2nd Biannual Post Graduate Conference
9th-11th November 2014, Seri Iskandar, Perak

Numerical Investigation of Turning Diffuser Performance: Validation and Verification

Normayati Nordin
Faculty of Mechanical and Manufacturing Engineering
Universiti Tun Hussein Onn Malaysia
Batu Pahat, Malaysia
mayati@uthm.edu.my

Zainal Ambri Abdul Karim
Department of Mechanical Engineering
Universiti Teknologi Petronas
Tronoh, Malaysia
ambri@petronas.com.my

Safiah Othman
Faculty of Mechanical and Manufacturing Engineering
Universiti Tun Hussein Onn Malaysia
Batu Pahat, Malaysia
safiah@uthm.edu.my

Vijay R. Raghavan
OYL Research & Development Centre
OYL Research & Development Sdn Bhd
Sungai Buloh, Malaysia
vijay@oyl.com.my

***Abstract*— This paper aims to validate the numerical method used to intensively study the performance of turning diffuser. A 90^o turning diffuser of area ratio, *AR* = 2.16 operated at inflow Reynolds numbers, Re_{in} = 5.786 x 10^4 and 1.775 x 10^5 was considered. Hybrid grid with a large number of grid points located at the inner and outer walls was generated. The applicability of k-ε turbulence models, i.e. standard k-ε (ske), renormalization group k-ε (rngke) and realizable k-ε (rke) by means of adopting appropriate near wall treatments to simulate the actual cases, was assessed. The enhanced wall treatment adopted ske appeared as the best validated model, producing minimal deviation with comparable flow structures to the actual cases.**

Keywords*— *CFD; turning diffuser; validation

INTRODUCTION

Diffusers are classified by their geometry. A diffuser that is introduced with no turn is known as a straight diffuser [1-3], whereas a diffuser introduced with certain angle of turn is called a turning diffuser or a curved diffuser [3-6]. Study of the geometry effect on diffuser performance has been of fundamental interest to researchers in the area of fluid mechanics since decades and it continues to grow [1-17].

The primary index used to measure the performance of a diffuser is outlet pressure recovery coefficient (C_p):

$$C_p = \frac{2(P_{out} - P_{in})}{\rho V_{in}^{\ 2}} \tag{1}$$

where,
P_{out}= outlet static pressure (Pa)
P_{in}= inlet static pressure (Pa)
ρ = flow density (kg/m^3)
V_{in} = inlet air velocity (m/s)

The value of C_p indicates how much kinetic energy is successfully converted to pressure energy. The main problem in achieving a high pressure recovery is flow separation, which results in non-uniform flow distribution and excessive energy losses. It is even worse, particularly when a 90^o turn together with a diffusing effect is applied. The flow through a turning diffuser with 90^o angle of turn is rather complex, apparently due to the expansion and sharp inflexion introduced along the direction of flow, causing strong adverse pressure gradient-driven streamwise vortices.

Computational fluid dynamics (CFD) as a tool has been widely employed by scientists and engineers in flow studies. The total dependence on experimental methods can be reduced by implementing the CFD techniques. There is basically a challenge in assigning the best model to represent the actual case when a complex flow is involved. The k-ε turbulence is the model introduced by Jones and Launder [7], which has been used widely in industry. This model along with appropriate setting of grid and wall boundary conditions managed to predict the onset flow separation accurately [2, 8]. There are several successful studies predicting the flow within a diffuser, which essentially employed k-ε turbulence model [2, 8-14].

Basically, the k- ε turbulence models are not valid in the near wall region. To work around this, standard wall functions, non-equilibrium wall functions or enhanced wall treatment should be applied. Standard wall functions by Launder and Spalding [15] are provided as a default option in ANSYS FLUENT, and have been widely used in industrial flows. Non-equilibrium wall functions are often applied to improve the results for flows with higher pressure gradient and mild separation [16]. Wall functions allow the use of relatively coarse mesh in the near wall region, 30< y^+ <300. Enhanced wall treatment is suitable for low-*Re* flows (Re<10^6) or flows with complex near wall phenomenon [16]. It requires very fine near-wall mesh, i.e. y^+≈1.0 capable of resolving the viscous sublayer to at least 10 cells within the inner layer.

In the present work, the applicability of k-ε turbulence models namely standard k-ε (ske), renormalization group k-ε (rngke) and realizable k-ε (rke) by means of adopting standard wall functions, non-equilibrium wall functions and enhanced wall treatment to simulate the flow within a turning diffuser are assessed. A 90° turning diffuser of area ratio, AR = 2.16 operated at inflow Reynolds numbers, Re_{in}=5.786 x 10^4 and 1.775 x 10^5 is considered.

I. PROJECT AND DATA MANAGEMENT

ANSYS Workbench offers a user friendly and better interface of project and data management. All the workflows including pre-processing, simulation and post-processing were linked one to another and viewed in the innovative project schematic, as shown in Fig. 1.

A computer with following specifications was used to run the simulations:

i. Intel (R) Core (TM) i7 Processor- 4700 HQ CPU @ 2.40 GHz 2.40 GHz
ii. Windows 8 of 64-bit operating system
iii. Installed memory of 8.0 GB RAM
iv. NVIDIA GeForce GTX 770M

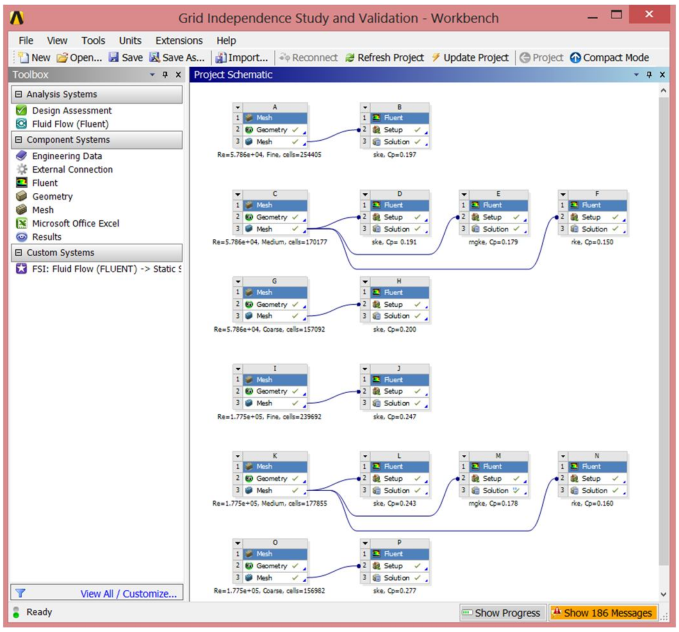


Fig. 1. Project schematic in Workbench

II. GEOMETRICAL DOMAIN AND BOUNDARY CONDITIONS

ANSYS DesignModeler was used to create the geometrical domain. As shown in Fig. 2, the inner-wall and center curves were constructed using quarter circles of radii 12 cm and 17.5 cm respectively. The outer-wall curve was shaped using circular-arcs tangent to the sequence of circles, thus an even area propagation between the inner and outer wall passages could be established relative to the center.

A three-dimensional flow domain in Fig. 3 was created by extruding the base object, i.e. solid line in Fig. 2 of 13 cm. The actual outlet was extended by a length equal to the center curve length, L_m to remedy the flow, after which the pressure could be considered as the atmospheric pressure.

Three types of boundary conditions were imposed. At the solid wall, the velocity was zero due to the no-slip condition. The inlet velocity, V_{in} respective to the Re_{in} were specified at 12.92 m/s and 39.66 m/s. This corresponded to the turbulent intensity, I_{in} of 4.1% and 3.5% respectively. At the outlet boundary, the pressure was set at the atmospheric pressure (0 gage pressure).

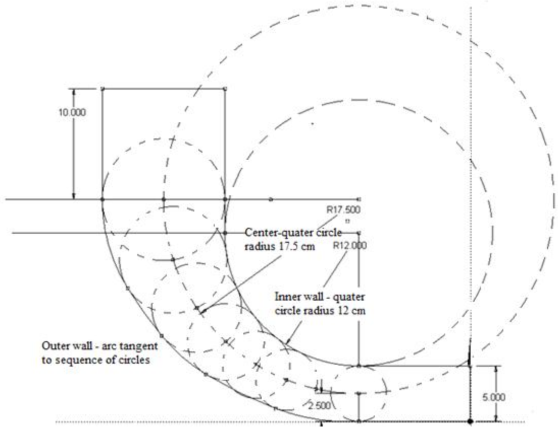


Fig. 2. Construction lines, i.e. dashed line of a 90° turning diffuser (all dimensions in centimeters)

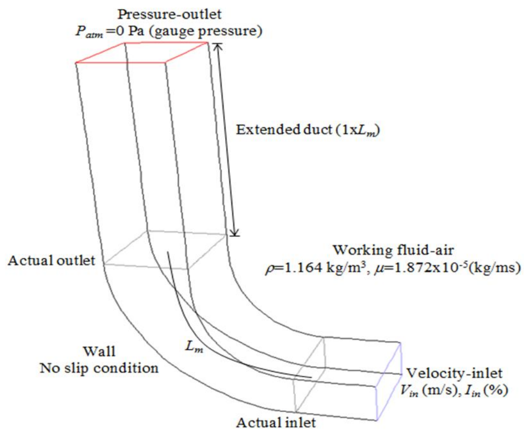


Fig. 3. Three dimensional flow domain

III. SOLVER SETTINGS

ANSYS Fluent 14.5 was used as a platform for the analysis. The flow was assumed to be incompressible, three-dimensional (x, y and z direction), fully-developed, steady state and isothermal. The gravitational effect was negligible. The Reynolds Average Navier Stokes (RANS) equations as follows were solved.

Continuity equation:

$$\frac{\delta u}{\delta x}+\frac{\delta v}{\delta y}+\frac{\delta w}{\delta z}=0 \tag{2}$$

x- momentum equation:

$$u\frac{\delta u}{\delta x}+v\frac{\delta u}{\delta y}+w\frac{\delta u}{\delta z}=-\frac{\delta P}{\delta x}+\mu\frac{\delta^2 u}{\delta x^2}+\mu\frac{\delta^2 u}{\delta y^2}+\mu\frac{\delta^2 u}{\delta z^2}-\frac{\delta(\overline{u'^2})}{\delta x}-\frac{\delta(\overline{u'v'})}{\delta y}-\frac{\delta(\overline{u'w'})}{\delta z}+S_{Mx} \tag{3}$$

y- momentum equation:

$$u\frac{\delta v}{\delta x}+v\frac{\delta v}{\delta y}+w\frac{\delta v}{\delta z}=-\frac{\delta P}{\delta y}+\mu\frac{\delta^2 v}{\delta x^2}+\mu\frac{\delta^2 v}{\delta y^2}+\mu\frac{\delta^2 v}{\delta z^2}-\frac{\delta(\overline{u'v'})}{\delta x}-\frac{\delta(\overline{v'^2})}{\delta y}-\frac{\delta(\overline{v'w'})}{\delta z}+S_{My} \tag{4}$$

z- momentum equation:

$$u\frac{\delta w}{\delta x}+v\frac{\delta w}{\delta y}+w\frac{\delta w}{\delta z}=-\frac{\delta P}{\delta z}+\mu\frac{\delta^2 w}{\delta x^2}+\mu\frac{\delta^2 w}{\delta y^2}+\mu\frac{\delta^2 w}{\delta z^2}-\frac{\delta(\overline{u'w'})}{\delta x}-\frac{\delta(\overline{v'w'})}{\delta y}-\frac{\delta(\overline{w'^2})}{\delta z}+S_{Mz} \tag{5}$$

The applicability of ske, rngke and rke turbulence models to close the RANS equations was verified. Pressure based solver with a robust pressure-velocity coupling scheme, SIMPLE was applied. The gradient was discretised by Green-Gauss Cell-based. As it involved high pressure gradients, pressure was discretised by PRESTO scheme. A 3rd order accuracy scheme, QUICK was used to discretise the convection terms, i.e. momentum, turbulent kinetic energy and turbulent dissipation rate owing to its proven capability to solve the flow in diffuser when hybrid mesh was applied. The convergence criterion was set to be 10^{-6}.

IV. GRID INDEPENDENCE STUDY

The grid was generated using ANSYS ICEM CFD with the size of wall-adjacent cell, y^+ was prescribed as follows:

$$y^+ = \frac{y\ u_\tau}{\nu} \tag{6}$$

where,

y = normal distance from the wall (m)

u_τ = friction velocity (m/s)

υ = kinematic viscosity (m^3/s)

The friction velocity was estimated as:

$$u_\tau = \sqrt{\frac{\tau_w}{\rho}} = V\sqrt{\frac{C_f}{2}} \tag{7}$$

where,

V= flow velocity (m/s)

$$\frac{C_f}{2} \approx 0.039\,\mathrm{Re}^{-1/4} \tag{8}$$

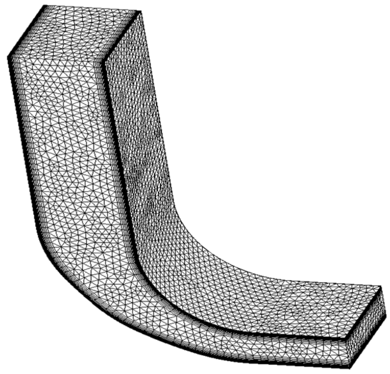


Fig. 4. Hybrid grid

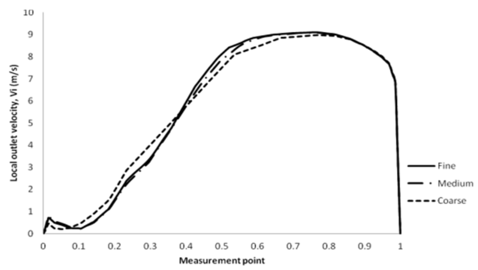
It is important to cautiously refine the grid along the inner wall and outer wall because these regions involve significant change of variables. Hexahedral mesh has been verified previously to provide the best continuity and fitted the curved geometries well [10, 11]. However, it was beyond the capacity of the computer in this study to generate uniform hexahedral mesh with adequate refinement to represent the actual flow. The adequate refinement particularly along the inner wall and outer wall was achieved merely by applying hybrid grid, i.e. tetrahedral and wedge elements as in Fig. 4.

For standard and non-equilibrium wall functions $30 < y^+ < 60$ was applied. Whereas, $0.5 < y^+ < 1.8$ was set for enhanced wall treatment, the average skewness of the elements produced was 0.2.

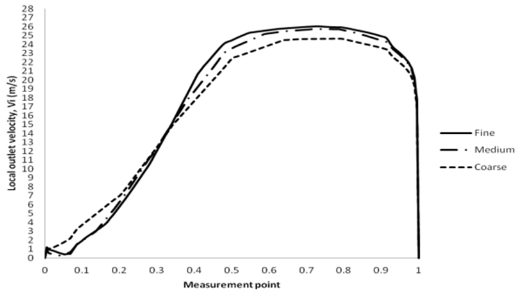
The grid independency was checked as depicted in Table 1. The ske turbulence model was applied for three kinds of grids, i.e. coarse, medium and fine by means of adopting standard, non-equilibrium and enhanced near wall treatments. The medium mesh was chosen as final meshing since it provided relatively less percentage of error with reasonable CPU time. Fig. 5 and 6 show the effect of refining the grid on the outlet velocity profiles extracted across the center of actual outlet. Basically, there was insignificant change of velocity profiles particularly between the medium and fine mesh.

Table 1. Grid independence study

Mesh	Standard and non-equilibrium wall functions						Enhanced wall treatment					
	5.786×10^4			1.775×10^5			5.786×10^4			1.775×10^5		
	Cells	C_p	Error (%)	Cells	C_p	Error (%)	Cells	C_p	Error (%)	Cells	C_p	Error (%)
Coarse	20763	0.528	2.5	25484	0.607	4.5	157092	0.200	1.5	156982	0.277	12.1
Medium	68153	0.513	0.4	87659	0.580	0.2	170177	0.191	3.0	177855	0.243	1.6
Fine	89729	0.515	-	123191	0.581	-	254405	0.197	-	239692	0.247	-

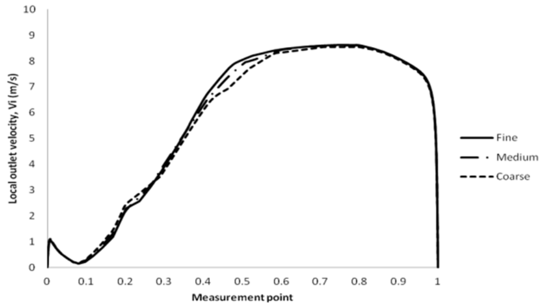


(a)

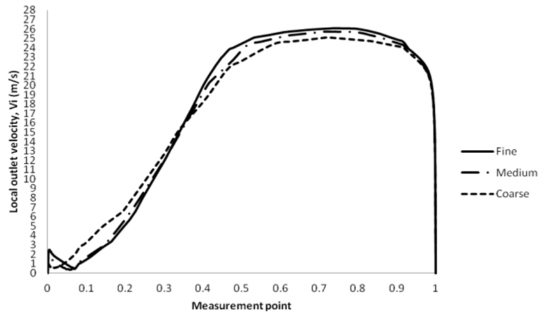


(c)

Fig. 5. Grid independence study for standard and non-equilibrium wall functions (a) Re_{in}=5.786 x 10^4 (b) Re_{in}=1.775 x 10^5



(a)



(b)

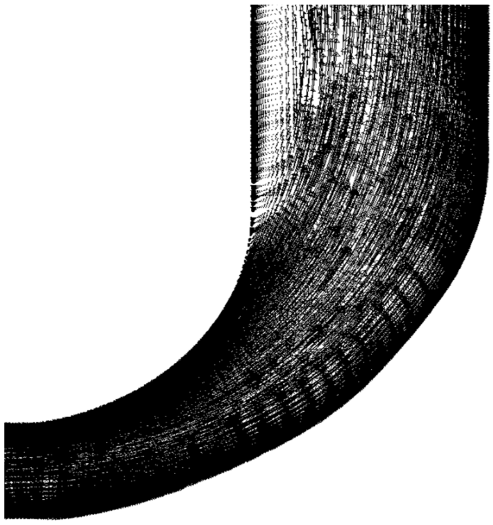
Fig. 6. Grid independence study for enhanced near wall treatment (a) Re_{in}=5.786 x 10^4 (b) Re_{in}=1.775 x 10^5

V. VALIDATION OF NUMERICAL METHOD

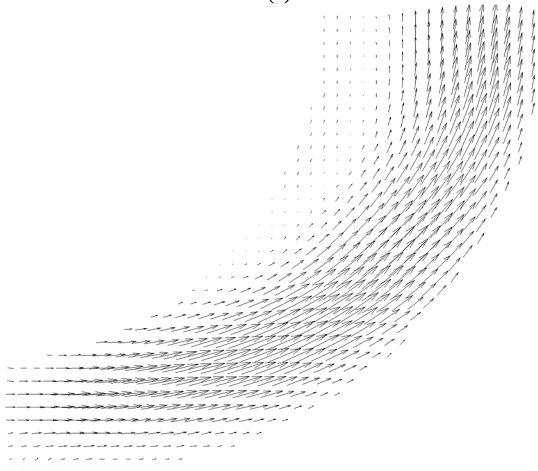
Validation of numerical work was carried out by comparing the simulation results with the experimental results [17]. The parameters considered for validation purpose are outlet pressure recovery coefficient, C_p and average velocity across the center of actual outlet, V_{avg}. Table 2 shows that the ske adopted enhanced wall treatment appears as the most optimum model, producing the least percentage of deviation for C_p= 0-1.7% and V_{avg}= 4.5-7.0%. Furthermore, as shown in Fig. 7 and 8, comparable flow structures with almost similar onset flow separation between numerical and experimental are obtained.

Table 2. Validation of numerical method

Re_{in}	Model	Near wall treatment	C_p	Deviation (%)	V_{avg} (m/s)	Deviation (%)
5.786 x 10^4	ske	Standard	0.513	168.6	5.30	41.0
	rngke		0.408	113.6	5.87	56.1
	rke		0.400	109.4	5.82	54.8
	ske	Non-equilibrium	0.572	199.5	5.16	37.2
	rngke		0.531	178.0	5.44	44.7
	rke		0.504	163.9	5.52	46.8
	ske	Enhanced	0.191	0.0	3.93	4.5
	rngke		0.179	6.3	4.21	12.0
	rke		0.150	21.5	4.43	17.8
	Exp [17]	-	0.191	-	3.76	-
1.775E+05	ske	Standard	0.580	142.7	14.43	34.9
	rngke		0.500	109.2	16.50	54.2
	rke		0.447	87.0	16.66	55.7
	ske	Non-equilibrium	0.590	146.9	14.48	35.3
	rngke		0.531	122.2	16.15	50.9
	rke		0.475	98.7	16.40	53.3
	ske	Enhanced	0.243	1.7	11.50	7.5
	rngke		0.178	25.5	14.17	32.4
	rke		0.160	33.1	13.86	29.5
	Exp [17]	-	0.239	-	10.70	-

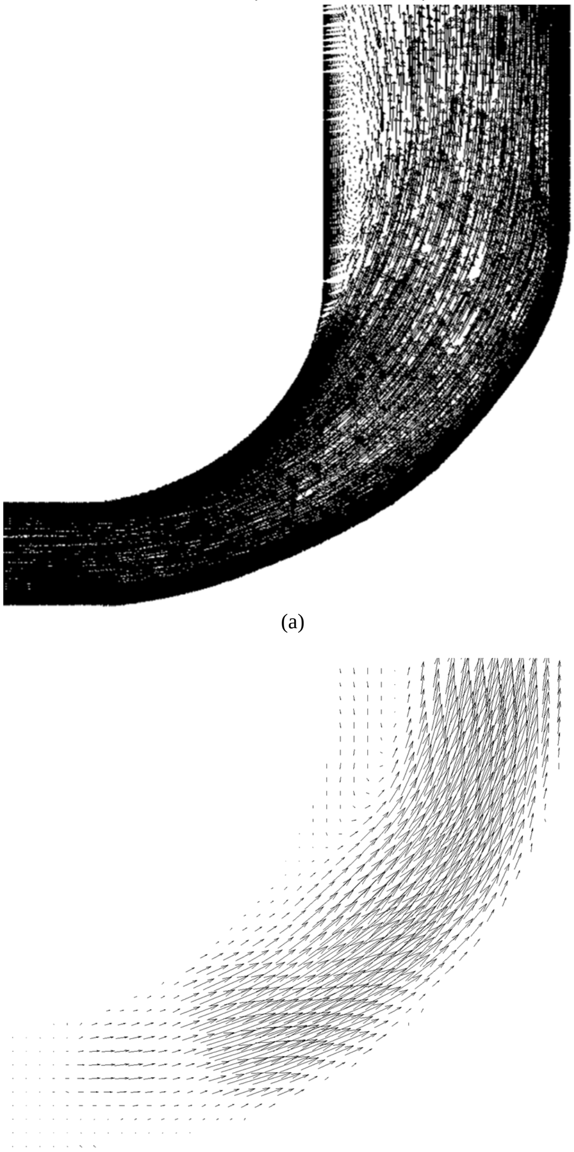


(a)



(b)

Fig. 8. Flow structure within the turning diffuser operated at Re_{in}=5.786 x 10^4 (a) ske + enhanced wall treatment and (b) experimental



(a)

(b)

Fig. 8. Flow structure within the turning diffuser operated at Re_{in}=1.775 x 10^5 (a) ske + enhanced wall treatment and (b) experimental

VI. CONCLUSION

In conclusion, the current work validates the numerical method used to intensively study the performance of turning diffusers. The ske adopted enhanced wall treatment appeared as the best model to represent the actual cases.

ACKNOWLEDGMENT

This work was financially supported by the Fundamental Research Grant Scheme (FRGS) of the Ministry of Higher Education, Malaysia.

REFERENCES

[1] G. Gan and S.B. Riffat, “Measurement and computational fluid dynamics prediction of diffuser pressure-loss coefficient,” *Applied Energy*, vol. 54(2), pp. 181-195, 1996.

[2] W.A. El-Askary and M. Nasr, “Performance of a bend diffuser system: Experimental and numerical studies,” *Computer & Fluids,* vol. 38, pp. 160-170, 2009.

[3] R.K. Sullerey, B. Chandra, and V. Muralidhar, "Performance comparison of straight and curved diffusers," *J. of Def. Sci.,* vol. 33, pp. 195-203, 1983.

[4] N. Nordin, V.R. Raghavan, S. Othman and Z.A.A. Karim,“Compatibility of 3-D turning diffusers by means of varying area ratios and outlet-inlet configurations", *ARPN Journal of Engineering and Applied Sciences*, Vol. 7, No. 6, pp 708-713, 2012.

[5] T.P. Chong, P.F. Joseph and P.O.A.L. Davies, “A parametric study of passive flow control for a short, high area ratio 90 deg curved diffuser,” *J. Fluids Eng.*, vol. 130, 2008.

[6] C.J. Sagi and J.P. Johnson, “The design and performance of two-dimensional, Curved Diffusers,” *J. Basic Eng. ASME,* vol. 89, pp. 715-731, 1967.

[7] W.P. Jones, B.E., Launder, “The calculation of low-reynolds number phenomena with a two equation model of turbulence,” *Int. J. Heat Mass Transfer*, vol. 6, pp. 1119-1130, 1973.

[8] D. Xu, M. A. Leschziner, B. C. Khoo, and C. Shu, "Numerical prediction of separation and reattachment of turbulent flow in axisymmetric diffuser," *Computers & Fluids,* vol. 26, pp. 417-423, 1997.

[9] Y.T. Yang and C. F. Hou, "Numerical calculation of turbulent flow in symmetric two-dimensional diffusers," *Acta Mechanica,* vol. 137, pp. 43-54, 1999.

[10] M.K. Gopaliya, M. Kumar, S. Kumar, and S. M. Gopaliya, "Analysis of performance characteristics of S-shaped diffuser with offset," *Aerospace Science & Tech.,* vol. 11, pp. 130-135, 2007.

[11] M. K. Gopaliya, P. Goel, S. Prashar, and A. Dutt, "CFD Analysis of performance characteristics of S-shaped diffusers with combined horizontal and vertical offsets," *Computer & fluids,* vol. 40, pp. 280-290, 2011.

[12] I.H. Ibrahim, E.Y.K. Ng, K. Wong, and R. Gunasekaran, "Effects of centerline curvature and cross-sectional shape transitioning in the subsonic diffuser of the f-5 fighter jet," *J. Mechanical and Science Technology*, vol. 22, pp. 1993-1997, 2008.

[13]S. Jakirlic, G. Kadavelil, M. Kornhaas, M. Schäfer, D.C. Sternel, and C. Tropea, "Numerical and physical aspects in LES and hybrid LES/RANS of turbulent flow separation in a 3-D diffuser," *International Journal of Heat and Fluid Flow,* vol. 31, pp. 820-832, 2010.

[14] N. Nordin, V. R. Raghavan, S. Othman and Z. A. A. Karim, “Numerical investigation of turning diffuser performance by varying geometric and operating parameters”, *Applied Mechanics and Materials Journal*, Vol. 229-231, pp. 2086-2093, 2012.

[15] B. E. Launder and D. B. Spalding, "The numerical computation of turbulent flows". *Computer Methods in Applied Mechanics and Engineering*, vol. 3, pp. 269–289, 1974.

[16] ANSYS FLUENT User’s Guide, release 14.0, Canonsburg, USA, 2011.

[17] N.Nordin, Z,A.A. Karim, S.Othman and V.R. Raghavan, “The performance of turning diffusers at various inlet conditions”. *Applied Mechanics and Materials Journal,* vol. 465-466, pp. 597-602, 2014.