VANOS
JOURNAL OF MECHANICAL ENGINEERING EDUCATION
http://jurnal.untirta.ac.id/index.php/vanos
ISSN 2528-2611, e-ISSN 2528-2700
Vol.2, No.2, December 2017, Page.127-134

NUMERICAL STUDY OF DIFFUSER EFFECT ON BLUFF BODY : AERODYNAMIC CHARACTERISTICS

M. Yasep Setiawan[1], Martias[1], Erzeddin Alwi[1]
[1]Automotive Engineering, Engineering Faculty,
University of State Padang
Prof. Dr. Hamka St., Kampus UNP Air Tawar, Padang, 25131
m.yasepsetiawan@ft.unp.ac.id

Accepted: 2 December 2017. Approved: 20 December 2017. Published: 30 December 2017

ABSTRACT

This study focuses on the numerical study of flow structure analysis around bluff bodies close to the downstream face of the free-channel surface. This was investigated using coefficient pressure, coefficient drag, lift coefficient and skin friction coefficient on high Reynolds number (ReH = 2.19 × 106). The general procedure for this research is formulated in detail for allocations in dynamic analysis of fluid computing. The numerical model used is the 2D RANS model (Steady Reynolds Averaged Navier Stokes) with the standard k-ε model by giving a flow rate of 40 m / s. This bluff body model will be given the addition of diffuser treatment on the back where the result will be compared to the model without the addition of diffuser and the addition of round shape on the top front on both models. From the research it was found that the coefficient drag bluff body with the addition of diffuser is bigger than bluff body without diffuser, it can be concluded that the diffuser angle used is too big causing the coefficient drag to become bigger.

Keywords: bluff body, diffuser, drag force, RANS, standar k-ε

INTRODUCTION

Many research has been done to determine the fluid flow characteristics in crossing a bluff body. It is done for the sake of academic as well as to examine various forms of construction related to the technical application. In this study, the researchers tried to compare the fluid flow characteristics in bluff body given additional diffuser and without the diffuser.

The diffuser is one of the most important aerodynamic devices often found on F1. It is often used to reduce lift for race cars. In recent years, the diffuser has also been widely used in ordinary cars. The diffuser can work to both reduce drag and increase the downforce of cars placed in the rear section of an underside of a car.

Yakhot and Liu conducted an experimental study of flow on bluff body showing that the flow type is characterized by a horseshoe whirl in the downstream face, an arc vortex in the wake region, the flow of separation on the side and top walls of obstacle and vortex shedding. Krajnovic and Davidson show numerical counter-rotating vortices downstream on the wall-mounted cube.

Lee investigated numerically the high-gap effect of the ground by using unsteady wake flow fields on the bluff body model area close to the ground. He reported that the vortex was started to explain the interaction between the upper shear layer separation and the upwash layer in the gap region. Gurlek et al. conducted experiments to investigate flow structures around the body-oriented bluff body with zero yaw angle referenced in the direction of free flow. In the vertical symmetry area, a large reversed flow is observed at the top of the model and found that the distance of the leading edge reattachment is x / H = 1.3. But the point spacing of the reattachment on the side wall in the horizontal symmetry of this model is about x / H = 1.06 with H as the height of the model. From the research, there is a higher fluctuation rate along the upper sliding layer due to the higher level of momentum.

This research aims to study the effect of adding a diffuser on the bluff body at the rear and comparison with no addition of diffuser. This research will use the 2D RANS (Steady Reynolds Averaged Navier Stokes) numerical method of standard k-ε models on high Reynolds number (ReH = 2.19 × 106).

Numerical Method

This study will be compared form bluff body without additional diffuser with a bluff body with an additional diffuser. In Figures 1 through 3, it describes the dimensional shape. The geometric model is used for meshing like mesh in Figures 4 and 5.

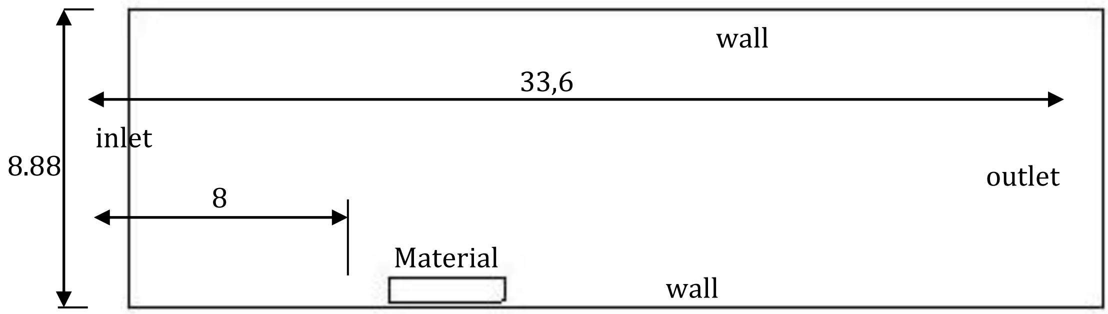

Fig 1. Dimensions and boundary conditions of numerical simulations

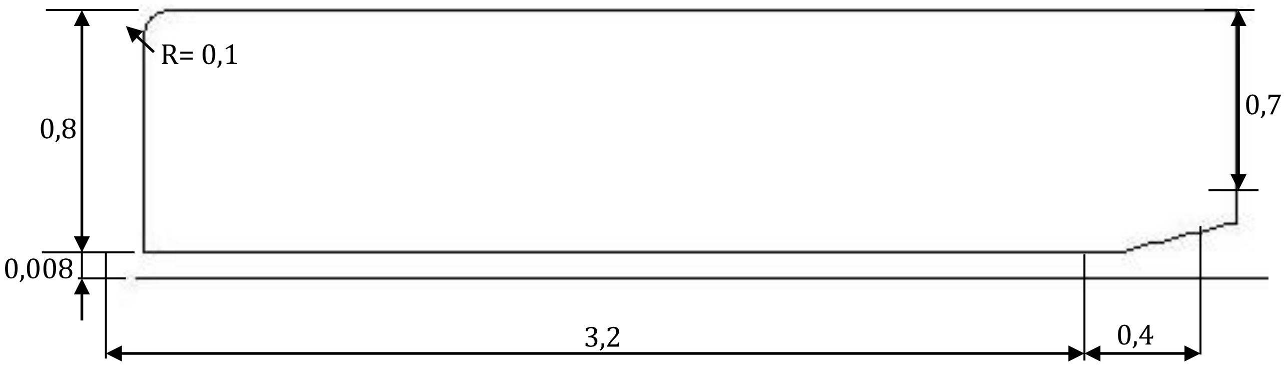

Fig 2. Bluff body With Additional Diffuser

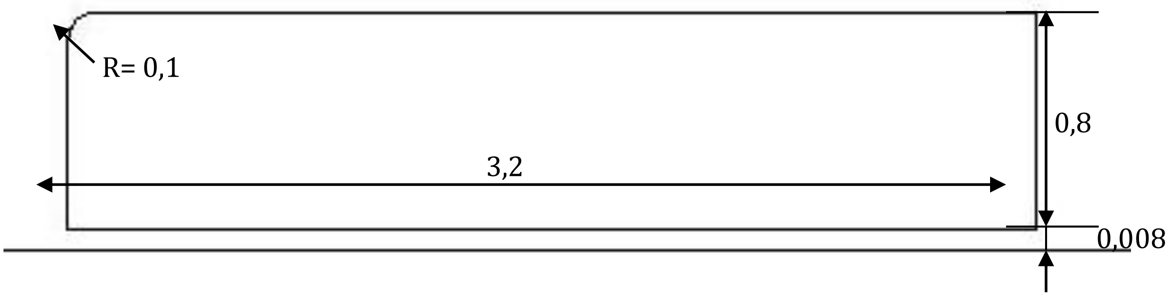

Fig 3. Bluff body Without Additional Diffuser

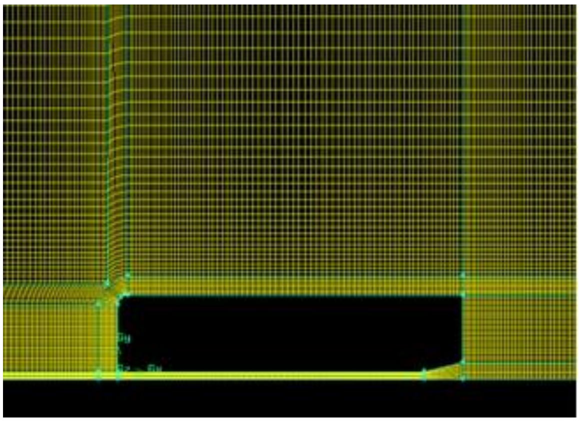

Fig 4. Meshing bluff body with the addition of Diffuser

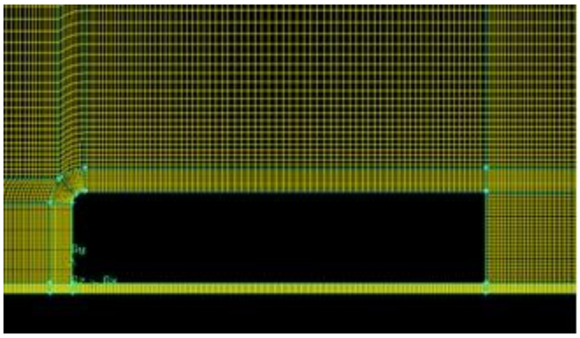

Fig 5. Meshing bluff body without adding Diffuser

a. Governing Equation

Using Reynold's average equation of mass and momentum:

$$\frac{\partial u_i}{\partial x_i} = 0$$

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial u_j} = -\frac{1}{\rho}\left(\frac{\partial P}{\partial x_i}\right) + v\left(\frac{\partial^2 u_i}{\partial x_j^2}\right) - \frac{\overline{\partial u_i' u_j'}}{\partial x_j}$$

where i, j = 1, 2. Here x1 and x2 show the horizontal and vertical directions, respectively; u1 and u2 are the components of average velocity; $\overline{u'_i u'_j}$ is a component of Reynolds voltage which indicates the speed fluctuations; P is pressure, and ρ is the density of the fluid.

The general equation in this simulation

Continuity:

$$\frac{\partial}{\partial x_i}\overline{(\rho u_i)} = 0 \quad (1)$$

Momentum:

$$\frac{\partial}{\partial x_j}\left(\overline{\rho u_i u_j}\right) = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} - \frac{2}{3}\delta_{ij}\frac{\partial \bar{u}_i}{\partial x_i}\right)\right] + \frac{\partial}{\partial x_j}\left(-\rho\overline{u'_i u'_j}\right) \quad (2)$$

b. *Two Equation*

Standard k-ε

The turbulent k-ε model the current standard of turbulence models is most widely used for practical engineering flows as they are strong, economical and provide adequate accuracy for various currents. The model transport equations for k and ε, For steady flow, are given in the literature.

The formula for k-ε for steady state and an incompressible flow:

$$\frac{\partial}{\partial x_i}(\rho k \overline{u_i}) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + G_k - Y_k$$

$$\frac{\partial}{\partial x_i}(\rho \varepsilon \overline{u_i}) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + G_\varepsilon - Y_\varepsilon$$

With value $C_{1\varepsilon}$ = 1.44, $C_{2\varepsilon}$ = 1.92, $C\mu$ = 0.09, σ_k = 1.0, σ_ε = 1.3

Discussion

The total drag and lift coefficient for the bluff body with the addition of diffuser and without the addition of diffuser are shown in Table 1. From Table 1, we can find when the bluff body with the addition of diffuser coefficient total drag is greater than bluff body bluff body without the diffuser and the total coefficient of lift larger than the bluff body without the diffuser. this is due to the addition of diffuser with an angle that is too large such as research conducted Xinjun Hu at al (2011) where they do research on the body sedan that was given the addition of diffuser on the back of the body with different diffuser angle variation (range 00-120). Research concludes that a diffuser with a large angle of 60 has a smaller drag coefficient.

Table 1. The amount of drag coefficient on the bluff body model

Model	Cd	Cl
Without Diffuser	0.119	-1.572
With Diffuser	0.602	-0.546

Comparative analysis can also be seen from the following figures 6 and 7:

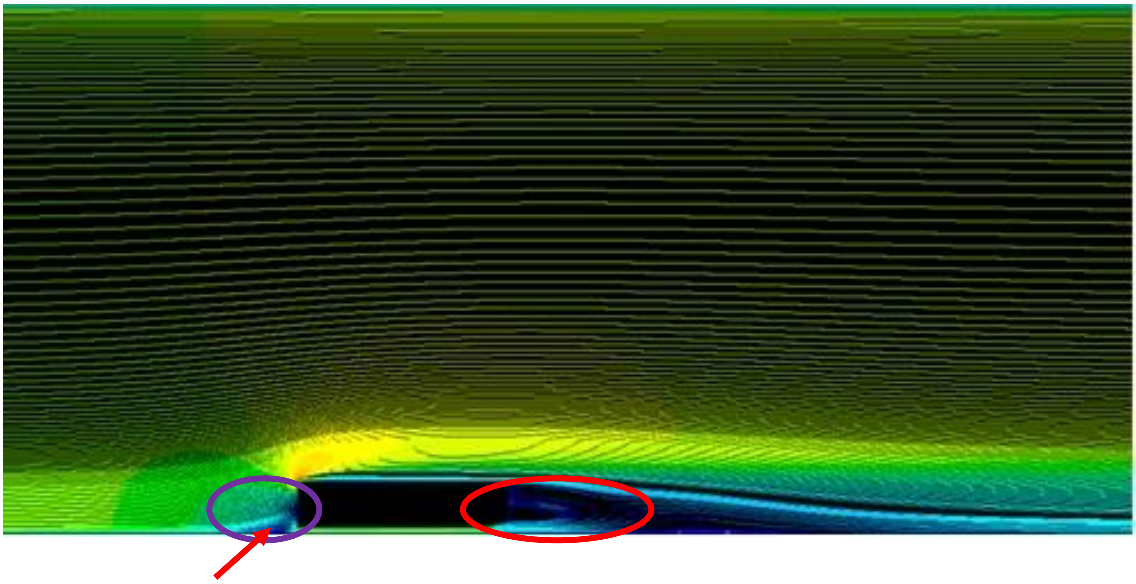

Fig 6. Pathline Velocity Model Bluff body with Additional Diffuser

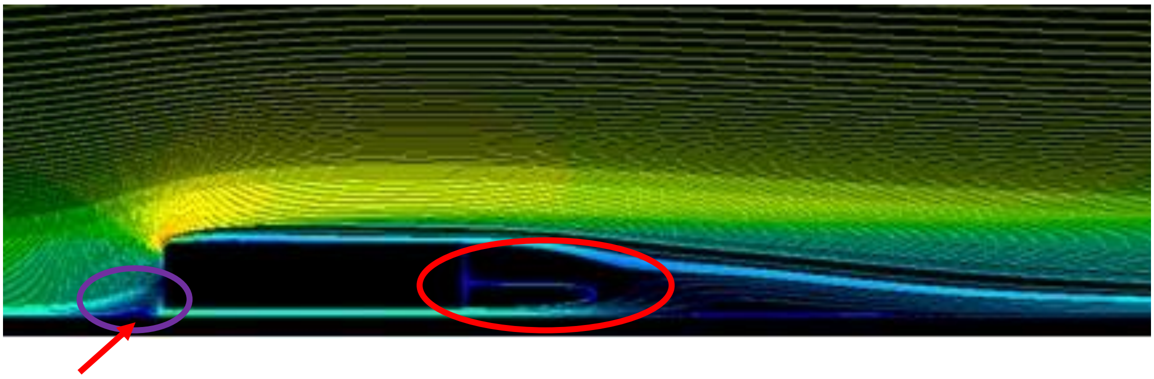

Fig 7. Pathline Velocity Model Bluff body Without Additional Diffuser

Figures 6 and 7, the point of stagnation can be seen in areas marked with red circles on the back of the body. The front of the bluff body is marked with a purple circle where there is a bubble separation, in addition, the diffuser has a smaller and shorter bubble separation than without the addition of diffuser. This is due to the bottom of the bluff body model there is a gap that caused a considerable pressure difference in the gap area and free stream near the body.

The front of the body is marked with a purple circle that also looks blue, shown by arrows. This states that the area has a small velocity and great pressure, while the area near the gap has a higher velocity and lower pressure. Because of this pressure difference, the adverse pressure gradient occurs so that the flow is subsequently separated by the flow of energy from the free stream that can define the flow attach back to the contours then bubble separation occurs. Because of the difference in speed in both large areas of the bluff body model with the addition of the diffuser, it provides a small bubble separation. For more details see Figures 8 and 9.

Figures 8 and 9 show the rear of the bluff body model with the addition of a diffuser there is a small bubble separation close to the top, while at the rear of the bluff body model without the addition of a diffuser provides a larger bubble separation compared to the model with the addition of diffuser. This is due to the back of the bluff body model with a diffuser there is a steering that causes a considerable pressure difference in the back of the upper and lower body. In an elliptical picture, the pressure on the upper area of the upper body is greater than the lower body of the addition of the diffuser and the pressure on the lower body is greater than the upper body of the model without the diffuser. Because of this pressure difference, an adverse pressure gradient occurs so that the flow is subsequently absorbed by the flow energy of the free stream that is capable of deflecting the attach flow back to the contour and bubble separation occurs. Because of the difference in speed in both large areas of the bluff body model with the addition of a diffuser, it provides a small bubble separation and the area is at the top. On the front with the addition of round shape on the top of the front can reduce the occurrence of bubble separation at the top of the two models bluff body.

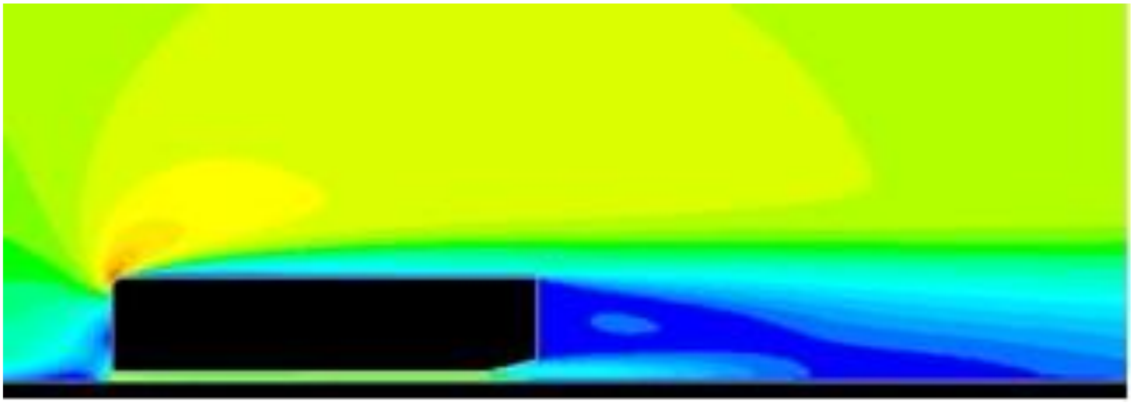

Fig 8. Contour Velocity On Bluff Body Model With Diffuser Addition

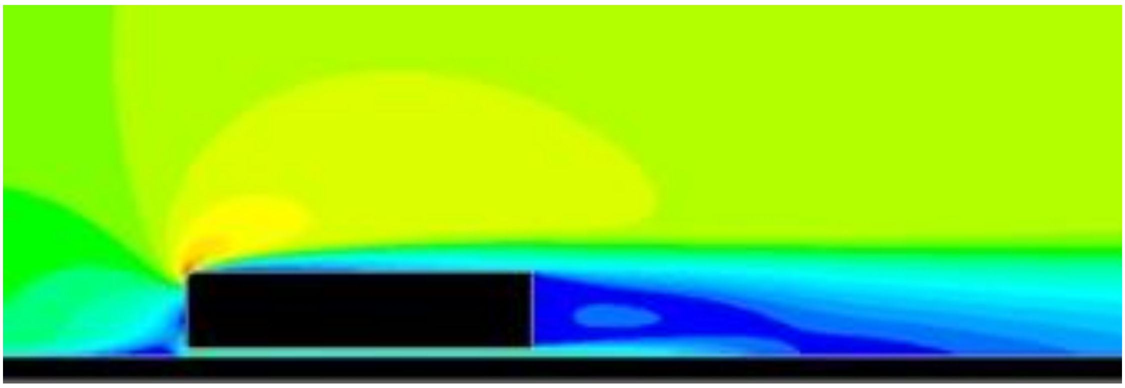

Fig 9. Contour Velocity In Bluff Body Model Without Diffuser Addition

CONCLUSION

Based on the results of the research and discussion above, some conclusions can be drawn as follows:

a. The addition of a diffuser with an overly velocity angle to the bluff body gives the result of a larger drag coefficient than the bluff body without additional diffuser.
b. In this research coefficient drag bluff body with the addition of diffuser is bigger than bluff body without diffuser, it can be concluded diffuser angle used is too big causing coefficient drag a become big.

REFERENCES

Gurlek, C.; Sahin B. Particle Image Velocimetry Studies Around A Bluff body Body Close To A Plane Wall. 2010. Department of MechanicalEngineering, University of Cumhuriyet, 58140 Sivas. Turkey

Gurlek C, Sahin B, Ozalp C, Akilli H. Flow structures around a three-dimensional bluff body body with ground effect. Wind Struct 2008;11(5):345–59.

Krajnovic S, Davidson L. Large eddy simulation of the flow around a three-dimensional bluff body. In: AIAA 2001-0432, 39th AIAA aerospace sciences meeting and exhibit. 2001.

Lee B, Kim T, Lee D. Control of vortex shedding behind a bluff body cylindernear the ground. Numer Heat Transfer A 2005;47:787–804.

Östh J., Krajnović S. The flow Around A Simplified T ractor-Trailer Model Studied By Large Eddy Simulation. 2012. Division of Fluid Dynamics, Department of Applied Mechanics, Chalmers University of Technology, SE-412 96 Göteborg. Sweden

Xingjun Hu,Rui Zhang,Jian Ye, Xu Yan, Zhiming Zhao. Influence of Different Diffuser Angle on Sedan's Aerodynamic Characteristics. International Conference on Physics Science and Technology. China 2011

Yakhot A, Liu H, Nikitin N. Turbulent flow around a wall-mounted cube: a direct numerical simulation. Int J Heat Fluid Flow 2006;27:994–1009.