EXPERIMENTAL AND NUMERICAL STUDY OF TURBULENT FLOW CHARACTERISTIC IN ASYMMETRIC DIFFUSER

Yiyin Klistafani*
Politeknik Negeri Ujung Pandang
*yiyin_klistafani@poliupg.ac.id

ABSTRACT

Diffuser is one of the constructions that is able to control the behavior of the fluid. Increase in pressure that occurs in the diffuser will generate a positive pressure gradient or collectively the adverse pressure gradient (APG) that affects the development of the boundary layer. The greater APG occurs, the greater the energy needed by the fluid to fight it, because it will lead to separation. One of all factors that strengthen the separation is the divergence angle of diffuser. Numerical analysis is carried out 3D steady-RANS using Fluent 6.3.26 to analyze the characteristics of turbulent flow in the asymmetric diffuser with constant aspect ratio, especially at straight wall. The simulation results compared with experimental results. Experimental studies carried out by measuring the static and stagnation pressure in the test section using static pressure tap and Pitot tube. Reynolds number based on the inlet diffuser height (W_1) and the maximum velocity at the inlet diffuser for $Re_{W1} = 8.7 \times 10^4$. Results obtained from experiments and numerical studies on the asymmetric diffuser with constant aspect ratio are SST k-ω is the best model in predicting the pressure coefficient (Cp) while the standard k-ε is the best model in predicting the skin friction coefficient (Cf). Establishment of huge vortex in the diffuser cause Cp value is not maximized. Based on Cf value of the experimental and numerical results, separation doesn't occur in a straight wall (the bottom wall and rear wall).

Key words: asymmetric diffuser, aspect ratio, pressure coefficient

INTRODUCTION

Diffuser is one of the constructions that able to control the behavior of the fluid. Increase in pressure that occurs in the diffuser will generate a positive pressure gradient or collectively the adverse pressure gradient (APG) that affects the development of the boundary layer. The greater APG occurs, the greater the energy needed by the fluid to fight it, because it will lead to separation. One of all factors that strengthen the separation is the divergence angle of diffuser. Separation is one of the factors that affect the value of pressure coefficient (Cp) and the pressure recovery coefficient (Cpr).

This study aims to determine differences of turbulent flow characteristics on a flat-walled asymmetric diffuser with different turbulance models which are standard k-ε, realizable k-ε, and shear stress transport (SST) k-ω. Reynolds numbers used in this study is Re_{W1} 8.7 x 10^4. The study also aimed to compare the results of numerical steady Avereged Reynolds Navier-Stokes (RANS) using the software Fluent 6.3.26 with the experimental results.

Buice and Eaton (1997) conducted a study towards the asymmetric plane diffuser. Model test section is similar to diffuser that used by Obi, et al research, asymmetric plane diffuser 10^o, with the addition of two splitter modifications in the test section. The data and results obtained boundary separation in the diffuser outlet is better, where vortex area that occurs to be reduced compared to the boundary separation without splitter. That's why splitter able to eliminate areas of separation and reduce vortex area. Installation splitter is a passive control method of the end-wall boundary layer inside diffuser, where the using splitter is also useful in generating better pressure distribution (Buice and Eaton, 1997).

Shear stress transport (SST) k-ω predictions is in a good agreement with the experimental data in predicting pressure coefficient (Cp), while k-epsilon turbulence models is very good at predicting the value of skin friction coefficient (Cf) (Charles and Benoit, 2009). On the other hand, El-Behery and Hamed also conduct research on flat-plane asymmetric diffuser and compare their results with the results research of Obi et al, Kaltenbach and Buice-Eaton. From six of the turbulence models used, show that v^2-f turbulence models gives the best agreement with the experimental data, then followed by a standard k-ω models and shear stress transport k-ω (SST). Whereas for the Standard k-ε models, Low Reynolds number k-ε models and Reynolds stress model (RSM) models are in poor agreement compared with experimental data (Behery and Hamed, 2011).

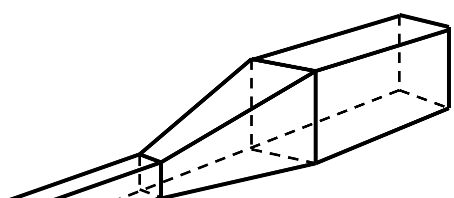
Divergence angle of diffuser affects the occurrence of separation, although it uses a great Reynolds number. The greater divergence angle of diffuser, the separation occurs (Merlina, 2011) early. In the previous study, used diffuser without constant aspect ratio (Merlina, 2011) and numerical methods were applied is two dimensional (2D) (Behery and Hamed, 2010), so it is necessary to conduct experimental study of diffuser with constant aspect ratio and compared with 3D numerical analysis.

RESEARCH METHODS

A. Experimental Methods

Test section in this study is a flat-walled asymmetric diffuser 10° constant aspect ratio from acrylic material with the following specifications (Figure 1).

Geometri 3D



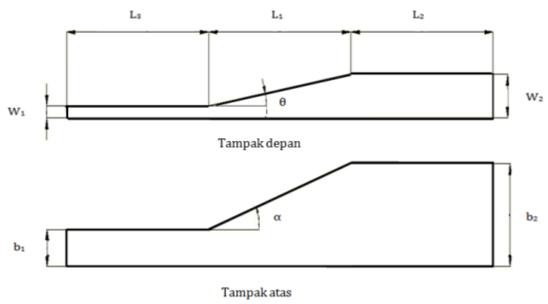


Fig. 1 *Test Section*

Length of diffuser (L_1) = 500 mm, downstream channel (L_2) = 500 mm, upstream channel (L_3) = 500 mm, lebar inlet span diffuser (b_1) = 100 mm, lebar outlet span diffuser (b_2) = 80 mm, tinggi inlet diffuser (W_1) = 50 mm, height outlet diffuser (W_2) = 140 mm, Divergence angle of top wall (θ) = 10°, dan Divergence angle of front wall (α) = 20°. Static and stagnation pressure data at midspan wall diffuser will be taken in experimental studies. Retrieval of static pressure data using wall prsessure tap and retrieval of stagnation pressure data using a Pitot tube.

B. Numerical Methods

Numerical study performed using Computational Fluid Dynamics (CFD) with FLUENT software 6.3.26 and GAMBIT software 2.4.6 to create the initial model and do discretization (meshing) on that model. There are two steps on numerical study procedure which *pre-processing* step using GAMBIT software 2.4.6 and followed by post-processing step using FLUENT software 6.3.26. The type of mesh used is hexahedral-map (figure 2) and the formation of mesh grid in the area close to the wall using double sided grading successive ratio. Boundary condition for inlet is velocity inlet and for the outlet is outflow. The level of convergence set is 10^{-6}.

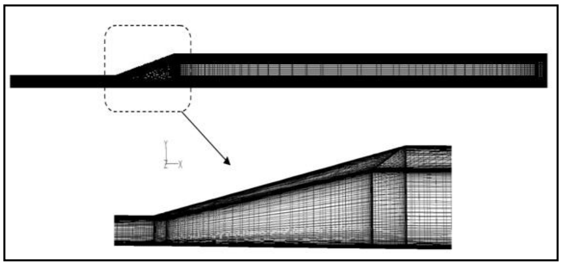


Fig. 2 Mesh on asymmetric flat-walled diffuser hexahedral-map type

RESULTS AND DISCUSSION

A. Pressure Coefficient (Cp) Distribution at Straight Wall

Distribution Cp of exmerimental and numerical at *straight walls*, bottom wall and rear wall, shown on figure 2 dan 3. In figure 2 shows that the pressure coefficient distribution of numerical simulation for all three turbulence models have differences with the experimental data in the upstream channel and in particular on a cross section along the diffuser. While on the downstream channel, Cp distribution of numerical simulation results (standard k-ε, realizable k-ε) tends to approach experimental data, except on shear stress transport (SST) k-ω that shows the results under prediction.

Impairement of Cp value occurs on the upstream channel to the inlet diffuser (cross section x/L_1 = -0.45 to x/L_1 = 0) at bottom wall, both experimental and numerical simulation results (Figure 3) due to the effect of thickening of the boundary layer that causes value of U_{max} increase along the distance changes. So that the range is no longer a zero pressure gradient region but is favorable pressure gradient (FPG) region. Seen in the figure 3, the SST k-ω gives better agreemet with experimental data than the standard k-ε and realizable k-ε on the upstream channel.

When entering the diffuser, static pressure increases on the fluid flow that quite large due to the influence of adverse pressure gradient (APG). The increasing of Cp value at bottom wall of the experimental results is greater than numerical simulation results for all three models of turbulence. The value Cpmax of experimental results which 0.156 at the cross section x/L_1 = 0.48 while the value Cpmax for standard k-ε, realizable k-ε and k-ω SST consecutively are 0.126 (x/L_1 = 0.42), 0.129 (x/L_1 = 0.84), and 0.105 (x/L_1 = 0.3). Theoretically, Cp_{max} should be located at x/L_1 = 1, or in other words Cp_{max} is Cpr but the actual condition is not, due to excistence of vortex inside diffuser. The existence of vortex, greatly affect the Cp distribution along the diffuser. Effective cross section area in diffuser narrowed due to the vortex, so the increase Cp value along diffuser is not achievable and quite the contraty, coefficient pressure decreased in value. Experimental Cp distribution in fig. 3, decreased by 0.041 at x/L_1 = 0.48 to x/L_1 = 1. *Cp* distribution of *SST k-ω* desreased by 0.55 (from x/L_1 = 0.3 to x/L_1 = 0.57) and increased (hingga x/L_1 = 0,8) before being declined again. While the standard k-ε and realizable k-ε has constant Cp distribution along the diffuser.

In the downstream channel, Cp distribution at bottom wall is relatively constant due to the absence of APG effect and vortex near the bottom wall, that's why so static pressure values along mid span of bottom wall are relatively constant. Among the three models of turbulence, seen in the fig. 2, the realizable k-ε has good agreements with experimental data and the second is a standard k-ε while the SST k-ω has poor agreements with experimental data.

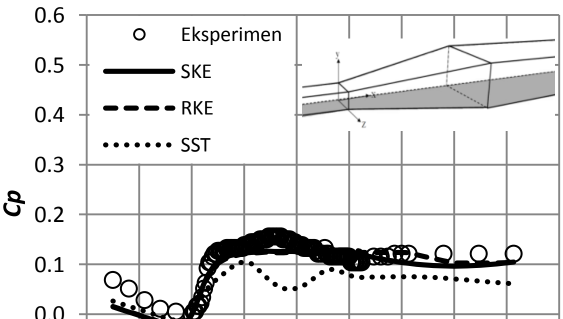


Fig. 3 Experimental and numerical pressure coefficient (Cp) distribution at bottom wall

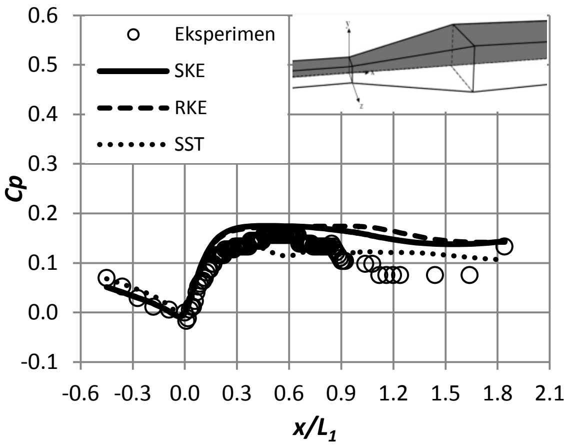


Fig. 4 Experimental and numerical pressure coefficient (Cp) distribution at rear wall

Figure 4 shows experimental and numerical pressure coefficient (Cp) distribution at rear wall. In the upstream channel, Cp distributions of standard k-ε and realizable k-ε are similar to the experimental data, while the SST k-ω has differences with the experimental data although not too significant.

Cp values at rear wall of SST k-ω have good agreements compared with experimental data at cross section x/L_1 = 0 to x/L_1 = 0.4 and have the significant differences at x/L_1 = 0.4 to x/L_1 = 0.7. Standard k-ε and realizable k-ε has same pressure coefficient from cross section x/L_1 = 0 to x/L_1 = 0.8 and impaired Cp to downstream channel. Experimental *Cp* values also decreased at x/L_1 = 0.6 to x/L_1 = 1.1. Impairment Cp along certain cross sections at the diffuser indicates existence of vortex in its area.

Rear wall's *Cp values in downstream channel* between experimental results and numerical simulations (for three turbulence models) have significant differences. But both of them show the same trend which is Cp values decrease along downstream channel and increase again for experimental and standard k-ε results. As well as the diffuser, impairment Cp on the downstream channel indicates that the vortex is formed in its area. It can be made clear by Figure 4 which displays the velocity pathline at horizontal mid span.

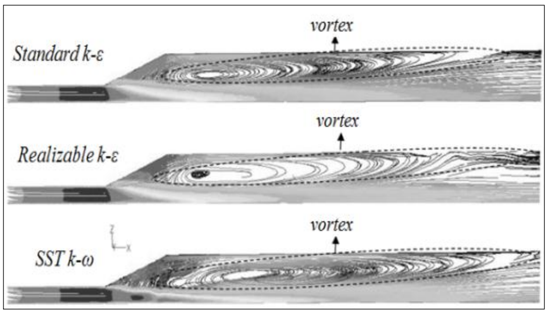


Fig.5 Velocity *path line* of numerical results on horizontal mid (*velocity magnitude* dalam *m/s*)

On figure 5 shows that vortex appears in downstream channel, either on velocity pathline of standard k-ε, realizable k-ε, and shear stress transport (SST) k-ω. The existence of vortex causes the flow rate between the rear wall and vortex as if through a nozzle, so the velocity of the fluid in the area is getting up and down again. That's why the Cp values obtained on the rear wall fluctuate in the downstream channel.

B. *Skin Friction Coefficient* (*Cf*) Distribution at *Straight Wall*

Figure 6 and 7 show experimental and numerical skin friction coefficient distribution at bottom wall and rear wall.

Cf distribution shown from x/L_1 = -0.2 to x/L_1 = 1.6.

On figure 6 shows that experimental and numerical Cf distributions have significant differences. However standard k-ε gives good agreements with experimental data. On the upstream channel (cross section x/L_1 = -0.2 hingga x/L_1 = 0), experimental and numerical Cf distribution have same trend which is decreas, even if the amount of decreasing between both of them are different. The percentage of CF impairment for experimental result is about 62%. For Numerical analysis, there are not any significant differences of Cf value at upstream channel for three turbulence models with percentage of impairment is about 15%.

When the fluid flows into the expansion area, the flow will be impeded by APG, so the stagnation pressure that closest to the wall would be impaired, in line with the flow direction trough the diffuser. It also cause the shear stress (τ_w) value impaired. The impairment of shear stress make Cf value decrease.

As shown on figure 6, numerical Cf value has decreased until cross section x/L_1 = 0.1. The conditions did not last long due to the Cf value increase until cross section x/L_1 = 0.5 for standard k-ε and realizable k-ε and x/L_1 = 0.2 for shear stress transport (SST) k-ω. While experimental Cf value increase until cross section x/L_1 = 0,2. On that cross section, experimental Cf value is same as SST k-ω. Theoretically, the increase in Cf value is unlikely to happen due to APG effect. However vortex that appears in the diffuser (as shown in fig. 5), make a flow rate between the bottom wall and the vortex increases, that's why the value of shear stress occurs also enlarged. It has pushed up the increase in Cf value at the particular cross section in diffuser.

The percentage of decresing *Cf* values at bottom wall (fig. 6) for numerical results are bigger than experimental results. SST k-ω even shows the impairment Cf value drastically, until it reaches a zero at x/L_1 = 0.6 and minus value at x/L_1 = 0.6 to x/L_1 = 0.8. Zero Cf value indicates the occurrence of separation flow, this is because the flow near the wall in deficit momentum as the APG effect and the friction, that's why the velocity gradient at the wall is zero $\left[\left(\frac{du}{dy}\right)_{y=0} = 0\right]$ and separation happen. While decreasing Cf value of standard k-ε, realizable k-ε, and the experiment did not reach a zero value. Thus the separation does not occur on the bottom wall if it is based on experimental results, standard k-ε and k-ε realizable.

There are some differences Cf value in the downstream channel for experimental and numerical results, as well as also with the trend graph, shown at Figure 6. The Cf value of experimental results is greater than the numericals and those have a tendency to rise. This is because there is no APG effect, so that the momentum of the flow recovered and the value of shear stress also enlarges accompanied by increased Cf value.

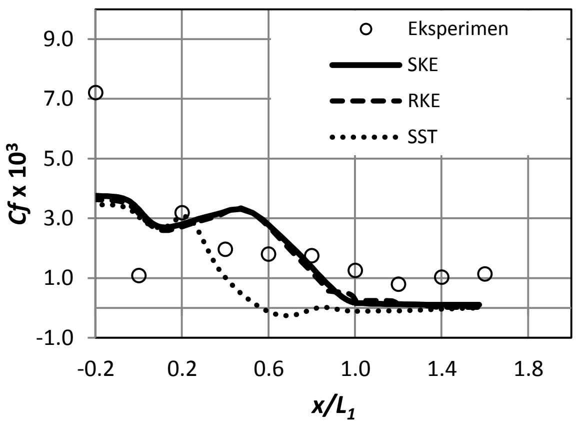


Fig. 6 Experimental and numerical skin friction coefficient (C_f) distribution at bottom wall

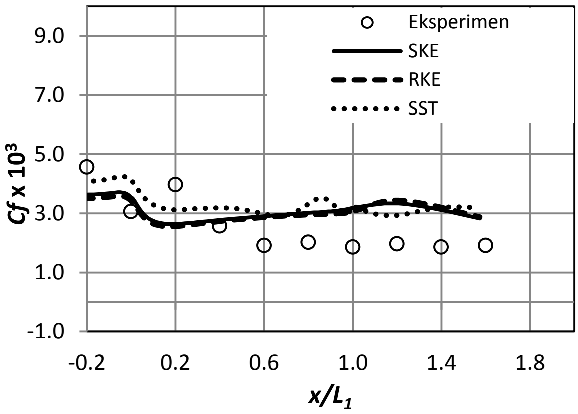


Fig. 7 Experimental and numerical skin friction coefficient (C_f) distribution at rear wall

Figure 7 shows a comparison of experimental and numerical Cf value at the rear wall. Overall, there are not any significant changes of the Cf value at the rear wall from upstream to downstream channel. Experimental Cf distribution is different with numericals. However the standard k-ε and k-ε realizable have a little able to approach the experimental results.

On the upstream channel, the percentage of Cf value impairment is about 49%. While numerical Cf value for standard k-ε and realizable k-ε are relatively constant and slightly increased for the SST k-ω. Then the numerical Cf value slightly decreased when shortly after entering the diffuser due to the lack of APG around the rear wall.

Impairment of experimental Cf value occur at cross section x/L_1 = 0,2 hingga x/L_1 = 0,6 and is relatively constant until the downstream channel. So it can be concluded that there is not separation along the rear wall. Either *standard k-ε* and *realizable k-ε* results, which tends to a constant value at cross section x/L_1 = 0,6 to stream out of the diffuser. In the downstream channel, Cf value has fluctuated up and down back under the influence of vortex in the rear wall. While Cf value for *SST k-ω* fluctuate shortly before exiting from the diffuser to the downstream channel. Based on numerical predictions, there is not separation along rear wall.

CONCLUSIONS

According to results and discussion, there are several conclusions:

- SST k-ω is the best model in predicting the pressure coefficient (Cp) while the standard k-ε is the best model in predicting the skin friction coefficient (Cf).
- The establishment of huge vortex in the diffuser cause Cp value is not maximized.
- Based on Cf value of the experimental and numerical results, separation doesn't occur in a straight wall (the bottom wall and rear wall).

ACKNOWLEDGEMENTS

I would like to express my appreciation to Prof. Ir. Sutardi, M.Eng., Ph.D, as my adviser, who provided me with direction, encouragement throughtout the course of this research and many helpful discussions. I also would like to express my gratitude for chance and facilities of our research by Politeknik Negeri Ujung Pandang and Institut Teknologi Sepuluh Nopember.

REFERENCES

Buice, C.U., Eaton, J.K. 1997. Experimental investigation of flow through an asymmetric plane diffuser. Technical Report. Department of Mechanical Engineering, Stanford University.

Charles, H., Benoit, T. 2009. Reynolds-Averaged Navier-Stokes Modelling for Industrial Applications and Some Challenging Issues. Department of Fluid Mechanics,Vrij Uuniversiteit, Brussel, Belgium.

El-Behery, S.M., Hamed, M.H. 2011. A Comparative Study of Turbulence Models Performance for Turbulent Flow in a Planar Asymmetric Diffuser. *International Journal of Mechanical Systems Science and Engineering* ; vol.2 no.2, 78-89.

Merlina, F. 2011. Experimental and Numerical Study of Turbulent Boundary Layer Characteristics in Asymmetric 3D Flat-Walled Diffuser 20^{o}. *Final Project*. Department of Mechanical Engineering, ITS.