, the eddy viscosity increases due to the influence of spacing occurrence between the two neighboring structures. So the eddy will grow with rising in strength. Figures 4a and 4b show the eddy viscosity efficiency with vane position for the various sections. For both cases: (maximum and averaged), it is clear when the cylinder has a direct touch with vane, the efficiency is higher as compared with the case when the vane is located at the distance from the cylinder. Figure 5 illustrated the eddy viscosity contour of the cylinder alone and the vane with the cylinder considering the effect of distance found between the vane and the cylinder.



(a)

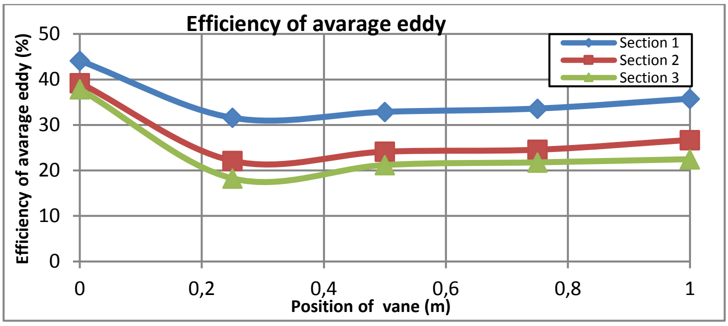


(b)

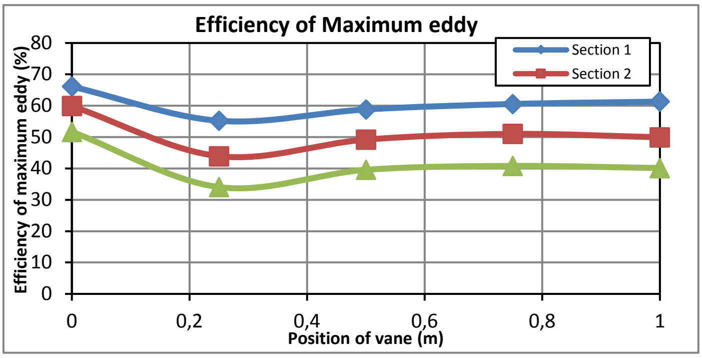


(c)

Fig. 3 - Eddy Viscosity (a) section 1, (b) section 2 and (c) section 3

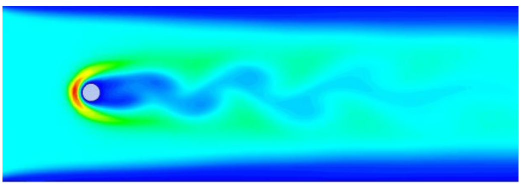


(a)

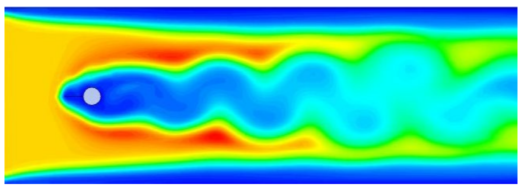


(b)

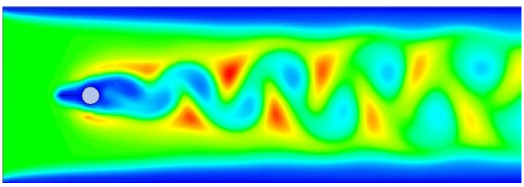
Fig. 4 - Eddy efficiency (a) efficiency of average eddy (b) efficiency of maximum eddy



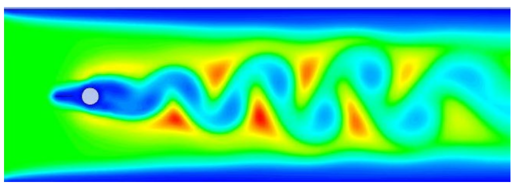
(a)



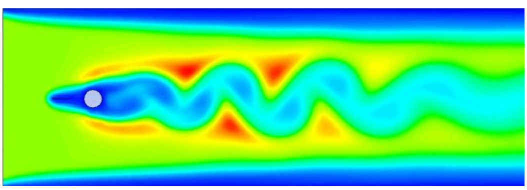
(b)



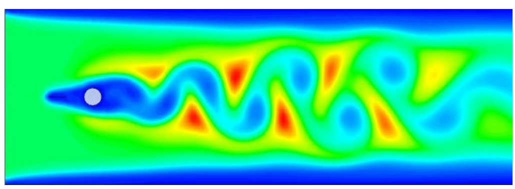
(c)



(d)



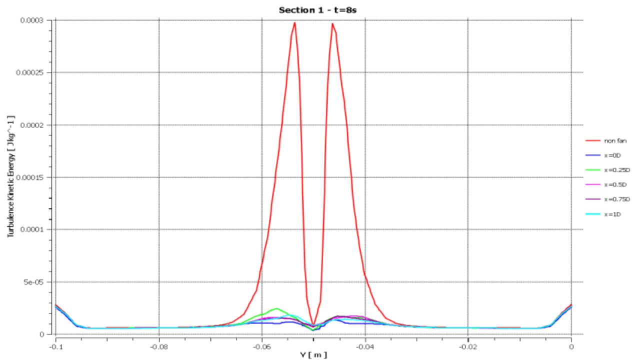
(e)



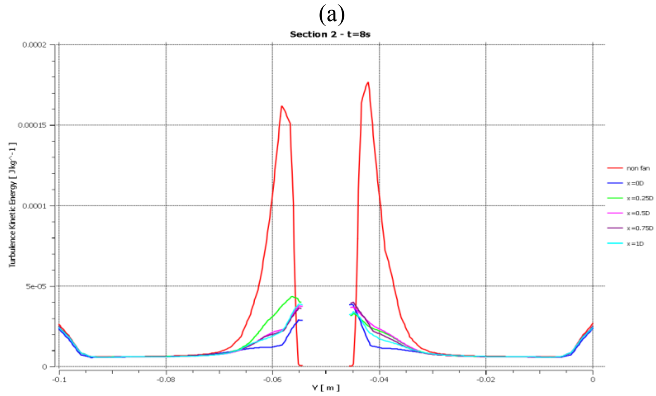
(f)

Fig. 5 - Eddy viscosity contour (a) non vane, (b) x=0D, (c) x=0.25D, (d) x=0.5D, (e) x=0.75D and (f) x=1D

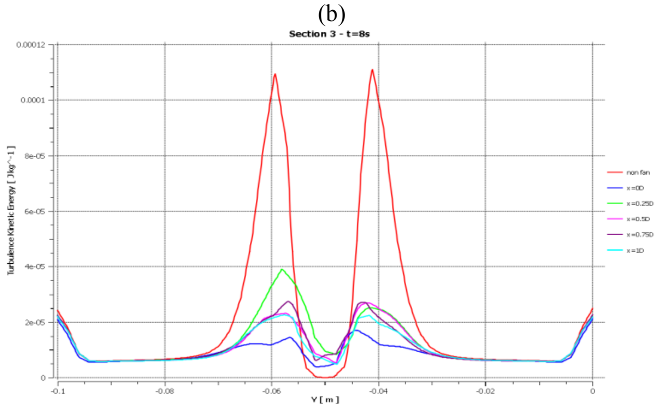
Turbulent Kinetic Energy: figure 6 illustrated the relationship between the turbulent kinetic energy and the transverse distance for various sections (sec. 1, sec. 2 and sec. 3). The turbulent kinetic energy is higher when the cylinder is alone as compared with the presence of vane in touch or neighboring to the cylinder. For the case when the vane is in direct touch with the cylinder, the decrease in the turbulent kinetic energy is completely obvious. It is also found that, when the vane moves at distance from the cylinder, the turbulent kinetic energy grows and develops but it always remains less than the case of when the cylinder is alone. The variation in results obtained will attribute or associate to the losses in the flow velocity, this will be reflected on the turbulent kinetic energy. When the water flow encounters the vane, the reduction in velocity will occur due to the friction force which grows and develops between the walls and the sharp corners of the vane with water flow which, in turn, will lead to the reduction in flow velocity. This reduction will be reflected on the turbulent kinetic energy. Figure 7 illustrated the turbulent kinetic energy contour of the cylinder alone and the vane with the cylinder, taking into consideration the effect of the distance found between the vane and the cylinder.



(a)



(b)



(c)

Fig. 6 - Turbulent Kinetic Energy (a) section 1, (b) section 2 and (c) section 3

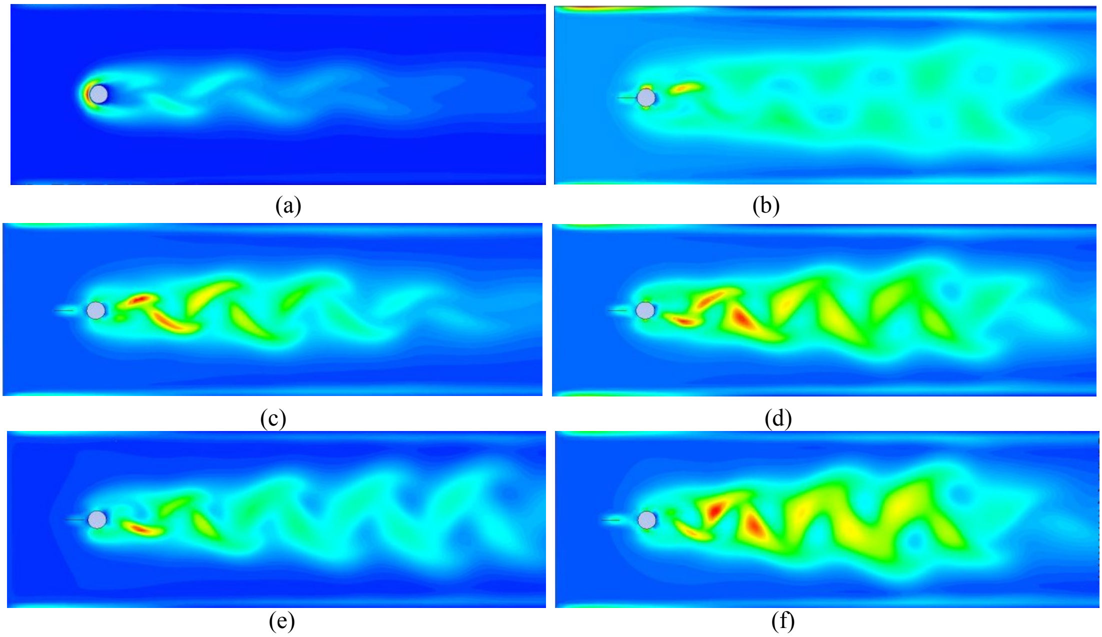
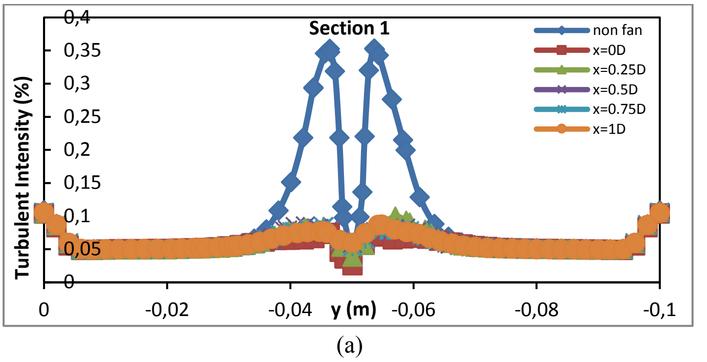
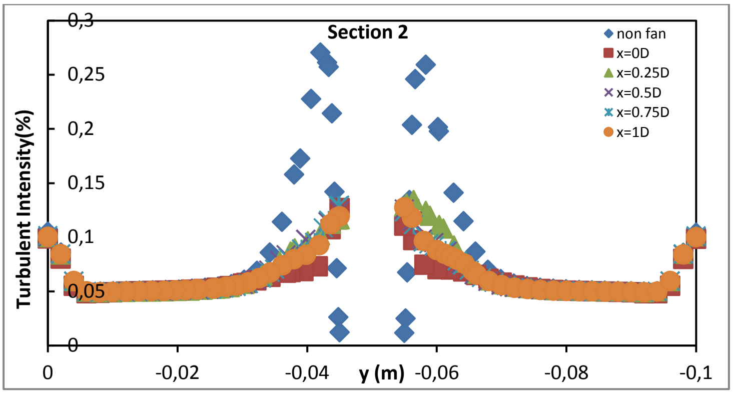


Fig. 7 - Turbulent kinetic energy contour (a) non vane, (b) x=0D, (c) x=0.25D, (d) x=0.5D, (e) x=0.75D and (f) x=1D

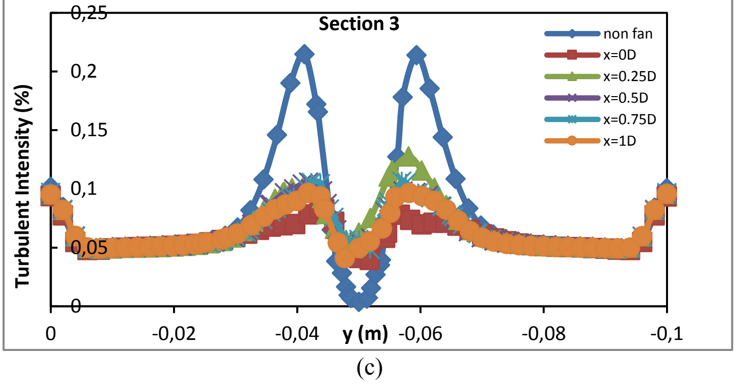
Fig. 8 - shows the distribution of the turbulent intensity with transverse distance for different sections (sec. 1, sce. 2 and sec. 3). The turbulent intensity of the flow pattern referred to the turbulence level around the cylinder and the cylinder with vane. Section 1 has the highest turbulent intensity as compared with the remainder sections. The turbulent intensity in case of the alone cylinder is higher as compared with the presence of the neighboring vane. This is due to the presence of the highest vortex in the zone near the alone cylinder, and the occurrence of the flow recirculation at this zone. Over all, for all sections, the presence of the vane leads to the sudden dramatic drop in the turbulent intensity with a noticeable variation in its values. The clarification of this dramatic drop is attributed to the flow separation which occurs at the vane entrance and continues to grow and develop till the end of the vane. When the water flow leaves the vane end it will suffer from turbulent kinetic energy magnitude shortages; in this case, there is no additional supply in the turbulent kinetic energy noticed. Therefore, the reduction in the turbulent intensity will occur. In general, the turbulent intensity has a direct relation with the flow velocity. As a result, any increase or decrease in the magnitude of the flow velocity will be reflected and appears clearly on the turbulent intensity. The presence of the vane leads to the decrease in flow velocity and this will be reflected on the turbulent intensity values.



(a)



(b)

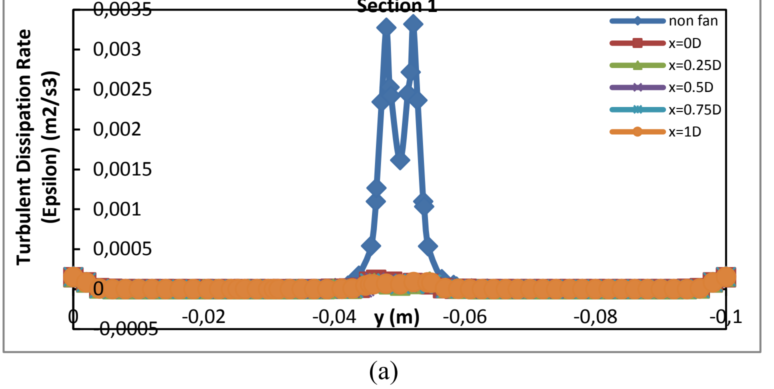


(c)

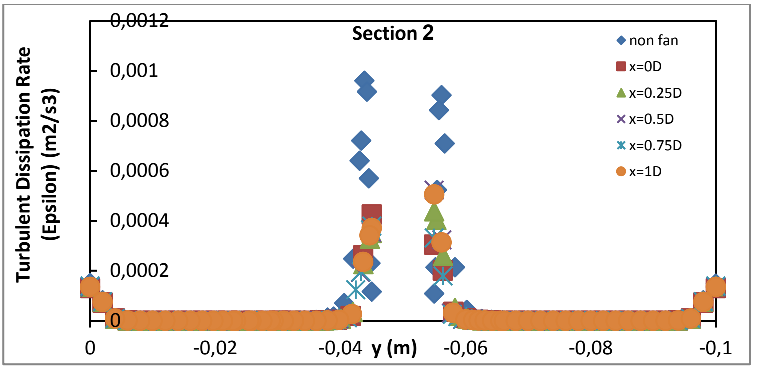
Fig. 8 - Turbulent intensity (a) section 1, (b) section 2 and (c) section 3

Turbulent Dissipation Rate Impact

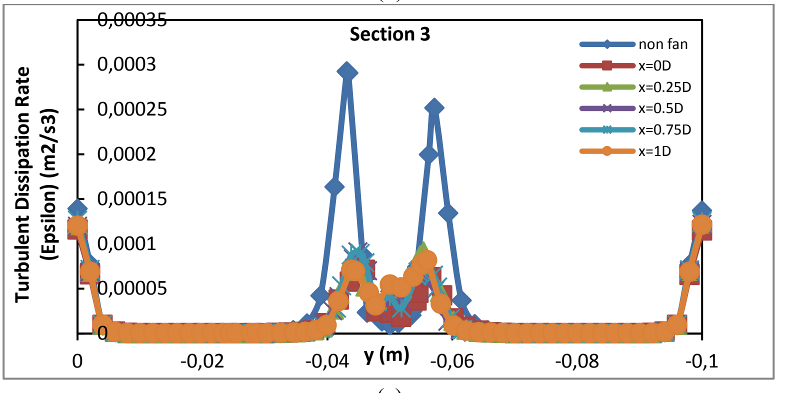
Figure 9 shows the distribution of the turbulent dissipation rate with the transverse distance for different sections (sec. 1, sec. 2 and sec. 3). Section.1 has the highest turbulent dissipation rate as compared with the remainder sections. The turbulent dissipation rate referred to the formation of eddies in a fluid flow. Here, the energy of turbulence was absorbed by breaking the large eddies into smaller and smaller ones until it reaches the least energy or vanishes. In the case of alone cylinder, the strong formation of eddies occurs in the upstream zone and this will lead to the drop in the turbulent dissipation rate, while in the case of a vane neighboring to the cylinder, here the vane continuously destroys the eddies and this will ultimately lead to the dramatic rise in the turbulent dissipation rate regardless of the certain distance found between the vane and the cylinder.



(a)



(b)



(c)

Fig. 9 - Turbulent Dissipation Rate (a) section 1, (b) section 2 and (c) section 3.

Velocity profile impact

Figure 10 shows the distribution of the flow velocity in the x-direction at different sections. It is clear from the figure that the transverse sections of the flow velocity are similar or identical to each other in their behavior regardless of the presence of a neighboring vane or/and distance between the vane and the cylinder.

Figure 11 shows the flow velocity contours. The figure shows that the flow crosses the alone cylinder or the cylinder with the neighboring vane.

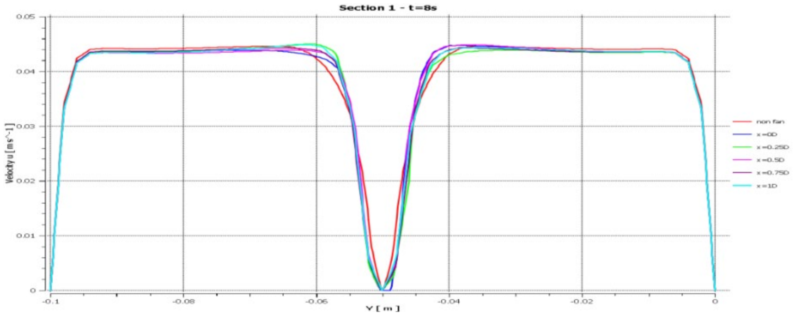
It also shows the turbulence and eddy increasing gradually regardless of the position of the vane from the cylinder and this means that the flow velocity after the downstream zone of the cylinder (section. 3) will increase suddenly.

Figure 12 shows the distribution of the flow velocity which is perpendicular to the flow velocity in the x-direction at different sections.

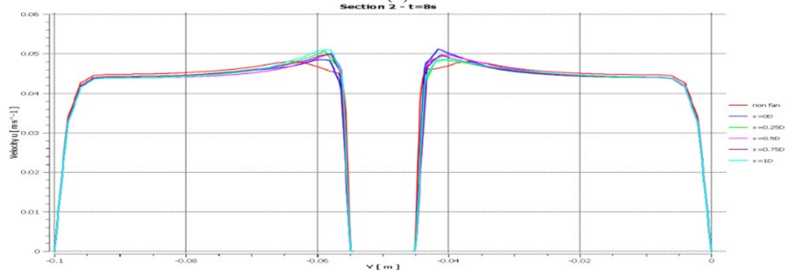
It is evident from the figure that the flow velocity profile will changes from section to section. This change depends mainly on the magnitude of the flow velocity in the x-direction and the formation of the eddies at the upstream zone.

But for the same section, the behavior is identical regardless of the presence of vane and the specified distance found between the vane and the cylinder. Figure 13 shows the distribution of tangential velocity at different sections.

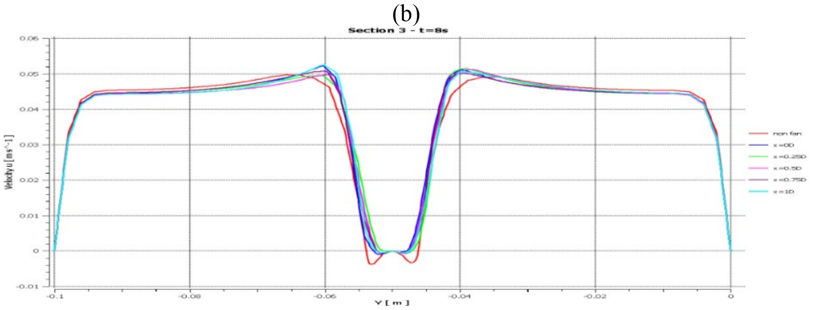
Here, it is clear from the figure that the transverse sections of tangential flow velocity are similar or identical to each other in the behavior regardless of the presence of neighboring vane or/and the distance found between the vane and the cylinder.



(a)

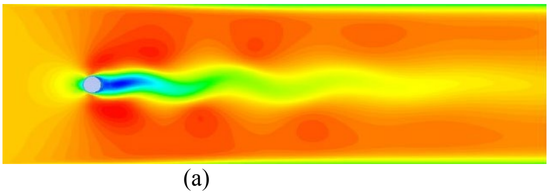


(b)

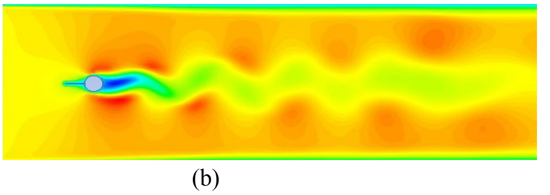


(c)

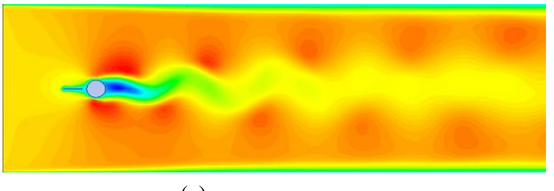
Fig. 10 - Velocity in the x-direction (a) section 1, (b) section 2 and (c) section 3



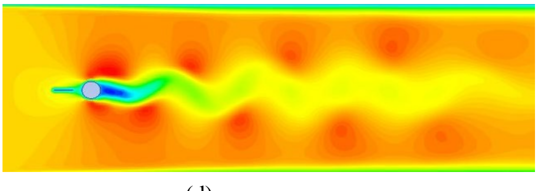
(a)



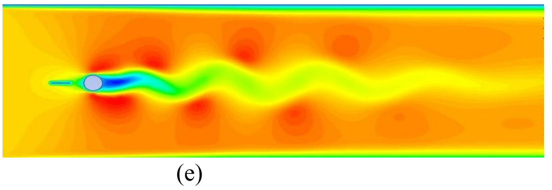
(b)



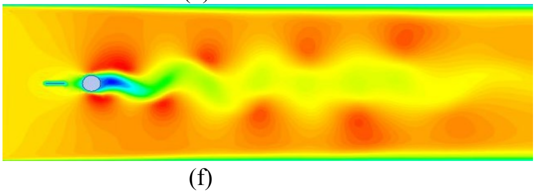
(c)



(d)

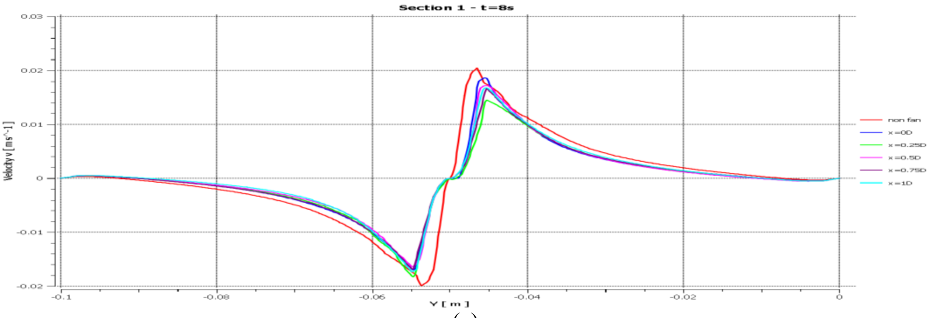


(e)

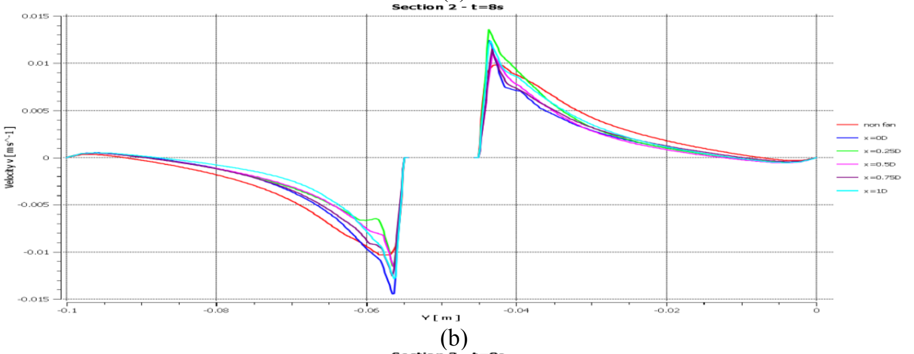


(f)

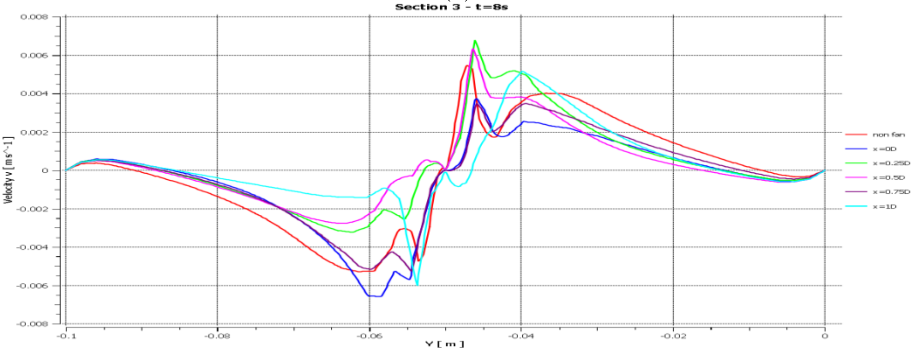
Fig. 11 - Velocity in the x-direction contour (a) non vane, (b) x=0D, (c) x=0.25D, (d) x=0.5D, (e) x=0.75D and (f) x=1D



(a)

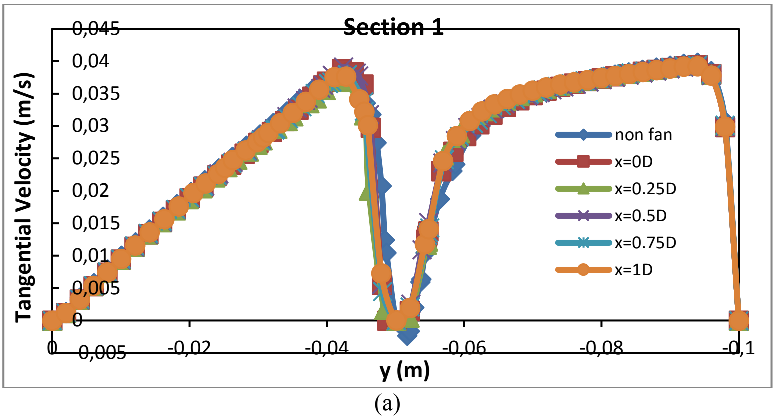


(b)

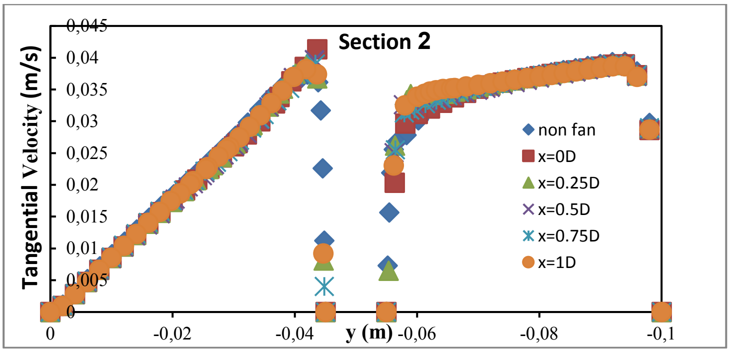


(c)

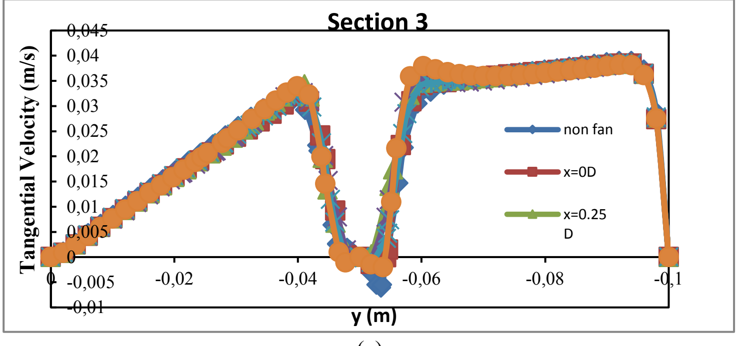
Fig. 12 - Velocity in the y-direction (a) section 1, (b) section 2 and (c) section 3



(a)



(b)

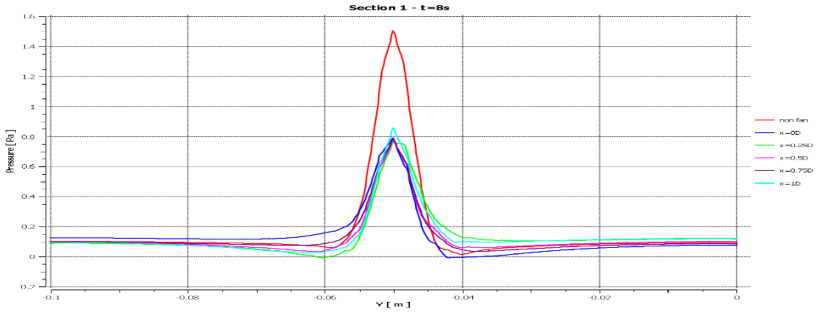


(c)

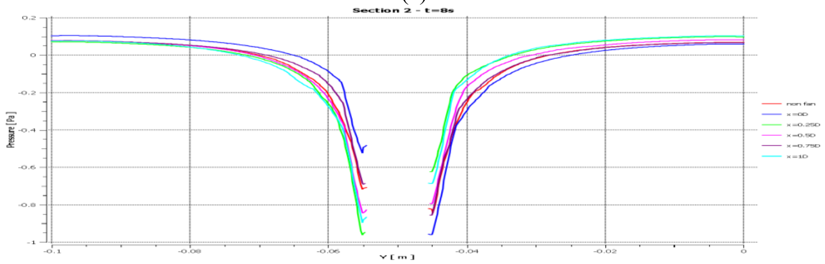
Fig. 13 - Tangential Velocity (a) section 1, (b) section 2 and (c) section 3

Static pressure impact

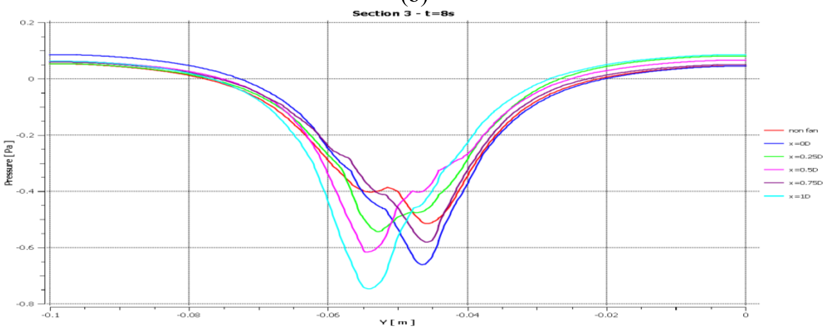
Figure 14 illustrated the static pressure distribution which has resulted from flow field around the circular cylinder at the different sections. At the upstream section, it is obvious that the maximum value of pressure occurs when the cylinder is alone where the pressure often has a maximum value on the front face of the cylinder, while in the case of a vane neighboring to the cylinder, the reduction in the pressure occurs due to the occurrence of the separation in flow and the loss which occurs on the flow velocity when the flow passes the vane. Consequently, all these previous reasons will have a certain reflection on the pressure. It is clear that the pressure at the upstream section has positive values while for the middle section and downstream section the pressure sign alters from the positive to the negative values regardless of the presence of vane and the distance found between the vane and the cylinder. For sections 2 and 3 it appears that the specified distance between the vane and the cylinder increases; the negative value of the pressure also increase as compared with the pressure value when the cylinder is alone. The variation in the pressure at sections 2 and 3 is related to the high separation of the flow which occurs at the vane before it crosses the cylinder. Figure 15 shows the contour map of the static pressure around the alone cylinder and the cylinder with a neighboring vane. The figure reflects the fluctuation in the distribution of the flow path, especially after the cylinder downstream zone. The contour indicated the maximum pressure that occurred at upstream. Also, it showed the stagnation zone due to the influence of the obstacle presence and shows approximately a periodic trend in the profile. Figure 16 illustrated the pressure coefficient distribution with the transverse sections. The pressure coefficient has positive values at upstream section and changes from the positive to the negative values at the middle and downstream sections regardless of the presence of the vane and the distance found between the vane and the cylinder. This occurs due to the effect of the flow separation and velocity losses when the flow crosses the vane.



(a)



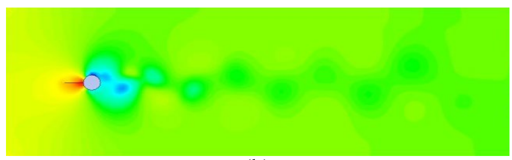
(b)



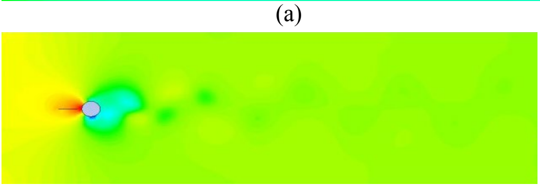
(c)

Fig. 14 - Static pressure (a) section 1, (b) section 2 and (c) section 3

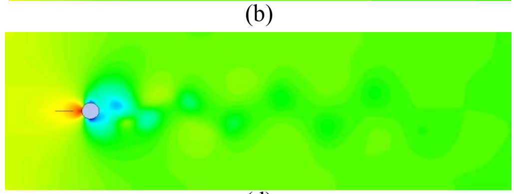
(a)



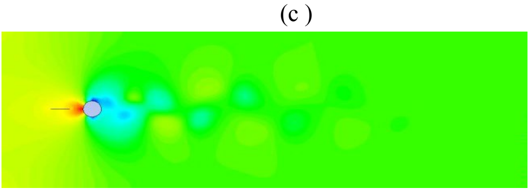
(b)



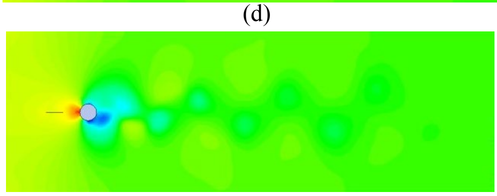
(c)



(d)

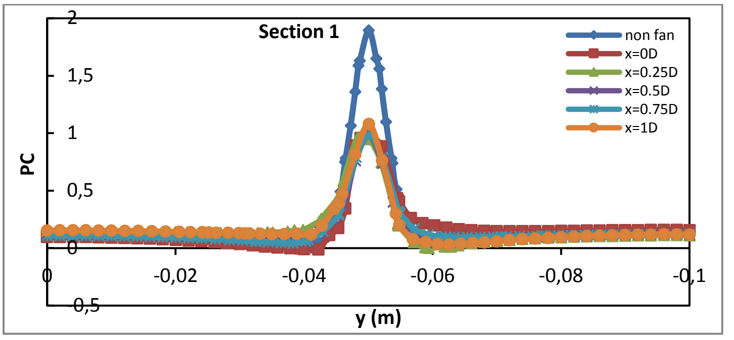


(e)

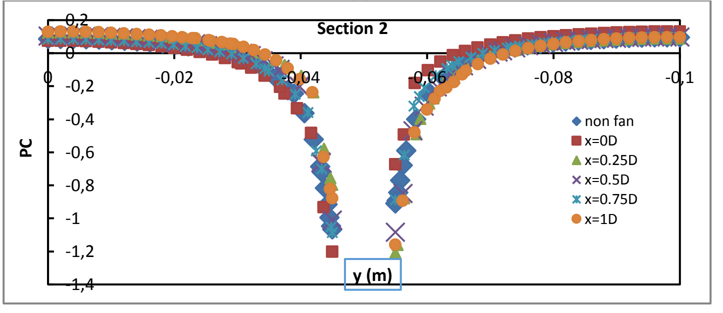


(f)

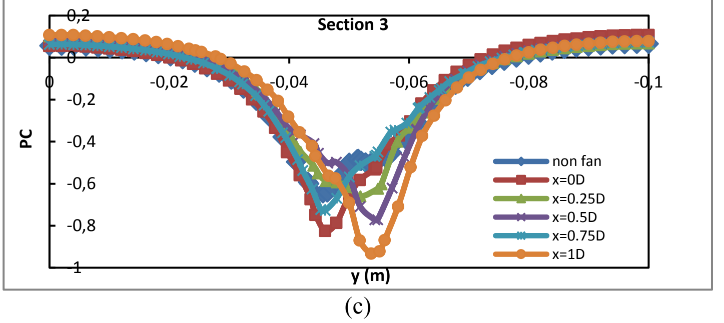
Fig. 15 - Static pressure in the x-direction contour (a) non vane, (b) x=0D, (c) x=0.25D, (d) x=0.5D, (e) x=0.75D and (f) x=1D



(a)



(b)



(c)

Fig. 16 - Pressure coefficient (a) section 1, (b) section 2 and (c) section 3

7. CONCLUSIONS

The major noticeable points which were inferred from the present study are listed below:

1. The vane has a sensitive significant vital role in increasing the flow separation. Also, it decreases the following hydraulic characteristics, like turbulent intensity, turbulent kinetic energy, turbulent dissipation rate, eddy viscosity, flow velocity, static pressure and flow recirculation.
2. Whenever the vane has a direct contact with the cylinder, a dramatic diminish will occur in the hydraulic variables which dominates the flow field around the cylinder.
3. The specified distance found between the vane and the cylinder has a major impact on the hydraulic variables which dominate the flow field around the cylinder, but the obtained results are always less than the result obtained when the vane has a direct contact with the cylinder.
4. The vane location has a major and reasonable effect on the result which is prevalent in the flow field around the cylinder as compared with the alone cylinder.

5. The vane has a major impact on the reduction and fluctuation in the flow velocity owing/due to the loss which occurs when the flow crosses the vane.
6. The vane represents a feasible way to reduce and control the turbulent intensity.
7. The hydraulic characteristics of flow field around the cylinder alter with the location of the transverse section.
8. The hydraulic variables at the upstream section of the cylinder are always higher than the other sections.
9. The vane is a good way in destroying the large eddies into smaller ones which will be reflected on the flow that crosses the cylinder.

REFERENCES

[1] A. Roshko, *Experiments on the flow past a circular cylinder at very high Reynolds number,* California Institute of technology, California – USA, 1960.

[2] R. Mittal, *Large-eddy simulation of flow past a circular cylinder,* Center for Turbulence Research, Kanpur – India, 1995.

[3] S. Dey, S. K. Bose and G. L. N. Sastry, Clear water scour at circular piers: a model, ASCE, *Journal of Hydraulic Engineering*, **121**(12), 869-876, 1995.

[4] M. Breuer, A challenging test case for large eddy simulation: high Reynolds number circular cylinder flow, *Int. J. Heat Fluid Flow.*, **21**, 648-654, 2000.

[5] T. M. Salaheldin, J. Imran, M. H. Chaudhry, Numerical modeling of three-dimensional flow field around circular piers, ASCE, *Journal of Hydraulic Engineering*, **130**(2):91-100, 2004.

[6] L.-P. Li, L. Cui, Prediction of maximum scour depth around large-diameter cylinder under the effects of both wave and current, *Journal of Hydrodynamics, Ser. B*, **17**(1): 73-79, 2005.

[7] M. Zhao, B. Teng, L.-p. Li, Local scour around a large-scale vertical circular cylinder due to combined wave-current action, *Journal of Hydrodynamics, Ser. B*, **17**(4): 344-351, 2005.

[8] W. Zhao, A. Huhe, Large- eddy simulation of three-dimensional turbulent flow around a circular pier, *Journal of Hydrodynamics, Ser. B*, **18**(6): 765-772, 2006.

[9] K. Marakkos, J. T. Turner, Vortex generation in the cross-flow around a cylinder attached to an end-wall, *Optics & Laser Technology*, **38**: 277-285, 2006.

[10] W. Zhao and A. Huhe, Numerical study of the turbulent flow around a circular pier, *Journal of Shanghai University (English Edition)*, **11**(1):17-21, 2007.

[11] B. N. Rajani, A. Kandasamy, S. Majumdar, Numerical simulation of laminar flow past a circular cylinder, *Applied Mathematical Modelling*, **33**, 1228-1247, 2009.

[12] F. Gao, D. McGovern, C. G. Mingham, S. Ilic and D. M. Causon, Numerical and experimental investigation of turbulent flow around a vertical circular cylinder, *Int. Society of Offshore and Polar Eng. (ISOPE)*, 2010.

[13] M. Tchawe, B. Djeumako, C. Koueni-Toko, D. Tcheukam-Toko and A. Kuitche, Study of dynamic field around a vertical circular cylinder placed in an open-channel flow, *International Journal of Innovative Science, Engineering & Technology*, Vol. **2**, Issue 2, February 2015.

[14] S. H. Azizi, D. Farsadizadeh, H. Arvanaghi, A. Abbaspour, Numerical simulation of flow pattern around the bridge pier with submerged vanes, *Journal of Hydraulic Structures*, **2**(2): 46-61, 2016.

[15] M. Al-Saffar, Flow characteristics around wall-mounted circular cylinder above the horseshoe vortex region in sub-critical, critical, and super-critical flow conditions, *Int. J. of Hydraulic Eng.*, **7**(1):1-10, 2018.

[16] M. Zaid, Z. Yazdanfar, H. Chowdhury, F. Alam, Numerical modeling of flow around a pier mounted in a flat and fixed bed, *Energy Procedia*, **160**:51-59, 2019.

[17] I.A. Abdulhussein, K. Al-Asadi, R.M. Qasim, Pier scouring reduction using a strip guide flow panel device, *RUDN Journal of engineering researches*, **20**(3):229-235, 2019.

[18] S. F. Dahkil, T. A. Gabbar and D. K. Jaber, Numerical Study of The Initial Pressure and Diameters Ratio Effect On The Jet Ejector Performance, *Basrah Journal for Engineering Science*, Vol. **14**, Issue 1, Pages 122-135, 2014.