

Drag Reduction of Circular Cylinder using Square Disturbance Body at 60° Angle by 2D Numerical Simulation Unsteady-RANS

Rina ^{1,*}, Meiki Eru Putra ¹, Nofriyandi. R ¹, Ruzita Sumiati ²

¹Department of Mechanical Engineering, Dharma Andalas University
Jl. Sawahan No. 103A, Padang, Indonesia

²Department of Mechanical Engineering, Politeknik Negeri Padang
Kampus Limau Manis, Padang, Indonesia

Received 12 October 2016; Revised 22 October 2016; Accepted 23 November 2016, Published 30 November 2016
<http://dx.doi.org/10.22216/JoD.2016.V1.88-95>

Academic Editor: Asmara Yanto (asmarayanto@yahoo.com)

*Correspondence should be addressed to rina.mesin@gmail.com

Copyright © 2016 Rina. This is an open access article distributed under the [Creative Commons Attribution License](https://creativecommons.org/licenses/by/4.0/).

Abstract

Drag is an aerodynamic force that appears when the flow past the bluff body circular cylinder. Drag strongly influenced by the flow separation point. One of the ways reducing the drag force that is to control the flow by placing the disturbance body on the upstream side at a certain angle. Previous research has found at 60° angle of flow separation is faster than a single cylinder that produced greater drag. Therefore, this research was conducted to reduce the drag force on the corner with disturbance dimension variation. This research was carried out numerically using a FLUENT 6.3.26 CFD software in 2D unsteady viscous-RANS models Turbulence Model-Shear-Stress Transport (SST) $k-\omega$ in a narrow channel. The geometry is simulated in a circular cylinder as the main body and the square cylinder as a disturbance body being placed in front of the main body by s/D ratio. *Dimensions of disturbance body varied at (s) 0,1; 0,2; 0,3; 0,4 dan 0,5 mm with a gap ($\delta=0,4mm$).* Reynolds number based on the diameter of the cylinder, ie $Re_D 2,32 \times 10^4$. The simulation results show that the transition flow on shifting 60° SDB angle for all SDB dimensional variations do not produce turbulent. The optimum condition for the drag force reduction is $s/D = 0.008$ about 48 %.

Keywords: bluff body square cylinder, Shear-Stress-Transport (SST) $k-\omega$, drag coefficient pressure, narrow channel, drag force

1. Introduction

Research about characteristic of flow around bluff body has been developed by boundary layer philosophy. The main purpose of the study is to reduce the drag force. The drag is force of aerodynamics that rises due the friction. To reduction of this drag force, contribution of the boundary layer is important because it is associated with shear stress. Shear stress on the body surface is affected by pressure distribution that occurs on the body contour.

Control of flow is one of way to see how the phenomenon of flow passed bluff body to reduced the drag force. The material will be saved with reduce drag force, therefore this research is very contribution in technology

advances. The body geometry shape that often used is a circular cylinder. The fluid through circular cylinder can applicated at bridges, chimneys, piping systems, cooling towers, poles, etc.

The curved surface contour of the circular cylinder has big Adverse Pressure Gradient (APG) characteristics. The flow pressure on the surface of the body occurs cause of the influence of the highly dominant drag pressure on the cylindrical circular geometry, hence the pressure on the surface of this body is high, and the transition of laminar boundary layer to turbulent boundary layer will be fasten.

The research about reduction of drag has been done by Tsutsui and Igarashi [1] and

Zhang [2] with placed disturbance body at the upstream side in tandem with variation of longitudinal distance between upstream disturbance body and main circular cylinder (L/D), diameter ratio upstream disturbance body and main circular cylinder (d/D) and Reynold Number (Re). Boundary layer that separation from surface contour of disturbance body will form a free shear layer which produces discrete vortices and touch surface of main body. The free shear layer then interacts with the layer on the back. This makes the laminar boundary transition to be turbulent in the main cylinder to be faster so that massive separation is delayed backward, consequently the drag force can be reduced.

Alam, et al [3], Putra [4], and Rina [5] used two disturbance body with varying the certain of angle. Alam, et al using circular cylinder as disturbance body. Putra and Rina using square cylinder as disturbance body experimentally and numerically respectively. The information can we take from their research is that addition of disturbance body on upstream side at certain of angle giving influence to reduce of the drag.

Alam, et al [3], Son [4] and Rina [5], using 2 disturbance body by varying the specific angle. Alam et al uses a circular cylinder as a disturbance body. Putra and Rina using a square cylinder such as a disturbance body experimentally and numerically respectively. The result showed that the addition of disturbance bodies on the upstream side on the specific angle influence to reduce of the drag. The optimum value in this case is at the angle $\alpha = 30^\circ$. This is because at that angle the massive separation is delayed very significant, and flow reattachment to the main cylinder resulting from the addition of body disorders. At the angle $\alpha = 60^\circ$ not contributing in reduced the drag force. The flow after passing through disturbance body separate from main body without deflected back to main bluff body. Further, Bell [6] and Weidman [7] have experienced bluff body ratio effects with channel width (blockage effect) on fluid speed and the drag coefficient. The free stream velocity will increase due the blockage effect. The result showed that the greater the blockage effect then the drag (C_d) will increase. Free stream velocity in the wind tunnel wall increased caused by solid blockage which also affect the wake growth.

The increased turbulence intensity from the free shear layers that separate from disturbance body and then attach on contour of main cylinder in free stream field have presented

numerically in more detail by Rina [5] using software FLUENT 6.3.26. The result showed that at the angle 60 degree, the drag force is greater than the drag force of single cylinder. Therefore, this research need developed furthermore with various dimension of disturbance body (smaller size from previous studies) at the angle of 60 degree.

2. Computational Details

Computational Fluid Dynamic (CFD) is an analysis system involving fluid flow, momentum, heat transfer and related phenomena such as chemical reaction by simulation of computer [8]. Turbulent flows are strongly influenced by the turbulence modeling used in numerical analysis, such as RANS, URANS, LES, DES, boundary conditions, initial conditions, and grid independence shapes, especially in areas near the wall (solid surface) [9]. The turbulence model used for flow analysis around a circular cylinder is Shear-Stress-Transport (SST) $k-\omega$. The model was developed by Menter, which is a combination of $k-\omega$ capability for near-wall flow with free-stream independence of $k-\epsilon$ models in far field. Hence, the turbulence model is more accurate and more reliable than the standard $k-\omega$ for a wider flow type including adverse pressure gradient flow [10].

This research was conducted numerically using CFD software FLUENT 6.3.26 in 2D Unsteady-RANS with viscous model of Shear-Stress-Transport (SST) $k-\omega$ in narrow channel. Simulated geometric shape is circular cylinder as main body and square cylinder as disturbance body placed in front of main body with ratio s / D 0,16. Dimension of disturbance body varied (s) 0.1; 0.2; 0.3; 0.4 and 0.5 mm with the gap between an inlet disturbance body and a circular cylinder was a constant ($\delta = 0.4\text{mm}$). Determination of Reynolds number was based on diameter of cylinder; $ReD = 2.32 \times 10^4$.

The schematic square disturbance body (SDB) and main circular cylinder shown in Fig. 1 and the numerical simulation domain was shown in Fig. 2. The boundary conditions on the inlet side was defined as inlet velocity and on the outlet side was pressure outlets.

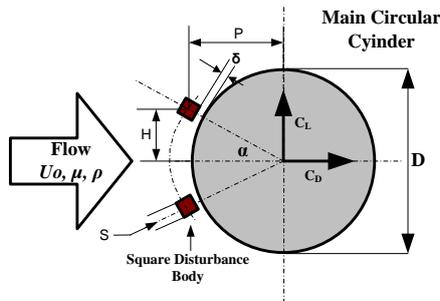


Figure 1. Schematic square disturbance body (SDB) and main circular cylinder

Quantitative data in the form of data graph were pressure coefficient (C_p), velocity profile (u / U_{max}), turbulence intensity (TI), shape factor (H), and pressure drag coefficient (C_{dp}). And the qualitative data in the form of flow visualization were pathline and velocity vector. The pressure coefficient is a quantitative data which is a static pressure divided by dynamic pressure. The equations can be expressed as follows:

$$C_p = (\rho_c - \rho_\infty) / \frac{1}{2} \rho U_\infty^2 \quad (1)$$

Where p_c is the pressure on the circular cylinder contour, P_∞ is the static pressure on the free-stream, and is the dynamic pressure in the free-stream. The pressure drag coefficient (C_{dp}) is obtained by integrating the pressure coefficient (C_p) of the cylindrical surface contour. The equations can be expressed as follows:

$$C_{dp} = \frac{1}{2} \int_0^{2\pi} C_p(\theta) \cos(\theta) d\theta \quad (2)$$

To obtain the value of drag pressure coefficient (C_{dp}) and the total drag coefficient (CDT) can be solved by numerical method of Simpson rule 1/3 double segment formulated equations can be expressed as follows:

$$I \cong (b-a) \frac{f(x_0) + 4 \sum_{i=1,3,5}^{n-1} f(x_i) + 2 \sum_{j=2,4,6}^{n-2} f(x_j) + f(x_n)}{3n} \quad (3)$$

Where:

- $b = 2\pi$ and $a = 0$, $f(x_0) = C_p(0) \cos(0)$ and $f(x_n) = C_p(2\pi) \cos(2\pi)$ to solve equation (3.3).
- $f(x_i)$ is the multiplication of gasal data function, where $i = 1, 3, 5 \dots n-1$.
- $f(x_j)$ is the multiplication of the even data

function, where $j = 2, 4, 6 \dots n-2$.

- $n =$ number of data.

The intensity of turbulence (TI) can be evaluated by using the formula:

$$TI = \frac{U'}{U} \times 100\% \quad (4)$$

$$U' = \delta = \sqrt{\frac{\sum (\bar{U} - U_n)^2}{n}} \quad (5)$$

While the shape factor is a comparison between displacement thickness (δ^*) with momentum thickness (θ^*).

$$H = \frac{\delta^*}{\theta^*} \quad (6)$$

Both quantitative and qualitative post-processing results would be complement each other to explain the flow phenomenon of the circular cylinder by the addition of square disturbance body.

3. Results and Discussion

3.1 Distribution of Pressure Coefficient (C_p)

The phenomena that occur along the cylindrical surface contours from the point of stagnation until finally the massive alert would be analyzed from the graph of the pressure coefficient shown in Fig. 3. From the picture, it can be seen that acceleration occurs in the increasing C_p with the addition of dimension of disturbance body. Consequently, the drag force becomes increasing. The maximum acceleration occurs until s/D 0.016 ($s=0.4$ mm).

The maximum acceleration of s/D 0.02, began to decrease. This happens because the flow from the free stream can not drive flow of separation from main body to re-attach. Consequently, it occurred separation of massive around the angle of 84 degree rapidly.

3.2 Distribution of Velocity Profil and Shape Factor (H)

The flat velocity profile and range shape factor value (H) 1.3 – 2.5 indicates a turbulent flow. Separation of flow is determined from shape factor value (H) 2.6 – 3.5. The field of adverse pressure gradient have shape factor value ($H > 3.5$).

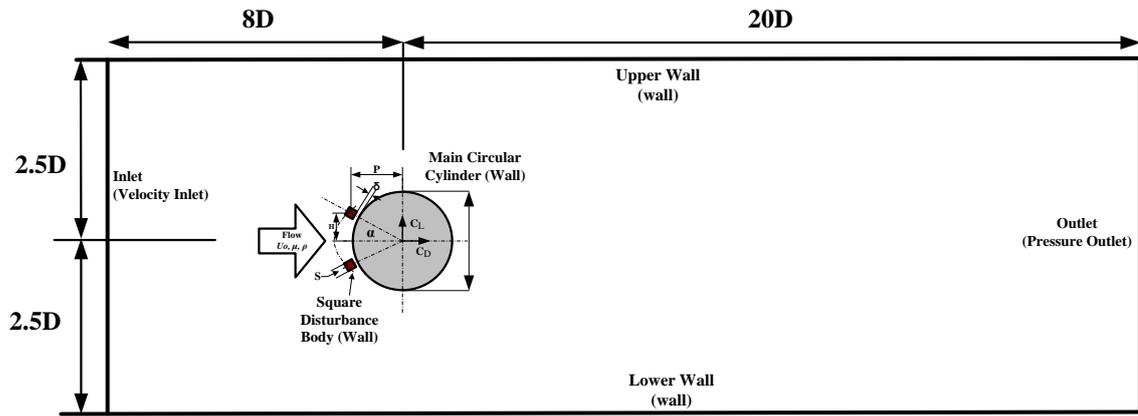


Figure 2. Numerical simulation domain

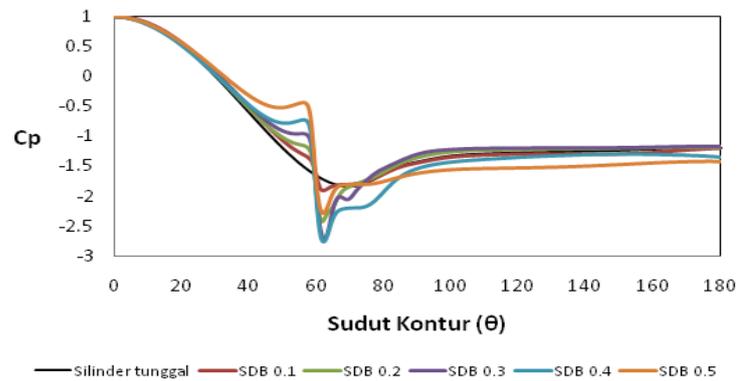


Figure 3. Pressure coefficient with variation of s/D

When it viewed from shape factor value on the table 1, at a single cylinder and cylinder with dimension of SDB of 0.1 mm, no transition of the flow from laminar to turbulent. With the result that separation of flow occurred faster. The transition of the flow occurred at the cylinder with dimensions of SDB of 0.1, 0.2, 0.4 and 0.5 mm are at the flow of contour $\theta = 75^\circ$, although the transitions were not until turbulent but it showed separation. This is because the shape factor value has exceeded of 3.5. It means that the field it has been in the adverse pressure region. Overall for all configurations, the flow was not transition until turbulent. This is because the phenomenon that occurs in all configuration showed not additional momentum from the flow of free stream so that directly separation.

Table 1. Shape Factor on contour of cylinder surface

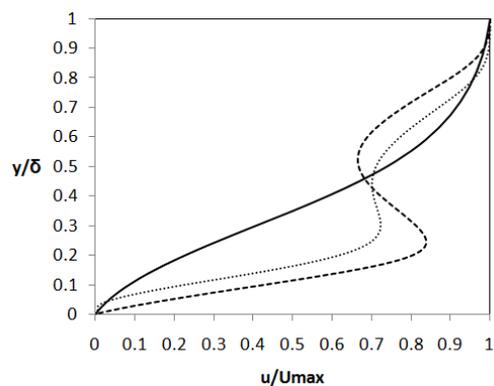
Variasi SDB (mm)	Shape Factor (H)		
	65°	75°	85°
Silinder Tunggal	2.53	3.05	6.98
SDB 0.1	6.45	7.03	12.50
SDB 0.2	10.75	6.58	9.07
SDB 0.3	12.87	8.20	6.97
SDB 0.4	9.33	8.14	9.16
SDB 0.5	10.86	9.77	10.63

Distribution of velocity profile of cylinder with variations of SDB given in fig. 4 for the present investigation is about increasing fluctuation of the flow, of about $0 < y/\delta < 0,2$ at angle of contour 65 degree. This position is very close in behind the disturber where the area has a fluctuating velocity.

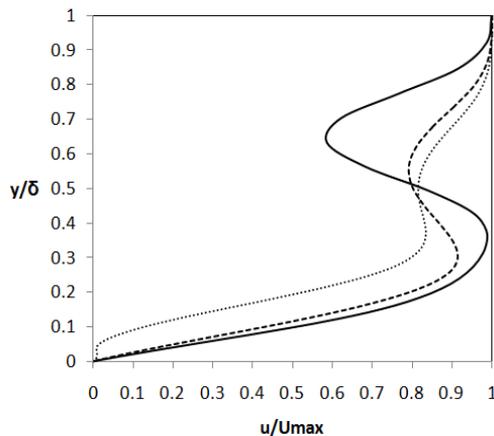
3.3 Turbulent Intensity

Fig. 5 shows the ratio of contour of turbulent intensity at each configuration. Red indicates the highest percentage of turbulent intensity while blue indicates the lowest percentage. From the picture, it can be seen that the greater Square Disturbance Body (SDB) dimension, will be the greater fluctuation of the flow because of the fluctuating speed behind the disturbance body (random flow), and highest turbulent intensity is in the wake area behind the disturbance body i.e the cylinder configuration using a disturbance of 0.5 mm. This seen from the wake value formed behind the upstream cylinder. Fig. 6 shows distribution of turbulent intensity. At $0.8 < y/\delta < 1$, it be seen that the placement of the two disturbance body with increasing dimensions has high turbulent intensity. This statement is supported by visualizing the contours of turbulent intensity in Fig. 5.

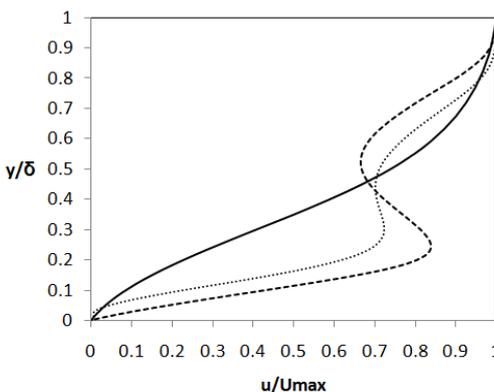
The larger the dimension of the disturbance body then will be formed the sharper angle at square cylinder, it produces higher energy content and turbulence intensity. This is because high fluctuations are generated behind the disturbance body and the shear layer that is deflected from the disturbance body can't again attach to the main cylinder surface. As a result, wake that formed behind the main cylinder is wider and drag becomes greater. This is in accordance with research that has been done by Rina [5].



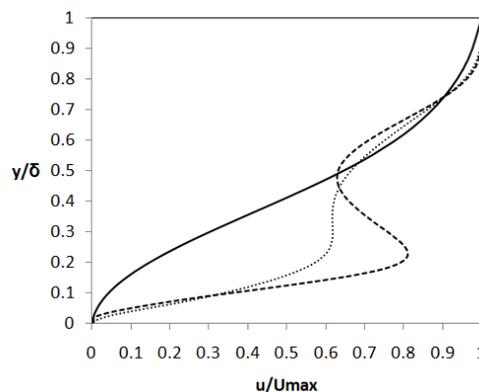
a



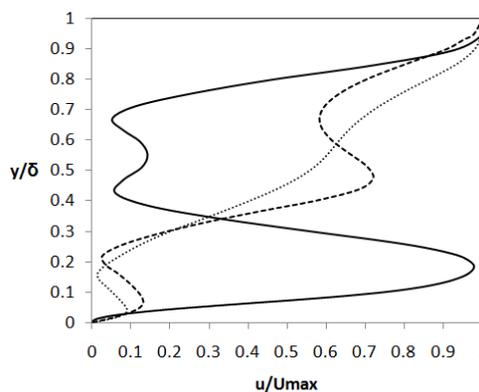
b



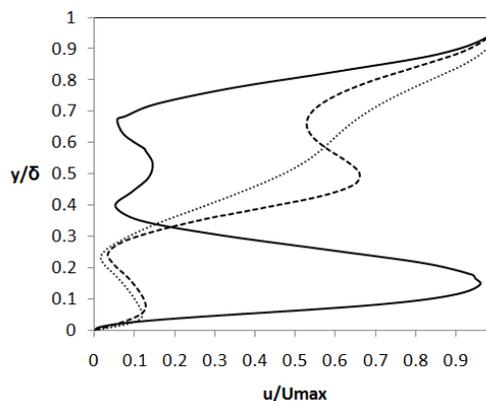
c



d



e



f

Figure 4. Velocity profil distribution (u/U_{max}).
 (a) Single cylinder [5]; (b) s/D 0.004 ; (c) s/D 0.008; (d) s/D 0.012; (e) s/D 0.016; (f) s/D 0.02
 — Sudut kontur 65°; --- Sudut kontur 75°;
 Sudut kontur 85°

3.4 Visualizations of the flow

Fig. 7(a) shows visualization of flow at single cylinder. From the picture, it can be seen that the flow has deflected from the contour at the angle 85 degree. This is indicated with a back flow at that position. Fig 7(b) – 7(f) shows visualization of flow at cylinder with SDB.

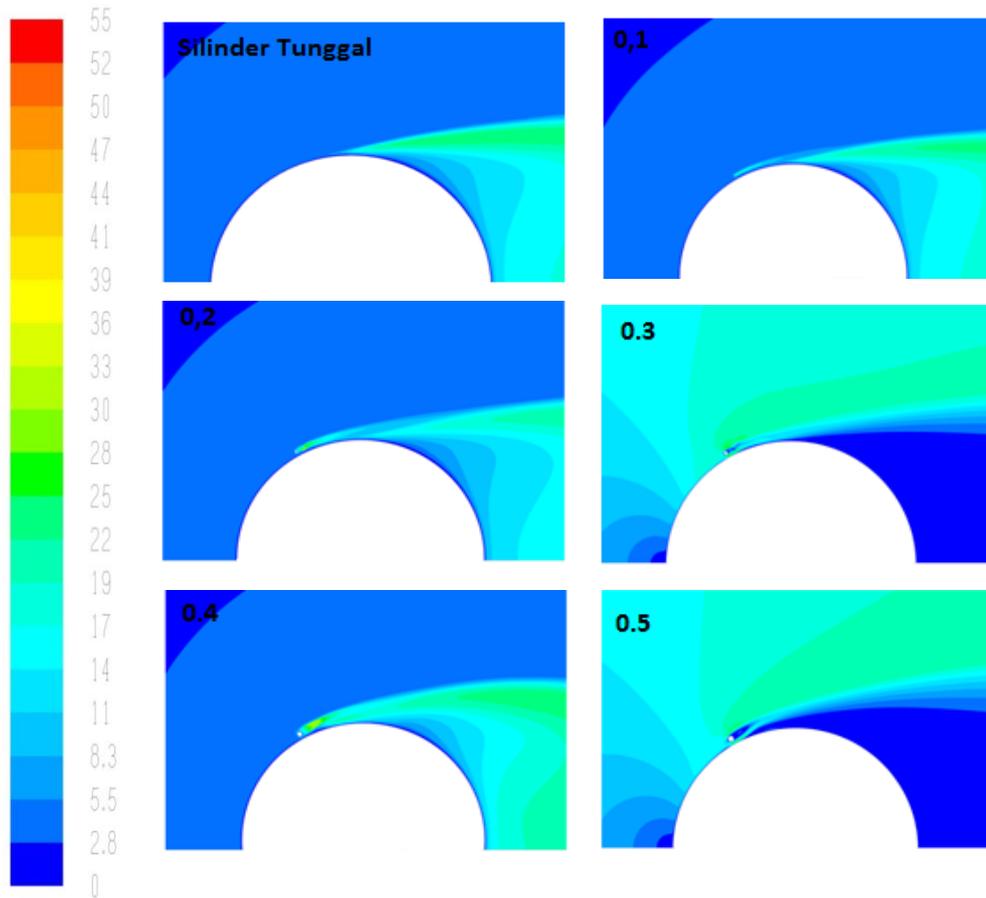


Figure 5. Contour of Turbulent Intensity of single cylinder and cylinder with SDB

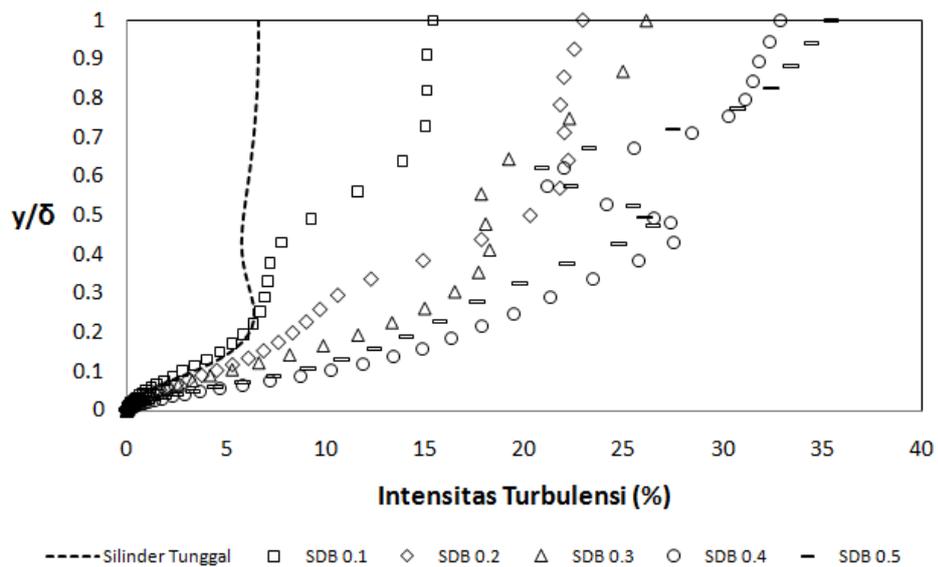


Figure 6. Distribution of turbulent Intensity on single cylinder and cylinder with variation of SDB at angle of contour 75 degree

From the picture, it can be seen that eddy behind square cylinder increase with increasing dimension of disturbance body. The cylinder with SDB 0.1 - 0.3 mm (fig. 7b – 7d) almost

like visualization of the flow at single cylinder. This is because dimensions of disturbance body are very small. At the cylinders with SDB 0.4 and 0.5 mm (fig. 7e - 7f) are clearly forming the

eddy behind the disturbance body. The flow has been deflection from the contour at the angle 65 degree.

3.4 Pressure Drag Coefficient

Drag one of them is caused by flow pressure on the cylinder surface. Drag due to pressure can be known through the value of drag pressure coefficient (C_{dp}). The graph of the drag pressure coefficient (C_{dp}) can be seen in Figure 8. From the graph, it can be seen that the optimum condition for drag force reduction is $s/D = 0.008$ ($s=0.2$ mm), with the value of C_{dp} is 0.55.

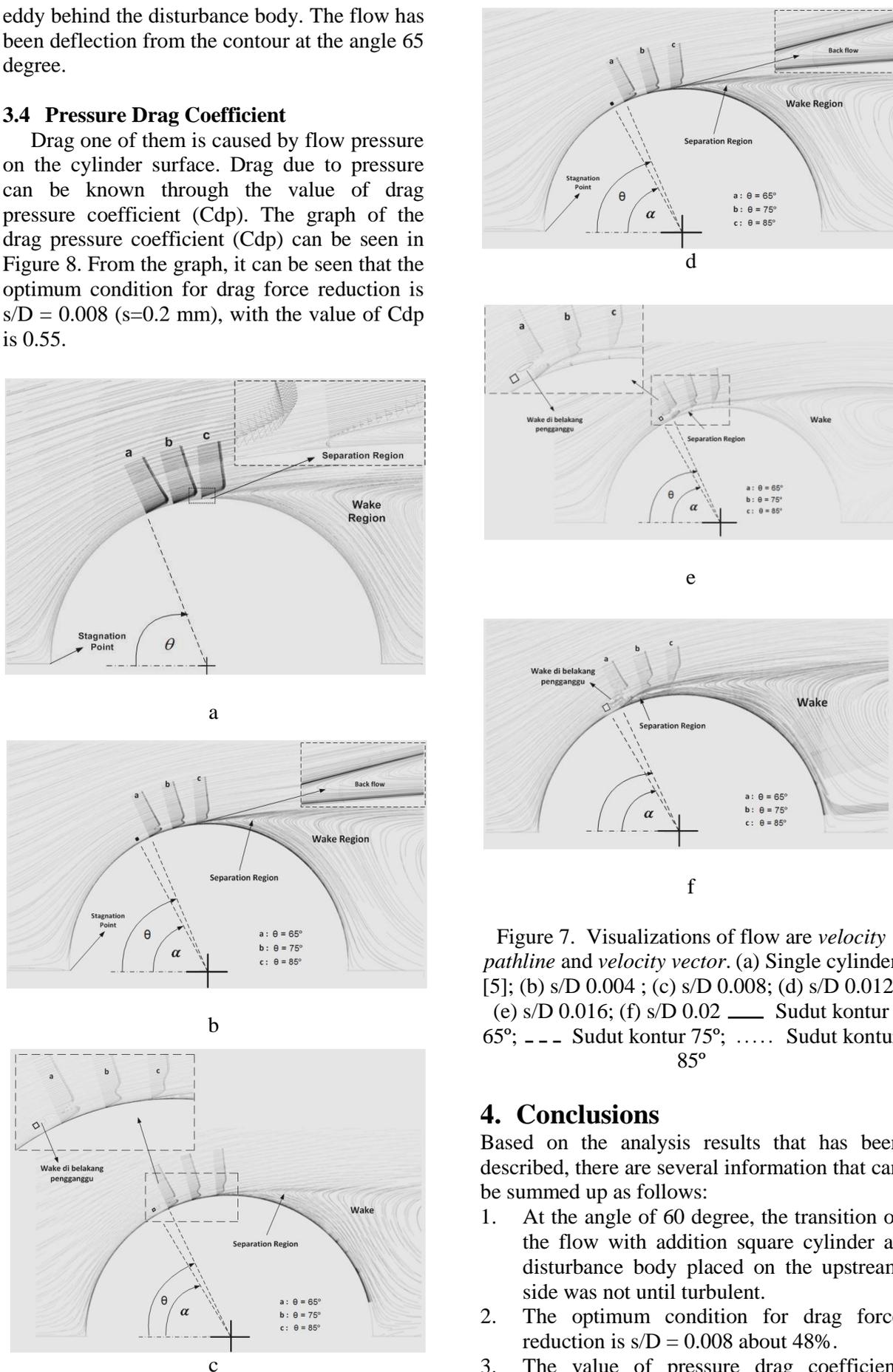


Figure 7. Visualizations of flow are *velocity pathline* and *velocity vector*. (a) Single cylinder [5]; (b) s/D 0.004 ; (c) s/D 0.008; (d) s/D 0.012; (e) s/D 0.016; (f) s/D 0.02 — Sudut kontur 65°; - - - Sudut kontur 75°; Sudut kontur 85°

4. Conclusions

Based on the analysis results that has been described, there are several information that can be summed up as follows:

1. At the angle of 60 degree, the transition of the flow with addition square cylinder as disturbance body placed on the upstream side was not until turbulent.
2. The optimum condition for drag force reduction is $s/D = 0.008$ about 48%.
3. The value of pressure drag coefficient (C_{dp}) increase with increasing s/D to begin with 0.008.

4. The value of turbulent intensity increase with increasing s/D .

References

- [1] Tsutsui, T., Igarashi, T., "Drag reduction of a circular cylinder in an air-stream", *Journal of wind engineering and industrial aerodynamics* Vol. 90, 527-541, 2002.
- [2] Zhang, P.F., Wang, J.J., Huang, L.X., "Numerical simulation of flow around cylinder with an upstream rod in tandem at low Reynolds number", *Applied Ocean Research* 28, 183-192, 2006.
- [3] Alam, M.D., Sakamoto. H., Moriya, M, "Reduction of fluid forces acting on a single circular cylinder and two circular cylinders by using tripping rods", *Journal of fluids and structures* Vol. 18, 347-366, 2003.
- [4] Putra, R.P, "Reduksi gaya hambat pada silinder sirkular dan reduksi pressure drop pada saluran sempit berpenampang bujur sangkar dengan menggunakan batang pengganggu berbentuk square cylinder", *Tesis*, Institut Teknologi Sepuluh Nopember, Surabaya, 2013.
- [5] Rina, "Studi Numerik 2D UNSTEADY-RANS Pengaruh Penambahan Square Disturbance Body Terhadap Reduksi Gaya Drag Circular Cylinder Pada Saluran Sempit", *Tesis*, Institut Teknologi Sepuluh Nopember, Surabaya, 2014.
- [6] Bell, W.H, Turbulence vs drag-some further consideration, *Ocean Engineering*. Vol. 10, No. 1, PP, 47-63,1983.
- [7] Weidman, P.D, Wake transition and blockage effect on cylinder base pressure, *Tesis*, California Institute of Technologi, Pasadena,1968.
- [8] Malalasekera, W., Versteeg, H.K, An introduction to computational fluid dynamics the finite volume method. *United Stated with John Wiley & Sons, Inc., 605 Third Avenue*, New York, 1995.
- [9] Freitas, J. C, The issue of numerical uncertainty, *2nd International Conference On CFD in the Australia*, 6-8 Desember, 1999.
- [10] Ansys Inc, *Fluent theory guide*. Software release 13, 2010.