VOL. 11, NO. 3, FEBRUARY 2016 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

THE EFFECTS OF ANGLE OF ATTACK ON 3-DIMENSIONAL TURNING DIFFUSER ON BAFFLE PERFORMANCES

NurHazirahBinti Noh Seth[1], Norasikin Binti Mat Isa[1], Safiah Binti Othman[1] and Vijay R. Raghavan[2]
[1]Faculty of Mechanical and Manufacturing Engineering, University of Tun Hussein Onn Malaysia, Malaysia
[2]OYL R and D Centre Sdn. Bhd, Taman Perindustrian Bukit Rahman Putra, Sungai Buloh, Selangor, Malaysia
E-Mail: n_hazirah1110@yahoo.com

ABSTRACT

The aim of installing baffles is to reduce or eliminate, if possible, secondary flow which exists mostly at the inner wall of the turning diffuser. Furthermore, other than distortion at the inner wall, 3-dimensional turning diffuser has secondary flow at both left and right wall. This was due to the diffusing activities which were not only in x-y direction but in y-z direction as well. Experiment on 3-dimensional turning diffuser with baffle has been conducted using airfoil baffle with AOA=17º. Present study focuses on changing angle of attack of the installed baffle and their effects on flow uniformity and pressure recovery using numerical approach. The baffle was rotated 3º clockwise and anti-clockwise resulting in AOA=20º and AOA=14º respectively. Qualitative and quantitative comparison was discussed in this paper. AOA=14º offers higher quality of flow structure as compared to AOA= 20º, but still could not surpass the performance using preliminary design baffle with AOA= 17º. The abnormality of flow in AOA=20º resulting in higher pressure loss, thus affecting pressure recovery. The optimum configuration can be developed if the effort of improving the airfoil design could be enhance in future works.

Keywords: turning diffuser, airfoil baffle, angle of attack.

INTRODUCTION

Flow separation due to adverse pressure gradient and curvature effects definitely contribute to pressure losses, which will eventually disrupt turning diffuser performance in terms of pressure recovery. Other than that, flow separation definitely contributes to flow non-uniformity. Both parameters were the quantitative measurement for turning diffuser performance. Experiment on 3-dimensional turning diffuser without baffle has already mentioned the starting point location of a secondary flow (Normayati *et al.*, 2013) supported with numerical approach using ANSYS FLUENT simulation using the appropriate turbulence model (Normayati *et al.*, 2014). Extension study on 3-dimensional turning diffuser were continued by installing airfoil baffle and the flow structure were observed using Particle Image Velocimetry (PIV).

A good criterion for a turning baffle is the design that helps to provide optimum velocity distribution on baffle surface (Sahlin and Johansson, 1991). The objective aims to help the flow to stay as long as possible at the baffle surface.It was recommended that baffle should be located slightly downstream form the narrowest diffuser throat (Fox and Kline, 1962). This will produce a smaller diffuser with maximum efficiency (Fox and Kline, 1962). If series of baffle were used, the baffle should be thick enough to maintain the distance between each baffle along the chord (Sahlin and Johansson, 1991). Other experiment involving installation of baffle in bends and expanding curves (Chong *et al.*, 2008) (Lindgren and Johansson, 1998) (Nakano *et al.*, 2007) (Majumdar *et al.*, 1998) shows increasing of pressure recovery (Schreiber *et al.*, 2004) (Farismadan *et al.*, 2010) and proves that installation of baffle do improve turning diffuser performance.

Combining data and baffle design from previous study, preliminary design of baffle were used in an experiment using the same rig as experiment of 3-dimensional turning diffuser without baffle (Normayati *et al.*, 2014). Data collected from the experiment were then used to validate numerical approach of the same case (Nur Hazirah *et al.*, 2015).Validated results shows agreement between both experiment and numerical approach. The selected Standard K-Epsilon (SKE) (Guohui and Saffa, 1996) (Gopaliya *et al.*, 2007) turbulence model was the best turbulence model to solve both 2-dimensional and 3-dimensional turning diffuser (Nur Hazirah *et al.*, 2015). Present study used the same approach on different design of baffle.

Modification of baffle using simulation reduces the cost and time consume compared to experimental approach. The result of turning diffuser performance was compared with preliminary design and optimum design could be proposed for the system. Parameters of the baffles are shown in Figure-1. With thickness of 0.53cm, 7cm chord length and leading edge located ½L_{in}/W (inner wall length to an inlet throat width ratio), the angle of attack were then measured as shown in Figure 2. Preliminary design of baffle constructed has angle of attack (AOA) of 17degree. The result of simulation has been validated using experimental values and lay out with conclusion of the best turbulence model to be used with 2-dimensional and 3-dimensional turning diffuser (Nur Hazirah *et al.*, 2015).

VOL. 11, NO. 3, FEBRUARY 2016 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

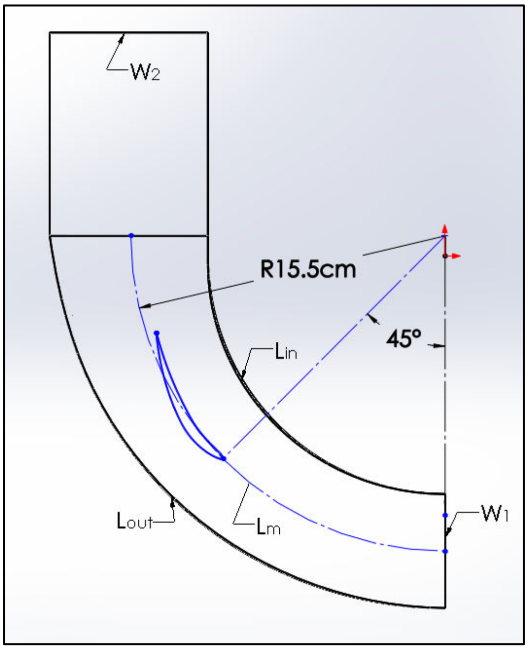


Figure-1. Location of airfoil baffle in 3-dimensional turning diffuser.

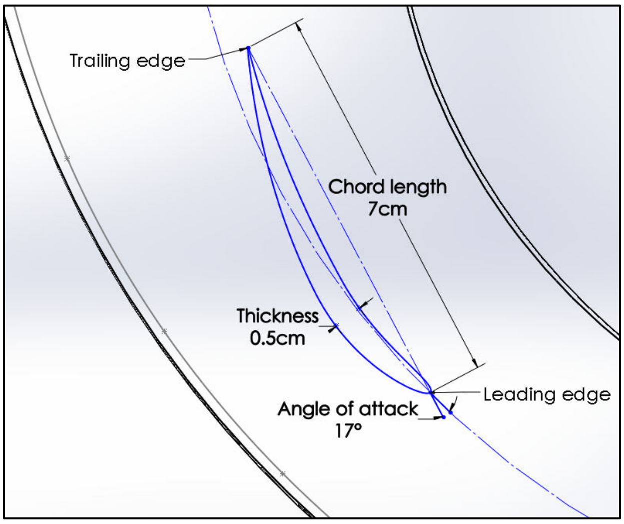


Figure-2. Airfoil baffle parameters (length and thickness in cm).

METHOD AND MODIFICATION

Preliminary airfoil baffle with angle of attack 17º has been proven able to increase turning diffuser performance when flow separation were seen reduced to some degree. However, present approach has the aim to improve flow uniformity and pressure recovery achieved previously and eliminates flow separation, if possible. The plane of interest remains the same (NurHazirah *et al.*, 2015) as shown in Figure-3, on the x-y plane.

Since 3-dimensional turning diffuser flow structure was more complex than 2-dimensional turning diffuser, many plane of interest could be study and brought up for discussion. Flow separation does not occur only at the inner wall, but at left and right wallas well. Diffusing activities at both x-y and y-z direction causes secondary flow which eventually resulting in flow uniformity disruption. Thus, present study added another plane of study, namely Plane C, on the y-z plane as shown in Figure-4.

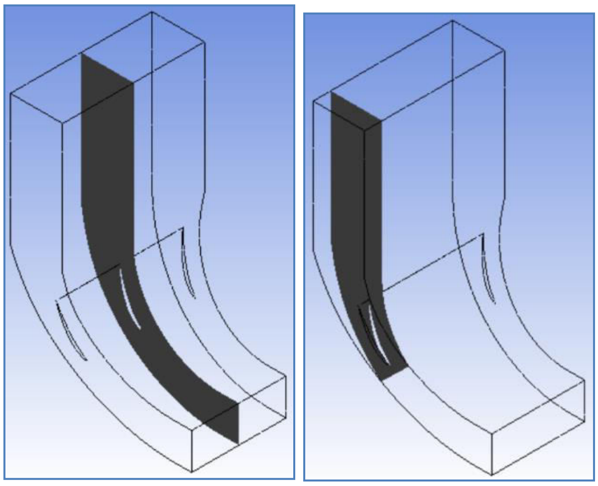


Figure-3. Plane of interest Plane A (left) and Plane B (right).

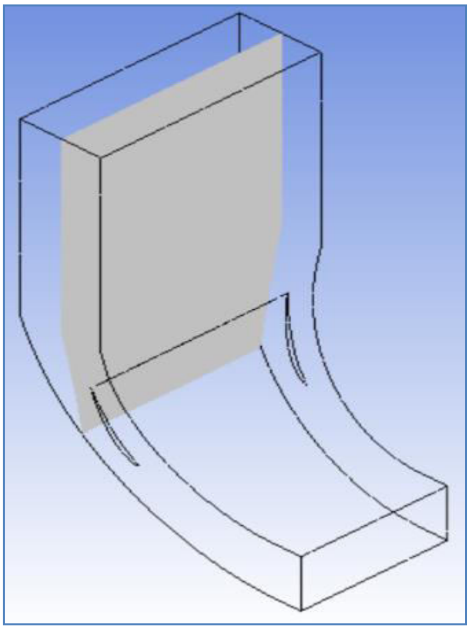


Figure-4. Additional plane of interest, Plane C.

Validation and verification of turbulence model were conducted previously (Nur Hazirah *et al.*, 2015). It was verified the best turbulence model to be used with 3-dimensional turning diffuser was Standard K and Epsilon (SKE) turbulence model. Since present study using the

ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

same Reynolds number tested as well as type of flow (turbulent), Enhanced Wall Treatment (EWT) were also adopted in present study model. Constructed mesh and calculation of first layer thickness were already been discussed in previous study (Nur Hazirah *et al*., 2015). Input parameters of ANSYS FLUENT used during simulation were summarized in Table-1.

Two modifications were made in present study. The airfoil was rotated 3 degree clockwise, enlarging the angle of attack to 20º. Another modification was rotating 3 degree anti-clockwise, reducing the angle of attack to 14º. Flow structure for each modification were compared with flow structure of preliminary airfoil (AOA=17º) at all planes as part of qualitative comparison. Pressure recovery coefficient was calculated using Equation 1 while flow uniformity was in terms of standard deviation was calculated using Equation 2. Both quantitative measures were calculated and compared together with mean outlet velocities extracted from simulation results.

Table-1. ANSYS FLUENT input parameters.

Parameters	Input
Time Viscous Wall Treatment	Steady Standard K-epsilon turbulence model Enhanced Wall Treatment (EWT)
Material Properties	
Fluid Density Viscosity Solid Density	Air 1.164 kg/m3 1.872e-05 kg/ms^{-1} Acrylic Plate 1180 kg/m3
Boundary Condition Inlet Outlet Wall **Discretization Scheme**	 Velocity inlet = 16.808m/s Pressure outlet No slip
Pressure Momentum Turbulent Kinetic Energy Turbulent Dissipation Rate	Second order Second order Second order upwind Second order upwind
Convergence Criteria Initialization method Reference frame	1.00E-05 Standard Absolute

$$C_p = \frac{2(P_{outlet} - P_{inlet})}{\rho V_{inlet}^{2}} \qquad (1)$$

Where

P_{outlet} = average static pressure at diffuser outlet (Pa)
P_{inlet} = average static pressure at diffuser inlet (Pa)
ρ = density of air, 1.228 (kg/m^3) (Adrian *et al*., 2000)
V_{inlet} = inlet mean velocity (m/s)

$$\sigma_u = \sqrt{\frac{1}{N-1}\sum_{i=1}^{N}(V_i - u_o)^2} \qquad (2)$$

Where:

N = number of measurement points
V_i = local outlet air velocity (m/s)
u_o = mean outlet air velocity (m/s)

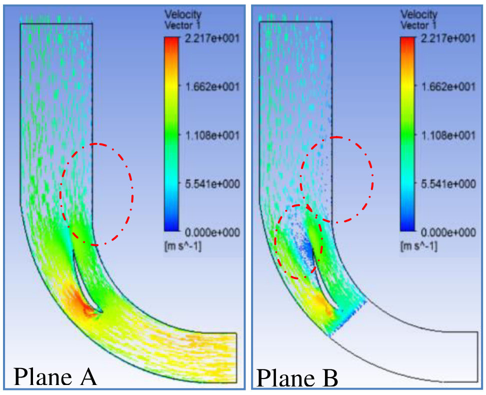
QUALITATIVE COMPARISON

The important key in studying flow structure comparison between modified airfoil and preliminary airfoil is the flow separation at the expected location. Since present study used 3-dimensional turning diffuser, it was expected that flow separation occurs at the inner wall as well as at both left and right wall. That is why, Plane A, Plane B and Plane C was selected previously. This plane could provide a better understanding on how the flow behaves inside the turning diffuser. Referring to Figure 5, flow structure for both angle of attack, AOA=20º and AOA=14º were compared to flow structure of preliminary design baffle, AOA=17º. Initially, AOA=17º has slight inner wall flow separation. When the baffle was modified to AOA=20º, inner wall separation can still be seen. Furthermore, large wake region at the trailing edge of the airfoil baffles were observed. Wake region is undesired in any study of flow around airfoil. It affects the total pressure loss in the system as well as disrupts flow uniformity. The results of this modification affected turning diffuser performance and can be clearly seen in quantitative analysis in next section.

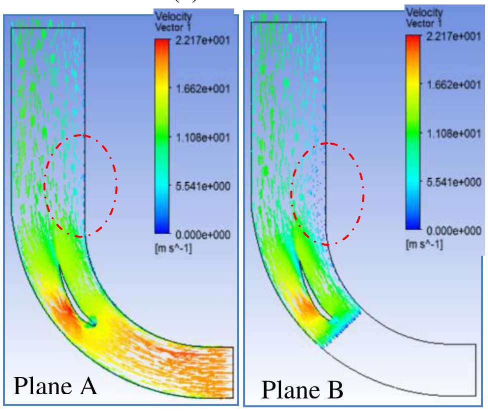
On the other hand, AOA=14º provide better flow structure than AOA=20º, even though it could not surpass flow structure of the preliminary design of baffle with AOA=17º. Flow separation at the inner wall for AOA=14º can still be seen, and even slightly more separation than AOA=17º. However, no wake region detected at the trailing edge.

ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

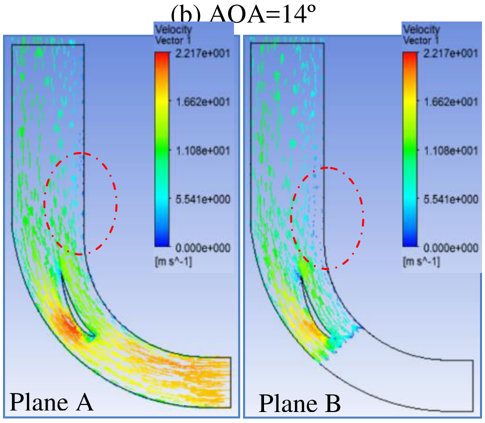
www.arpnjournals.com



(a) AOA=20º



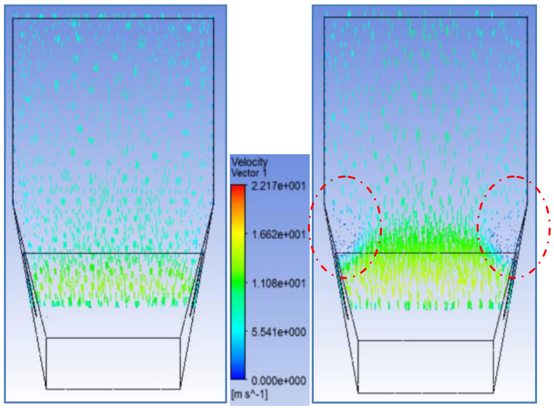
(b) AOA=14º



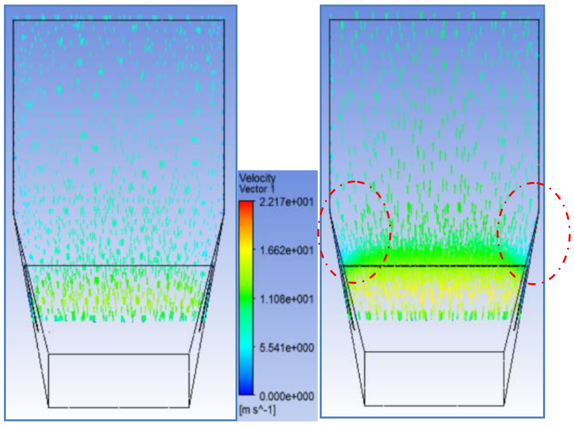
(c)AOA=17º

Figure-5. Flow structure comparison for all AOA tested at both Plane A and Plane B.

As for Plane C flow structure comparison, according to Figure-6, AOA=20º shows clear flow separation at both left and right wall. This supports the velocity profile plot (Nur Hazirah *et al.*, 2015) that concludes the low velocity at both left and right wall are due to flow separation. This modification affects the flow structure, and definitely will affect quantitative measures in next section. Fortunately, on the other hand, AOA=14º shows similar flow pattern as preliminary design baffle AOA=17º. Flow structure at all planes shows that AOA=14º was more favorable to be used together with 3-dimensional turning diffuser compared to AOA=20º.



(a) AOA=20º (right) and AOA=17º (left)



(b) AOA=14º (right) and AOA=17º (left)

Figure-6. Flow structure comparison for AOA=20º and AOA=14º with preliminary airfoil AOA=17º at Plane C.

Study and comparison on flow structure alone could not determine which baffle modification was better, but it can support the quantitative results discussed in next section. From Figure-5 and Figure-6, AOA=14º provide better flow structure compared to AOA=20º. Without the existing of wake region at the trailing edge as well as at both left and right wall, AOA=14º offers similar flow pattern as preliminary design baffle AOA=17º. Both flow separation and wake region seen in flow structure of AOA=20º would later on affect turning diffuser performance in terms of pressure recovery, C_p and flow uniformity, σ_u.

QUANTITATIVE COMPARISON

As explained previously, turning diffuser performance indicator was pressure recovery and flow uniformity. Higher value of C_p reflects higher pressure recovery while higher σ_u reflecting low flow uniformity. These calculated value were extracted from the simulation and compared with the preliminary design AOA=17º value as shown in Table-2 and Table-3. AOA=20º as expected did not perform better than AOA=17º since C_p reduce up to 13.46% while σ_u increase 13.73%. Flow separation at both left and right wall, as well as wake region at the trailing edge definitely causes these values to deteriorate. The abnormality of flow in AOA=20º resulting in higher pressure loss, thus affecting pressure recovery of the system. Non-uniformity at the outlet plane definitely disturb the total mean velocity, which reduce 29% from the preliminary design baffle AOA=17º.

However, with AOA=14º, pressure recovery drops only 9% while flow uniformity show almost similar to AOA=17º with 1.01% difference. However, total mean outlet velocity was still slightly lower than AOA=17º. Even though turning diffuser performance could not be improve entirely, the values was slightly better than AOA=20º.Modification of airfoil baffle could be further studied using angle of attack ranging from 14º to 17º since good flow uniformity were measured for both angle of attack.

Table-2. Quantitative comparison between modification (20º) and preliminary airfoil (17º).

Angle of attack	17º	20º	%
C_p	0.58	0.50	13.46
V_m	5.61	3.97	29.10
σ_u	2.72	3.15	13.73

Table-3. Quantitative comparison between modification (14º) and preliminary airfoil (17º).

Angle of attack	17º	14º	%
C_p	0.58	0.53	9.89
V_m	5.61	3.90	30.46
σ_u	2.72	2.74	1.01

CONCLUSIONS

In order to propose the optimum design of baffle for 3-dimensional turning diffuser, modifications were made on the preliminary design tested in the experiment conducted previously. Even though higher pressure recovery and flow uniformity were measured during the experiment, further modification can be made to improve the flow structure as well. Changing the angle of attack is one of the approaches that could be studied. Angle of attack 14º offers higher quality of flow structure as compared to angle of attack 20º, but still could not surpass the performance using preliminary design baffle with angle of attack 17º. Further modification can also be made by changing the thickness of airfoil, chord length and other parameters to find the optimum design that gives out the best performance for 3-dimensional turning diffuser.

ACKNOWLEDGEMENT

This work was financially supported by Short Term Grant vot 1289 of the Ministry of Higher Education, Malaysia.

REFERENCES

Fox, R.W. and Kline, S.J. 1962. Flow regime data and design methods for curved subsonic diffusers.J. Basic Eng. ASME, 84, pp. 303-312.

Normayati, N., Safiah, O., Raghavan, V. R., Mohd Faizal, M.B. 2011. Experimental Investigation of Pressure Losses and Flow Characteristics in Bend-Diffusers by Means of Installing Turning Baffles.International Conference of Mechanical Engineering, Putrajaya.

Normayati, N., Raghavan, V. R., Safiah, O. and Karim,Z.A.A. 2012. Numerical Investigation of Turning Diffuser Performance by Varying Geometric and Operating Parameters. Applied Mechanics and Materials Journal, 229-231, pp.2086-2093.

Normayati, N., Karim, Z. A. A., Safiah, O. and Raghavan V. R. 2013. Design and Development of Low Subsonic Wind Tunnel for Turning Diffuser Application. Advanced Material Research Journal. 614-615, pp. 586-591.

Normayati, N., Karim, Z.A.A., Safiah, O. and Raghavan V. R. 2014. The Performance of Turning Diffusers at Various Inlet Condition. Applied Mechanics and Materials Journal. 465-466(2014) pp. 597-602.

Lindgren, B. and Johansson, A.V. 2002. Design and EvaluationofaLow-Speed WindTunnel with Expanding Corners. Technical Report No. TRITA-MEK, 2002:14, Dept. of Mechanics, KTH, SE-100 44 Stockholm, Sweden.

Farsimadan, E. and Mokhtarzadeh-Dehghan, M.R. 2010. An Experimental Study of the Turbulence Quantities in the Boundary Layer and Near-Wake of an Airfoil Placed at Upstream of a 90° bend. Journal of Exp. Thermal and Fluid Science. 34, pp. 979-991.

Nakano, T., Fujisawa, N., Oguma, Y., Takagi, Y., and Lee, S. 2007. "Experimental Study on Flow and Noise Characteristics of NACA0018 Airfoil.Journal of Wind Engineering. 95, pp. 511-531.

Chong, T. P., Joseph, P. F. and Davies P.O.A.L. 2008. A parametric Study of Passive Flow Control for a Short, High Area Ratio 90 Degree Curved Diffuser. ASME J. Fluids Eng., 130(11), pp. 212-220.

Majumdar, B., Mohan, R., Singh, S. N. and Agrawal, D. P. 1998. Experimental Study of Flow in a High Aspect Ratio 90 Degree Curved Diffuser. ASME J. Fluids Eng. 120(1), pp. 83-89.

Adrian,R.J., Meinhart,C.D. and Tomkins,C.D. 2000. Vortex Organization inthe Outer Region ofthe Turbulent Boundary Layer. Journal of Fluid Mechanics. 422, pp. 1-54.

Guohui, G. and Saffa, B.R. 1996. Measurement and computational fluid dynamics prediction of diffuser pressure-loss coefficient. Applied Energy. 54(2), pp. 181-195.

Gopaliya, M.K., Kumar, M., Kumar, S. and Gopaliya, S.M. 2007. Analysis of performance characteristics of S-Shaped diffuser with offset. Aerospace Science and Technology. 11, pp. 130-135.

NurHazirah, N., Normayati, N., Safiah, O. and Vijay, R.R. 2014. Investigation of Flow Uniformity and Pressure Recovery in a Turning Diffuser by Means of Baffles. Applied Mechanics and Materials. 465-466, pp. 526-530.

Schreiber, H.A. and Steinert, W. 2004. Advanced High-Turning Compressor Airfoils for Low Reynolds Number Condition - Part II: Experimental and Numerical Analysis. Journal of Turbomachinery. 126, pp. 482-492.