Numerical Investigation of Turning Diffuser Performance by Varying Geometric and Operating Parameters

Normayati Nordin[1, a], Vijay R. Raghavan[2], Safiah Othman[1]

and Zainal Ambri Abdul Karim[3]

[1]Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia, Parit Raja, 86400 Batu Pahat, Johor, Malaysia

[2]OYL R&D Center Sdn Bhd, Selangor, Malaysia

[3]Department of Mechanical Engineering, Universiti Teknologi PETRONAS, Tronoh, Perak, Malaysia

[a] mayati@uthm.edu.my

Abstract: This paper presents a numerical investigation of pressure recovery and flow uniformity in turning diffusers with 90° angle of turn by varying geometric and operating parameters. The geometric and operating parameters considered in this study are area ratio (AR= 1.6, 2.0 and 3.0) and inflow Reynolds number (Re_{in}=23, 2.653E+04, 7.959E+04, 1.592E+05 and 2.123E+05). Three turbulence models, i.e. the standard k-ε turbulence model (std k-ε), the shear stress transport model (SST-k-Ω) and the Reynolds stress model (RSM) were assessed in terms of their applicability to simulate the actual cases. The standard k-ε turbulence model appeared as the best validated model, with the percentage of deviation to the experimental being the least recorded. Results show that the outlet pressure recovery of a turning diffuser at specified Re_{in} improves approximately 32% by varying the AR from 1.6 to 3.0. Whereas, by varying the Re_{in} from 2.653E+04 to 2.123E+05, the outlet pressure recovery at specified AR turning diffuser improves of approximately 24%. The flow uniformity is considerably distorted with the increase of AR and Re_{in}. Therefore, there should be a compromise between achieving the maximum pressure recovery and the maximum possible flow uniformity. The present work proposes the turning diffuser with AR=1.6 operated at Re_{in}=2.653E+04 as the optimum set of parameters, producing pressure recovery of C_p=0.320 and flow uniformity of σ_u=1.62, with minimal flow separation occurring in the system.

Introduction

There are various types of diffusers which are commonly classified by their geometries and applications. Study of the geometric and operating parameters that affect on the diffuser performance has been of fundamental interest to researchers in the area of fluid mechanics since decades and it continues to grow [1]-[12]. Basically, the performance of a diffuser is evaluated in terms of its pressure recovery and flow uniformity. The maximum possible pressure recovery and flow uniformity can be obtained by setting the geometric and operating parameters optimally. In the present work, a turning diffuser with 90° angle of turn is considered. Basically, the inner wall is subjected to the curvature induced effects, where under a strong adverse pressure gradient, the boundary layer on the inner wall (*i.e.*convex region) is likely to separate, and the core flow tends to deflect to the outer wall (*i.e.* concave region) [1], [7]. This eventually leads to the formation of pressure-driven secondary flows that thicken the inner wall boundary layer and makes it susceptible to flow separation.

Computational fluid dynamics (CFD) as a tool has been widely employed by scientists and engineers in flow studies. The total dependance 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 particularly when involving a complex flow [10]. Bourgeois *et al.* [10] have considered four turbulence models to predict the mean flow fields of a fish tail-shape diffuser that was incorporated in the aero engine centrifugal compressor. The models analysed were the k-ε turbulence model, the shear stress transport (SST) model, a proposed modification SST model denoted as the SST-reattachment modification (SST-RM) and the Speziale, Sarkar and Gatski Reynolds stress model (RSM-SSG). The RSM-SSG and SST models gave more accurate predictions of mean flow fields as compared to the k-ε turbulence model and the shear stress transport (SST) model.

The standard k- ε turbulence is the model introduced by Jones and Launder [11], which has been used widely in industry. This model has managed to predict the onset flow separation accurately [3]. There are several successful studies for predicting the flow within a diffuser, which have employed k-ε turbulence model [3], [8], [12]. The k-Ω and k- ε turbulence models have been used by Ibrahim *et al.* [13] to investigate the effect of curvature and cross-sectional transitioning on the performance of S-shape diffuser. Both models have shown good agreement with the experimental data. However, the shape of the core flow obtained by the k-Ω model has provided slighly better accuracy.

The current work uses the commercial CFD code FLUENT to model and simulate the performance of turning diffusers. Three potential turbulence models, *i.e.* the standard k-ε turbulence model (std k-ε), the shear stress transport model (SST k-Ω) and the Reynolds stress model (RSM), are assessed and validated with the experimental data. The best validated model is used to further investigate the effect of varying geometric and operating parameters on the pressure recovery and flow uniformity of turning diffusers. The optimum geometric and operating parameters setting that leads to a compromise between pressure recovery and flow uniformity is suggested.

Geometric and Operating Parameters

The turning diffuser with 90° angle of turn is considered in this study. The diffuser inlet is 130 mm x 50 mm. As shown in Figure 1 both inner and centerline are quadrant of a circle. The outer wall is shaped such that equal area distributions are established for the inner wall and outer wall relative to the centerline. In this study, the geometric and operating parameters considered are area ratio (*i.e.* AR=1.6, 2.0 and 3.0) and inflow Reynolds number (*i.e.* Re_{in}=23, 2.653E+04, 7.959E+04, 1.592E+05 and 2.123E+05).
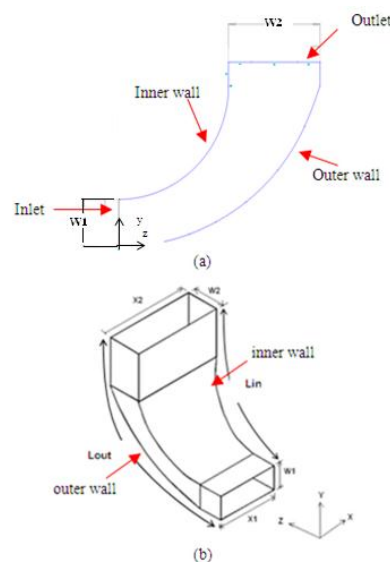

Fig. 1 - (a) 2-D geometric layout of turning diffuser with 90° angle of turn (b) 3-D geometric layout of turning turning diffuser with 90° angle of turn

Performance Parameters and Governing Equations

The performance of turning diffuser is evaluated in terms of outlet pressure recovery coefficient (C_p) and flow uniformity index (σ_u). C_p represents the kinetic energy that has been converted into pressure energy due to diffusing action,

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

where,

P_{outlet} = average static pressure at diffuser outlet (Pa)

P_{inlet} = average static pressure at diffuser inlet (Pa)

ρ = air density (kg/m^3)

V_{inlet} = inlet air velocity (m/s)

The flow uniformity is evaluated by calculating standard deviations (σ_u) of outlet velocity. The least of absolute deviation corresponds to the greatest uniformity of flow. Standard deviation (σ_u) can be expressed as,

$$\sigma_u = \sqrt{\frac{1}{N-1} \sum_{i=1}^{N} (V_i - V_{ave})^2} \tag{2}$$

where,

N = number of measurement points

Vi = local velocity (m/s)

V_{ave} = average velocity (m/s)

Besides that, the performance of turning diffuser can also be described by means of the overall loss coefficient (K),

$$K = 1 - Cp \tag{3}$$

The flow is assumed to be incompressible, two-dimensional (y and z direction), fully-developed, steady state and isothermal. The gravitational effect is negligible. By applying the boundary conditions, Reynolds Average Navier Stokes (RANS) equations as following are solved,

Continuity equation:

$$\frac{\delta v}{\delta y} + \frac{\delta w}{\delta z} = 0 \tag{4}$$

y- momentum equation:

$$v \frac{\delta v}{\delta y} + w \frac{\delta v}{\delta z} = -\frac{1}{\rho} \frac{\delta P}{\delta y} + \upsilon \frac{\delta^2 v}{\delta y^2} + \upsilon \frac{\delta^2 v}{\delta z^2} - \frac{\delta(\overline{v'^2})}{\delta y} - \frac{\delta(\overline{v'w'})}{\delta z} + S_{My} \tag{5}$$

z- momentum equation:

$$v \frac{\delta w}{\delta y} + w \frac{\delta w}{\delta z} = -\frac{1}{\rho} \frac{\delta P}{\delta z} + \upsilon \frac{\delta^2 w}{\delta y^2} + \upsilon \frac{\delta^2 w}{\delta z^2} - \frac{\delta(\overline{v'w'})}{\delta y} - \frac{\delta(\overline{w'^2})}{\delta z} + S_{Mz} \tag{6}$$

Computational Details

The 2-D geometric model, as shown in Figure 1, is created in GAMBIT 2.4.6. Triangular meshing shape with a pave scheme is generated, forming total of 3599 cells. The commercial CFD code FLUENT 6.3.26 is used to simulate the model by adopting the SIMPLE algorithm solution. Pressure is discretised using second order scheme, whereas momentum and the rest of turbulence parameters are discretised by means of second order upwind scheme. A convergence criterion of 1E-05 is applied. The description of applied boundary and operating conditions is in Table 1.

Validation between Computational and Experimental Works

The applicability of three turbulence models, *i.e.* the std k-ε model, the SST k-Ω model and the RSM model, to simulate the flow in turning diffuser is assessed. The mean and outlet velocity profile obtained numerically is validated with the experimental results. A turning diffuser with AR = 2.0 operated at low inflow Reynolds number, Re_{in}=23 is considered for the validation. Despite a slight difference to the experimental, all the models manage to simulate meaningful and explainable outlet velocity profiles (see Figure 2(a)). The outlet velocity is fairly distributed when the Re_{in} is low. However, there is insufficient pressure energy built up, thus the diffuser makes a great loss of approximately K=5.9. By introducing the high Re_{in} of 2.653E+05 the loss reduces to 0.6. Nevertheless, the distribution of the outlet velocity is relatively disrupted (see Figure 2(b)). As

depicted in Table 2, the percentage of deviation between numerical and experimental result is considerably large up to 34.1%. This is mainly due to the assumption made in the simulation which the inlet velocity is fully developed and uniform. However, this is not the case in the actual as the inlet velocity is fairly distorted due to the imperfect duct joint and the abrupt change of the inlet cross-section introduced as shown in Figure 3(a) and (b).

Table 1- Description of boundary and operating conditions

Inlet:		
i	Type of boundary	Velocity-inlet
ii	Inflow Reynolds number (Re_{in})	23 (0.004 m/s)
		2.653E+04 (5 m/s)
		7.959E+04 (15 m/s)
		1.592E+05 (30 m/s)
		2.123E+05 (40 m/s)
iii	Inlet turbulence intensity, I_{in} (%)	$16(Re_{in})^{-1/8}$
iv	Inlet hydraulic diameter, D_{hin} (m)	$4A_{in}/P_{in}$
Outlet:		
i	Type of boundary	Pressure-inlet
ii	Pressure specified	0 Pa gage pressure
iii	Outlet turbulence intensity, I_{out} (%)	$16(Re_{out})^{-1/8}$
iv	Outlet hydraulic diameter, D_{hout} (m)	$4A_{out}/P_{out}$
Inner and outer wall:		
i	Type of boundary	Smooth wall
ii	Shear condition	No- slip
Working fluid properties:		
i	Working fluid	Air
ii	Temperature (^{o}C)	30
iii	Density of working fluid (kg/m^3)	1.164
iv	Viscosity of working fluid (kg/ms)	1.872E-05

Table 2- Comparison of mean outlet velocity for numerical and experimental works

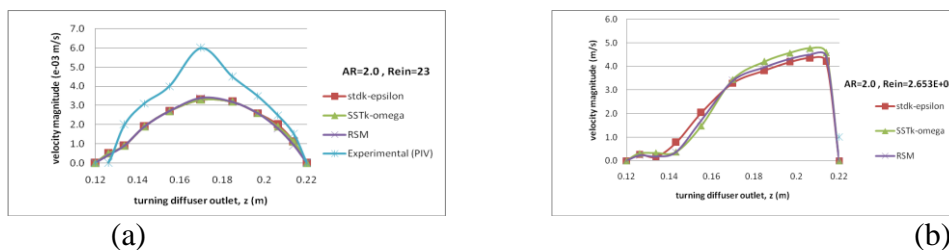| CFD models	Mean outlet velocity ($x10^{-3}$ m/s)		
	Computational	*Experimental*	*Deviation (%)*
Std k-ε	1.642		33.3
SST k-Ω	1.640	2.464	33.4
RSM	1.624		34.1

Fig. 2 (a) Comparison between outlet velocity profiles obtained numerically, i.e. using std k- ε, SST k-Ω, and RSM models, with the experimental (b)- Outlet velocity profile obtained numerically using std k- ε, SST k-Ω, and RSM models in the case of high Re_{in}
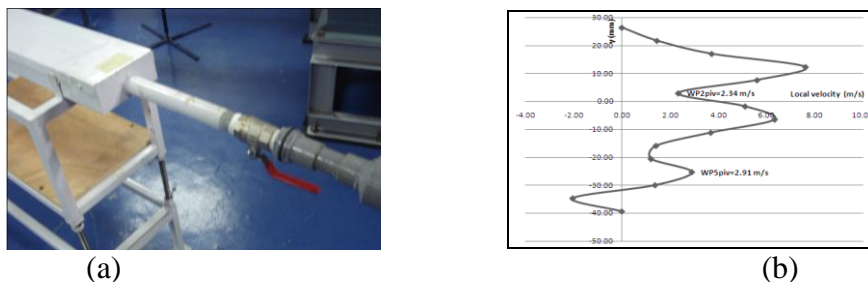

Fig. 3 - (a) The abrupt change of inlet cross section introduced before the test section disrupts the inlet velocity (b) The actual inlet velocity is not perfectly developed

Improvement to the existing rig is thus proposed. The settling chamber with multiple screens arrangement and contraction cone will be installed before the diffuser inlet. This is to ensure the flow entering diffuser is fully developed and uniformly distributed. The contraction cone is designed based on the fifth order polynomial proposed by Bell and Mehta [14]. In the present work, the std k-ε turbulence model appears as the best validated model since the percentage of deviation to the experimental is the least recorded. In fact, this model has been proven adequate to represent the flow phenomenon in diffusers [3], [8], [10], [12]. Therefore, further simulations to predict the effects of geometric and operating parameters on the pressure recovery and flow uniformity of turning diffuser are performed using this model. The closure coefficients applied are $C_\mu=0.09$, $C_{\varepsilon1}=1.44$, $C_{\varepsilon2}=1.92$, $\sigma_k=1.0$ and $\sigma_\varepsilon=1.3$.

Effect of Area Ratio (AR) and Inflow Reynolds Number (Re_{in}) on Pressure Recovery (Cp) Results

Table 3 shows the effect of varying AR and Re_{in} on the outlet pressure recovery. Negative pressure recovery is obtained when the low inflow Reynolds number, $Re_{in}=23$ is applied. Due to insufficient pressure energy provided at the inlet, the system fails to sustain the flow to get a good recovery.

Table 3. Effect of varying area ratio (AR) and inflow Reynolds number (Rein) on outlet pressure recovery (Cp)

Area ratio (AR)	Outlet pressure recovery (C_p)					Maximum percentage of improvement (%)
	$Re_{in}=23$	$Re_{in}=2.653E+04$	$Re_{in}=7.959E+04$	$Re_{in}=1.592E+05$	$Re_{in}=2.123E+05$	
1.6	-5.744	0.320	0.368	0.392	0.403	26
2.0	-4.861	0.373	0.425	0.448	0.458	23
3.0	-4.023	0.422	0.492	0.515	0.524	24
Maximum percentage of improvement (%)		32	34	31	30	

According to Wang et al. [9], the high inflow Reynolds number, $Re_{in}>500$ should be used in order to improve the recovery. Hence, the effect of high Re_{in} on pressure recovery is emphasized more in this work. Results show that by varying the Re_{in} from 2.653E+04 to 2.123E+05, the outlet pressure recovery for a specified AR turning diffuser improves approximately 24% on average. On the other hand, by varying the AR of turning diffuser from 1.6 to 3.0 at specified Re_{in} the outlet pressure recovery improves approximately 32% on average. Therefore, the effect of varying AR is more significant than the effect of varying Re_{in} on the improvement of pressure recovery.

Effect of Area Ratio (AR) and Inflow Reynolds Number (Re_{in}) on Flow Uniformity (σ_u) Results

As depicted in Table 4, the flow uniformity is extremely distorted with the increase of Re_{in} from 2.653E+04 to 2.123E+05 at specified AR maximum up to 669%. On the other hand, the increase of AR from 1.6 to 3.0 at specified Re_{in} disrupts the flow maximum only up to 15%. The flow uniformity at the turning diffuser outlet is considerably distorted, with the core flow deflected much to the outer wall section when the large AR and high Re_{in} is introduced (see Figure 8). Under a strong adverse pressure gradient, the boundary layer on the inner wall is thickened. Ultimately, the flow detachment occurs because of the fluid particles at the near inner wall region experience a greater retarding shearing force than the pressure pushing it. Overall, the increase of AR less affects the flow uniformity, as compared to the increase of Re_{in}.

Table 4. Effect of varying area ratio (AR) and inflow Reynolds number (Rein) on flow uniformity(σu)

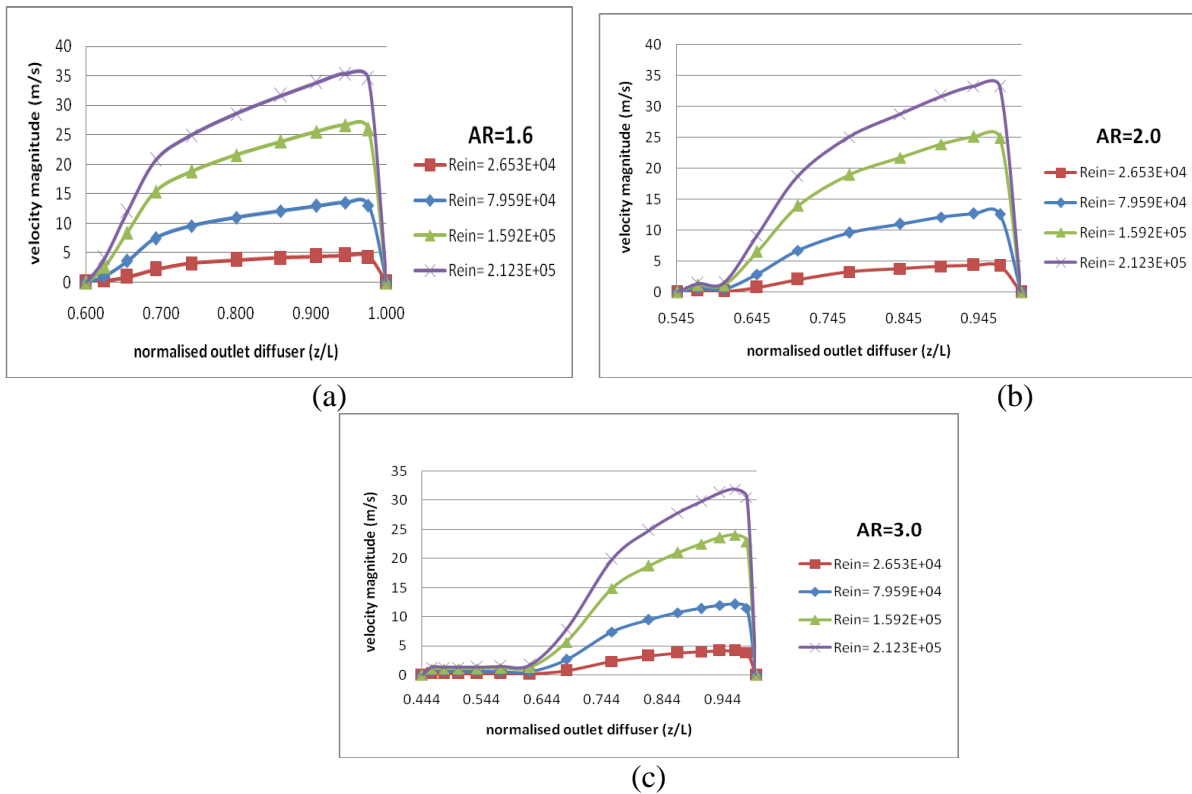| Area ratio (AR)	Standard deviation, σ_u (m/s)					Maximum percentage of distortion (%)
	$Re_{in}=23$	$Re_{in}=2.653E+04$	$Re_{in}=7.959E+04$	$Re_{in}=1.592E+05$	$Re_{in}=2.123E+05$	
1.6	0.0013	1.62	4.58	8.87	11.56	614
2.0	0.0011	1.72	4.97	9.76	12.92	651
3.0	0.0008	1.73	5.05	10.01	13.30	669
Maximum percentage of distortion (%)		7	10	13	15	

Fig. 8- The effect of high Re_{in} on the outlet velocity profile of turning diffuser (a) AR=1.6, (b) AR=2.0 and (c) AR=3.0

Compromise between Pressure Recovery and Flow Uniformity

The maximum outlet pressure recovery could be obtained by adopting the large AR turning diffuser, that is operated at high Re_{in}. However, this measure particularly whereby increasing the Re_{in} could substantially distort the flow uniformity. A compromise between the best produced pressure recovery and the maximum possible flow uniformity is always on the top when designing a turning diffuser. Figure 9 shows that by maximizing the area ratio from 1.6 to 3.0, considerable flow separation occurs on the inner wall, particularly when the AR=3.0 is considered, producing the maximum non-uniformity of σ_u=13.3. Despite the best recovery provided, the turning diffuser with AR=3.0, operated at Re_{in}= 2.123E+05 cannot not be considered as the optimum design due to major flow disruption. Basically, the flow separation is undesirable in many fluid systems as it would increase the pressure drag, decrease the core flow area, reduce the handling stability, generate noise and enhance the structural vibration [7]. For a turning diffuser with AR=2.0, the flow separation starts to occur even when it is operated at Re_{in}= 2.653E+04.

As shown in Figure 9 (a), a turning diffuser with minimum area ratio of 1.6 seems to have relatively minor flow separation with the least non-uniformity of σ_u=1.62 recorded when it is operated at Re_{in}=2.653E+04. The outlet pressure recovery provided by this system is C_p=0.320. Guohui and Saffa [3] have proposed that the optimum area ratio for a straight pyramidal diffuser was from 1.73 to 1.95, giving the maximum pressure recovery of C_p=0.480, with no flow separation occurred. In the present work, the maximum possible pressure recovery of C_p=0.403, with minimal flow separation could be provided by turning diffuser of AR=1.6, operated at Re_{in}= 2.123E+05. However, the flow uniformity is highly distorted, σ_u=11.56.

Hence, a turning diffuser with AR=1.6 operated at Re_{in}=2.653E+04 is presently proposed to be the most optimum set of geometric and operating parameter. With minimal flow separation occurred, this system could provide the maximum possible pressure recovery of C_p=0.320 without so much affecting the flow uniformity, σ_u=1.62.
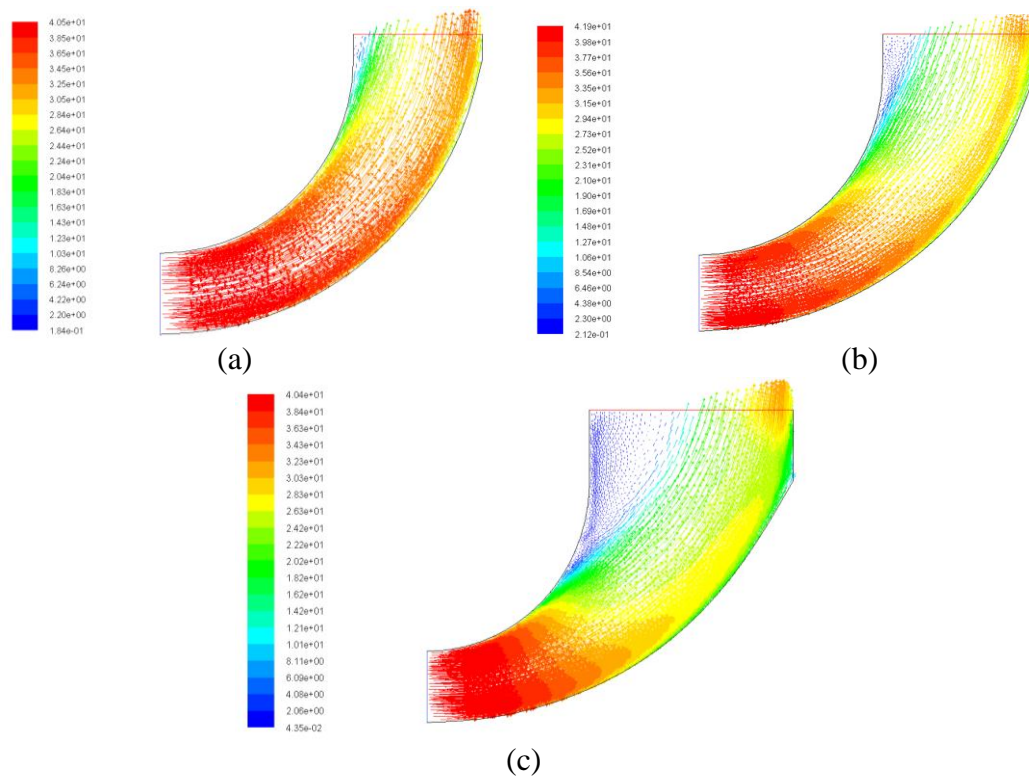
Fig. 9- Flow structure of turning diffuser with (a) AR=1.6, (b) AR=2.0 and (c) AR=3.0 operated at Re_{in} =2.123E+05

Conclusion & Recommendation

The effect of geometric and operating parameters (*i.e.* AR=1.6, 2.0, 3.0 and Re_{in}=23, 2.653E+04, 7.959E+04, 1.592E+05, 2.123E+05) on the pressure recovery and flow uniformity of turning diffuser with 90° angle of turn was investigated numerically. On the whole, the use of a large AR turning diffuser that is operated at high Re_{in} could give a favourable effect to the pressure recovery. However, the flow uniformity is considerably distorted. Therefore, a trade-off between good pressure recovery and flow uniformity has to be sought.

The present work proposes the turning diffuser with AR=1.6 operated at Re_{in}=2.653E+04 as the optimum geometric and operating parameters set, producing the pressure recovery of 0.320 and flow uniformity of 1.62 with minimal flow separation occurring in the system. In future, several more geometric and operating parameters will be considered. Besides, the effect of inner wall length (L_{in}/W_1) on the performance of turning diffuser will be in detail investigated.

Acknowledgements

References

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

[2] 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.

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

[4] B. Majumdar and D.P. Agrawal, "Flow characteristics in a large area ratio curved diffuser," *Proc. Instn. Mech.* Engrs.,vol. 210, pp.65, 1996.

[5]E.G. Tulapurkara, A.B. Khoshnevis and J.L. Narasimhan, "Wake boundary layer interaction subject to convex and concave curvatures and adverse pressure gradient,"*Exp. In Fluids*, vol. 31, pp. 697-707, 2001.

[6] C.K. Nguyen, T.D. Ngo, P.A. Mendis and J.C.K. Cheung, "A flow analysis for a turning rapid diffuser using CFD," *J. Wind Eng.*, vol. 108, pp. 749-752, 2006.

[7] 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.

[8] 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.

[9] Y.C. Wang, J.C. Hsu and Y.C., Lee, "9],"Loss characteristics and flow rectification property of diffuser valves for micropump applications" *Int. J. of Heat and Mass Transfer*, vol. 52, pp. 328-336, 2009.

[10] J.A. Bourgeois, R.J. Martinuzzi, E. Savory, C. Zhang and D. A.Roberts, "Assessment of turbulence model predictions for an aero-engine centrifugal compressor" *J. Turbomachinery*, vol.133, pp.1-15, 2010.

[11] 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.

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

[13] 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.

[14] J.H. Bell and R.D. Mehta, "Contraction design for low-speed wind tunnels", NASA 177488, Contract NAS2-NCC-2-294, 1988.

Mechanical and Electrical Technology IV

10.4028/www.scientific.net/AMM.229-231

Numerical Investigation of Turning Diffuser Performance by Varying Geometric and Operating Parameters

10.4028/www.scientific.net/AMM.229-231.2086