Penerbit Akademia Baru

CFD Letters

Journal homepage: www.akademiabaru.com/cfdl.html
ISSN: 2180-1363

Numerical Investigation of 180° Curved Diffuser Performance by Varying Geometrical and Operating Parameters

Open Access

Shamsuri Rasidi[1], Suzairin Md Seri[1], Normayati Nordin[1,*], Muhammad Zahid Shariff[1], Nurul Fitriah Nasir[1], Sharifah Adzila[1], Raudhah Othman[1]

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

ARTICLE INFO

Article history:
Received 21 May 2020
Received in revised form 20 July 2020
Accepted 25 July 2020
Available online 31 July 2020

Keywords:
Pressure recovery; flow uniformity; CFD; curved diffuser

ABSTRACT

The geometry of a curved diffuser often inviting the flow to separate. The flow separation phenomenon plays a vital role in the recovery of pressure inside the diffuser. Besides, the geometrical and operating aspects of a curved diffuser have always been taken into consideration in most relatable researches. Pressure recovery and flow uniformity of a steady, developed entering turbulent flow of a circular-sectioned $180°$ curved diffuser has been investigated numerically by varying geometrical and operating parameters. The curved diffuser with area ratio (2, 3 and 4), radius of inlet and curvature of respectively 50 mm and 180 mm was considered. Different values of Reynolds number $6 \times 10^4, 12 \times 10^4, 18 \times 10^4, 24 \times 10^4$, and 30×10^4 were tested in every area ratio set. ANSYS code Fluent was used to run the simulation by considering different turbulence models, i.e. the standard k-ε turbulence model (std k- ε), Spalart-Allmaras turbulence model (SA), k-kl-ω transition turbulence model and transition SST model. The present work proposed the curved diffuser with AR=2 operated at Re=6x10^4as the optimum set of parameters, with minimal flow separation occurring in the system.

1. Introduction

Flows in diffuser occurs in various engineering devices such as wind tunnel system, HVAC, aircraft engineering etc. Moreover, various types of diffusers which are commonly classified by their geometries and applications such as annular, pyramidal, slab, S-shaped and Y-shaped diffusers [1-16]. The performance of curved diffuser often evaluated based on its pressure recovery and flow uniformity index. The flow separation is the natural effect that normally occurs at flow of fluid in curved diffuser because of the angle of attack itself. This phenomenon had led to a pressure loss for a flow inside a diffuser with the turning angle. A strong adverse pressure gradient is the effect of a curvature induced that subjected towards the inner wall of diffusers. The expansion and sharp inflection along the direction of flow in a curved diffuser make the flow complex and hardly to brief.

* *Corresponding author.*
E-mail address: mayati@uthm.edu.my (Normayati Nordin)

https://doi.org/10.37934/cfdl.12.7.100109

Furthermore, the boundary layer of inner wall is likely to separate, and the core flow tends to deflect toward the outer wall region. This flow separation is undesirable as it could adversely affect the overall performance of diffusers.

Throughout the past few decades, there are several types of diffuser had been tested on its performance and applicability for a very specific application. A turning or curved diffuser are commonly used in HVAC systems [10]. S-shaped diffuser is applied as an intake duct in dual engine fighter aircrafts [2] while Y-shaped diffuser is used for almost all modern combat aircrafts. In this research, a two-dimensional 180^o turning angle curved diffuser has been tested based on its performance of pressure recovery and flow uniformity by using ANSYS Computational Fluid Dynamics (CFD).

Referring to Fox & Kline [11], the study of two-dimensional curved diffuser with turning angle of 0° to 90° have been made. The geometrical and operating parameter used in this study were area ratio (AR=1.2-4.0), the inner wall length to the inlet throat width ratio (L_{in}/W_1=1.5-30). This experiment focused on the flow regimes variation and the performance of diffuser. It has been founded that the increment of turning angle on a curved diffuser has affected the stability of flow regime due to severe separation. Sudo *et al.,* [12] have run experimental works for 90° and 180° turning angle on a curved diffuser with an objective also to describe the performance based on its pressure recovery. The curved diffuser was built with the same outlet and inlet diameter of 104 mm, radius of curvature of 208 mm and operated at Reynolds number of 6x10^4. The result of pressure recovery displayed at turning angle of 90° were higher than 180° curved diffuser as the effect of flow separation occur is lower. Nordin *et al.,* [13] investigated the performance of 90° turning angle of 2-D and 3-D curved diffuser with varying the geometrical and operating parameters. From the study, it showed that 3-D turning diffuser provided higher pressure recovery at low inflow Reynolds number of 5.786 x 10^4 to 6.382 x 10^4 and better flow uniformity at high inflow Reynolds number of 1.027 x 10^5 to 1.775 x 10^5 than the 2-D turning diffuser.

Pressure recovery C_p and flow uniformity index σ_u performance of a curved diffuser is the prevalent research's objective studied from the past researcher. The flow uniformity index (σ_u) is used to measure the extent of dispersion of local outlet velocity from the mean outlet velocity. It was strongly dependent on the dispersion of core flow and the presence of secondary flow throughout the outlet cross-section of a curved diffuser [3]. The geometrical and operating parameters of a curved diffuser are the major factor that affected the performance. From the past research made, it has been proven that the higher Reynolds number tested on a curved diffuser will affect the result on pressure recovery to slightly decrease [12].

Based on the review of paper make, there still slight less researchers considered a 180° curved diffuser work with a different operating and geometrical parameters. In the present work, the performances of $180°$ curved diffuser by varying geometrical and operating parameters are investigated using ANSYS CFD. The inlet diameter of the diffuser is fixed at 50 mm meanwhile the outlet diameter value is depending on the various area ratio (AR) considered 2, 3, and 4. The radius of curvature is set at 180 mm and the inlet Reynolds number are varied at 6 x 10^4, 12 x 10^4, 18 x 10^4, 24 x 10^4, and 30 x 10^4. The working fluid is air with density 1.225 kg/m^3 and viscosity 1.7895 x 10^{-5} kg/ms.

2. Methodology
2.1 Design and Model of Diffuser

The present work focuses on defining a proper configuration of 180° curved diffuser to provide promising flow uniformity and pressure recovery. Hence, CFD is used as a tool to simulate the flow performance inside the c-shaped 180° diffuser.

The two-dimensional simulation performance of 180° diffuser is performed using ANSYS CFD software. A numerical approach can be a solution for governing equation of fluid motion in differential form resulted from the various fluid flow manners. For ANSYS CFD, it is built by two major elements which are Finite Elements Method (FEM) and Time-Averaged Navier Stokes to solve the governing equation of fluid motion.

Figure 1 illustrates the geometry of $180°$ curved diffuser. The main geometrical parameters of the diffuser should be turning angle (θ), inlet width (w_1), outlet width (w_2), radius of curvature (RC), and area ratio (AR) which are listed in Table 1.

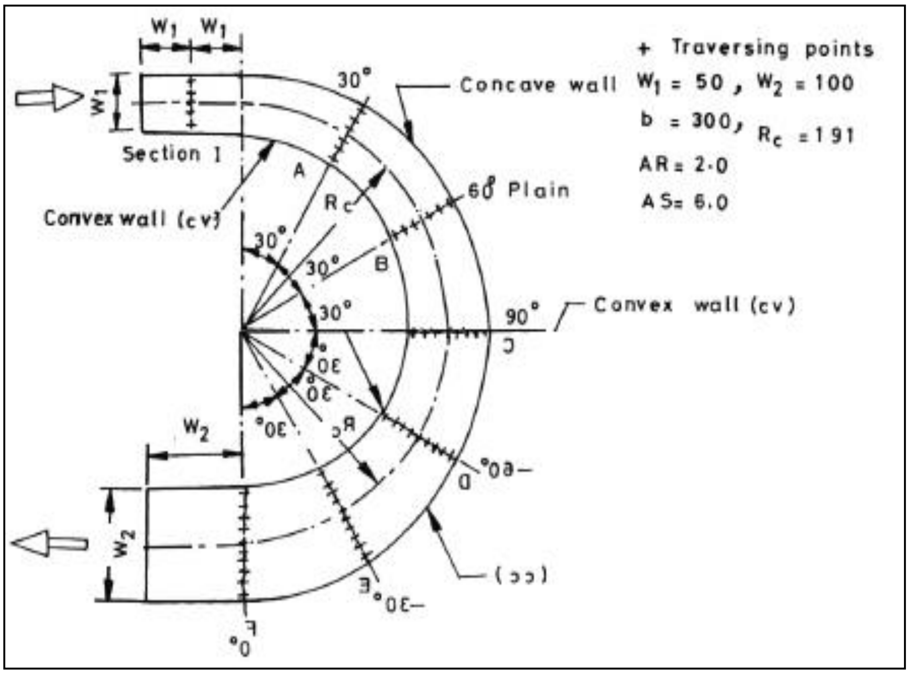

Fig. 1. Geometry of 180° curved diffuser

Table 1
Geometrical setup value for curved diffuser

Geometrical setup	Value
Inlet width, w_1	50 mm
Area of ratio, AR	2,3, and 4
Turning angle, θ	180°

2.2 Grid Independency Test

ANSYS CFD code FLUENT was used to perform the numerical works on the performances of the turning diffusers. Enhanced wall treatment is applied with wall-adjacent cell centroid was placed within the viscous sublayer, $y+ \approx 1.0$. With the Re = 6 x 10^4, 12 x 10^4, 18 x 10^4, 24 x 10^4, and 30 x 10^4, the corresponding first grid point off the wall was calculated as 2.278×10^{-5}m (Re = 6 x 10^4). The grid independency study was made by decreasing the value of sizing for every meshing body until the Cp results show insignificant changes. Table 2 displayed the grid independency study for 2-D 180° curved diffuser.

Table 2
Grid independency study for 2-D 180° curved diffuser

Angle of diffuser	Sizing (mm)	Nodes	Elements	Pressure recovery, Cp	Deviation, %
180°	7	28748	89233	0.1901	23.1
	6	41153	135502	0.1913	17.5
	5	64071	226338	0.2109	12.1
	4	115893	438403	0.2397	7.9
	3	245126	1017534	0.2488	3.1

2.3 Boundary Layer Condition

Boundary operating conditions are applied with inlet velocity set at five different Reynolds number of $6 \times 10^4, 12 \times 10^4, 18 \times 10^4, 24 \times 10^4$, and 30×10^4. The pressure at the diffuser exit is atmospheric pressure and wall is treated as a smooth wall with no slip condition. A solver detail which relates to governing equation was independently solved using a double precision pressure-based solver with the assumption of steady-state, 2-D and incompressible flow. The governing equation or Reynolds-Averaged Navier-Stokes equations (RANS) for two-dimensional laminar steady flow are as equations state below:

i. x-momentum equation

$$\frac{\partial}{\partial x}(\rho uu) + \frac{\partial}{\partial y}(\rho uv) = \frac{\partial}{\partial x}\left(\mu \frac{\partial u}{\partial x}\right) + \frac{\partial}{\partial y}\left(\mu \frac{\partial u}{\partial y}\right) - \frac{\partial p}{\partial x} + S_u \tag{1}$$

ii. y-momentum equation

$$\frac{\partial}{\partial x}(\rho w) + \frac{\partial}{\partial y}(\rho vv) = \frac{\partial}{\partial x}\left(\mu \frac{\partial v}{\partial x}\right) + \frac{\partial}{\partial y}\left(\mu \frac{\partial v}{\partial y}\right) - \frac{\partial p}{\partial y} + S_v \tag{2}$$

iii. Continuity equation:

$$\frac{\partial}{\partial x}(\rho u) + \frac{\partial}{\partial y}(\rho v) = 0 \tag{3}$$

CFD validation is a process of comparing the data to the requirement in a set of documented acceptance criteria. A numerical data set in scope of study will be compared by the experimental data from the previous study. The author used to set a diffuser with a 180° bend by using a working fluid of air with a Reynolds number of 6×10^4 [12].

The result for pressure recovery by Sudo *et al.,* [12] is chosen for validation purpose. Figure 2 shows the graphical result of pressure recovery obtained for 180° curved diffuser. Eq. (4) represents the formula used to calculate the value of working pressure recovery from the analysis of data that came out with the value of pressure at inlet and outlet of diffuser. Meanwhile, Eq. (5) is applied to determine flow uniformity index.

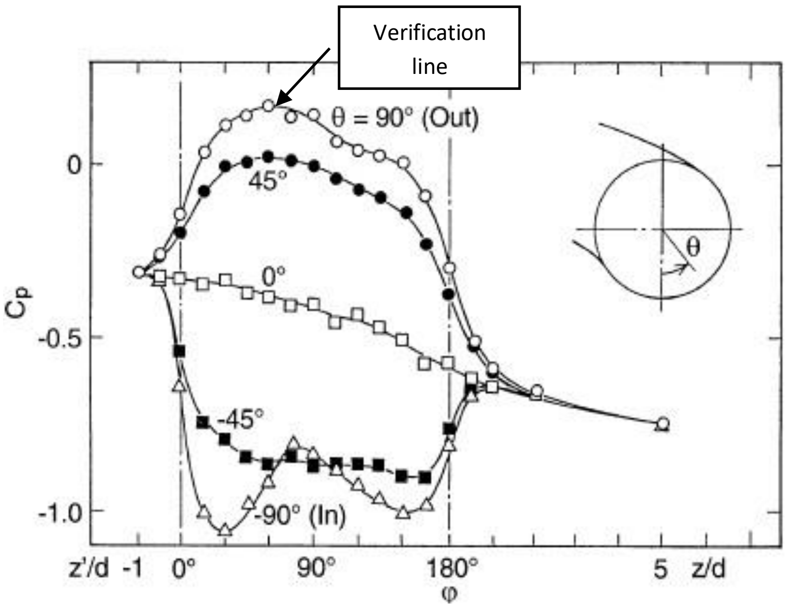

Fig. 2. Graphical result for pressure recovery, $\boldsymbol{C_p}$ [12]

a) Pressure recovery coefficient, C_p

$$C_p = (p_o - p_i)/(\frac{\rho Vin^2}{2}) \quad (4)$$

where,

C_p = Coefficient of pressure
p_o = outlet pressure
p_i = inlet pressure
ρ = density of fluid (air)
Vin = Inlet velocity

b) Flow uniformity, σ_{out}

$$\sigma_{out} = \sqrt{\frac{1}{N-1}\sum_{i=1}^{N}(V_{out}i - V_{out})^2} \quad (5)$$

where,

σ_{out}= flow uniformity index
N= number of measurement points
$V_{out}i$ = local outlet air velocity magnitude (m/s)
V_{out} = mean outlet air velocity magnitude (m/s)

3. Results

3.1 Verification and Validation Results

In the present work, the Normal Root Mean Square Error (NRSME) method is used to simply differentiate and compare the best model of solver with the existing research results made by Sudo *et al.,* [12]. Besides, this method used as a preliminary work in the present research and a process of research verification. Table 3 displayed the results of NRSME method made for five different model of solver. The graph of comparison is made as shown is Figure 3.

From CFD validation results in Table 3, Spalart-Allmaras model provided the most accurate prediction among all the solvers with average deviation percentage of 3.4%. The transition onset characteristics of the boundary layer shows the best result. Thus, it is proven the reliability of the numerical method applied in the current work. Then, Spalart-Allmaras(SA**)** was selected to examine the performance of 2-D 180° curved diffuser in intensive simulations.

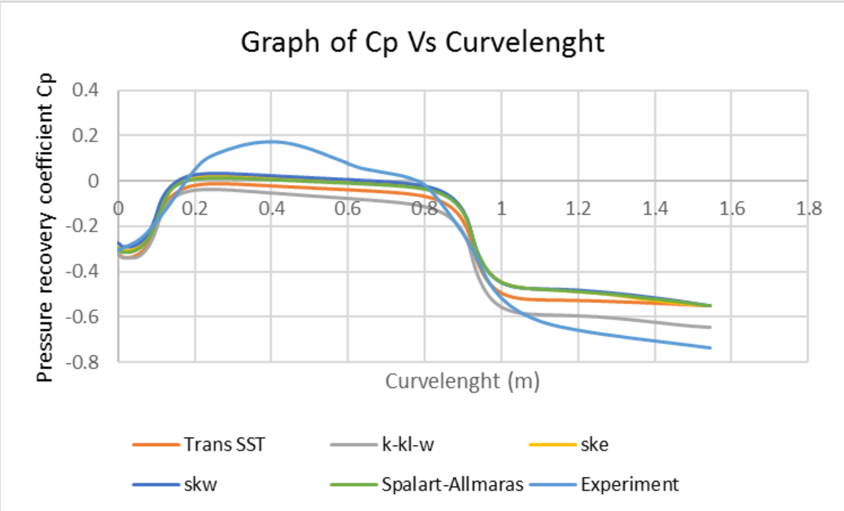

Fig. 3. Experiment result for different turbulent models

Table 3
NRSME value for validation of model solvers

Model of solvers	NRSME (%)
Spalart-Allmaras (SA)	3.4
k-kl-ω transition	5.8
Standard k-e (SKE)	3.6
Standard $k-\omega$	3.5
transition SST	4.5

3.2 Pressure Recovery (Cp) against Area Ratio (AR) and Reynolds Number (Re)

Intensive simulations were made for 180 ° by using Sparlart Allmaras (SA) solver. The value of Cp is calculated by using Eq. (4) from the reading of inlet and outlet pressure (As shown in Table 4). The results turn out as shown in Figure 4 for the graph of pressure recovery coefficient for different Reynolds number with a different geometry size of Area of ratio. Area ratio of 2 have the highest value of pressure recovery meanwhile for area of 4 is the least once. Figure 5 shows the graph also for pressure recovery but it against the area of ratio with different values of Reynolds number. Re=6x10^4 produced the highest Cp at every different AR used.

Table 4
Table of Cp for different Re and AR

AR=2	Pinlet(Pa)	Poutlet(Pa)	Re	Cp
	-83.969	-0.142	$6x10^4$	0.445
	-331.514	-0.517	$12x10^4$	0.439
	-747.391	-1.059	$18x10^4$	0.441
	-1321.69	-2.0761	$24x10^4$	0.438
	-2058.24	-3.0521	$30x10^4$	0.437
AR=3	Pinlet(Pa)	Poutlet(Pa)	Re	Cp
	-70.7867	-0.256	$6x10^4$	0.375
	-283.192	-1.118	$12x10^4$	0.375
	-631.55	-2.657	$18x10^4$	0.371
	-1114.69	-5.109	$24x10^4$	0.368
	-1732.53	-8.055	$30x10^4$	0.367
AR=4	Pinlet(Pa)	Poutlet(Pa)	Re	Cp
	-57.369	-0.537	$6x10^4$	0.302
	-246.749	-2.206	$12x10^4$	0.325
	-559.046	-4.848	$18x10^4$	0.327
	-991.539	-8.793	$24x10^4$	0.326
	-1548.35	-13.437	$30x10^4$	0.326

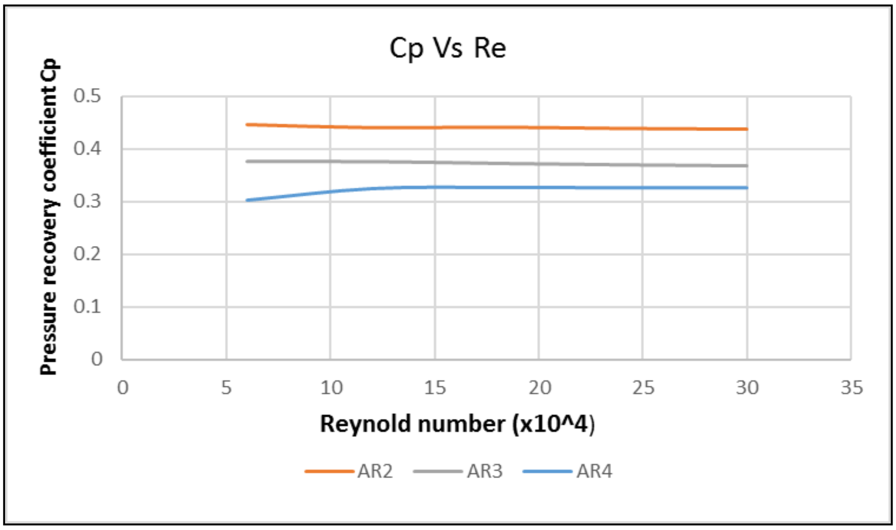

Fig. 4. Graph of pressure recovery (C_p) vs Reynolds number (Re)

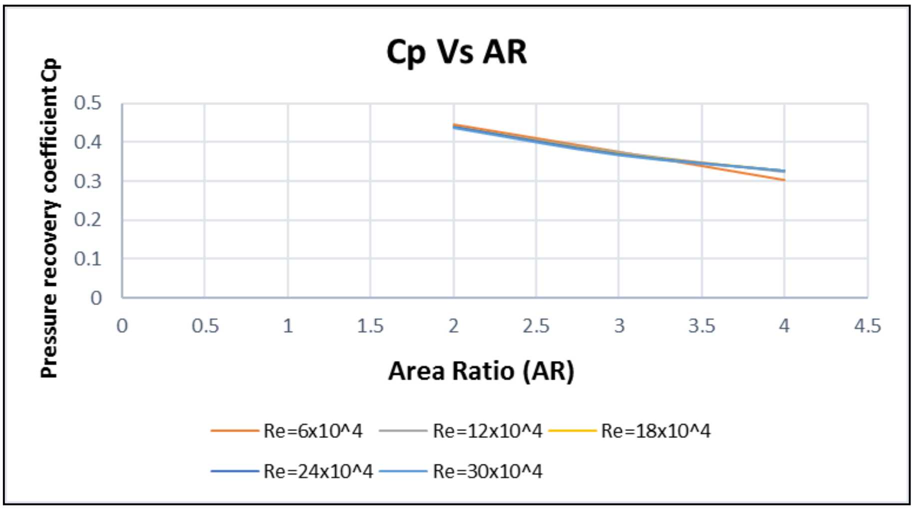

Fig. 5. Graph of C_p vs area ratio (AR)

3.3 Flow Uniformity σ_{out} against AR and Re

Table 5 displays the results of flow uniformity from intensive simulation made for the different operating parameters. From Figure 6, area ratio of 3 at Re=30×10^4 has the highest value of flow uniformity which is 32.04 comparing to other area ratios. Meanwhile, from Figure 7, it is proved that at Re=30×10^4, the flow uniformity is the highest as the velocity is high.

Table 5
Value of σ_{out} for different AR and Re

Reynolds number	*Flow uniformity*, σ_{out}		
	AR=2	AR=3	AR=4
6x10^4	5.69	6.39	6.01
12x10^4	11.74	12.74	11.81
18x10^4	17.82	19.17	17.88
24x10^4	23.99	25.55	23.84
30x10^4	30.32	32.04	30.03

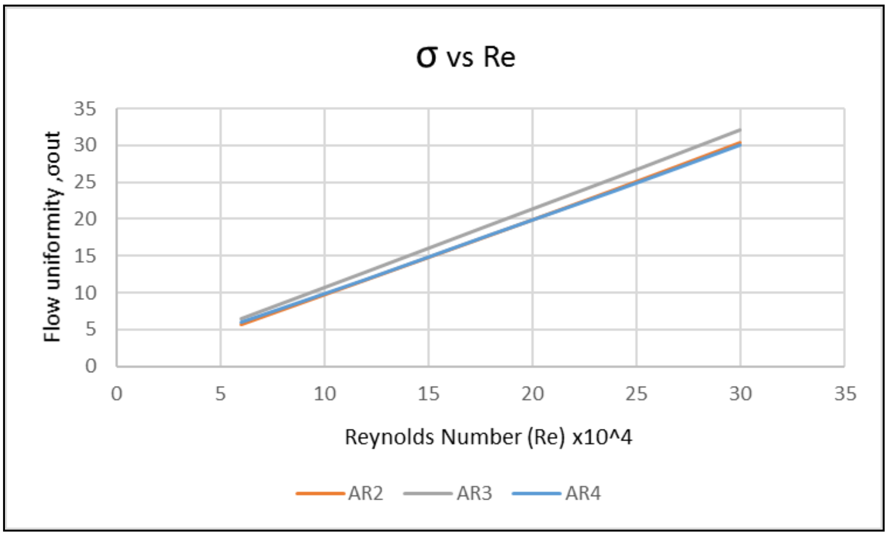

Fig. 6. Graph of flow uniformity (σ_{out}) vs Reynolds number (Re)

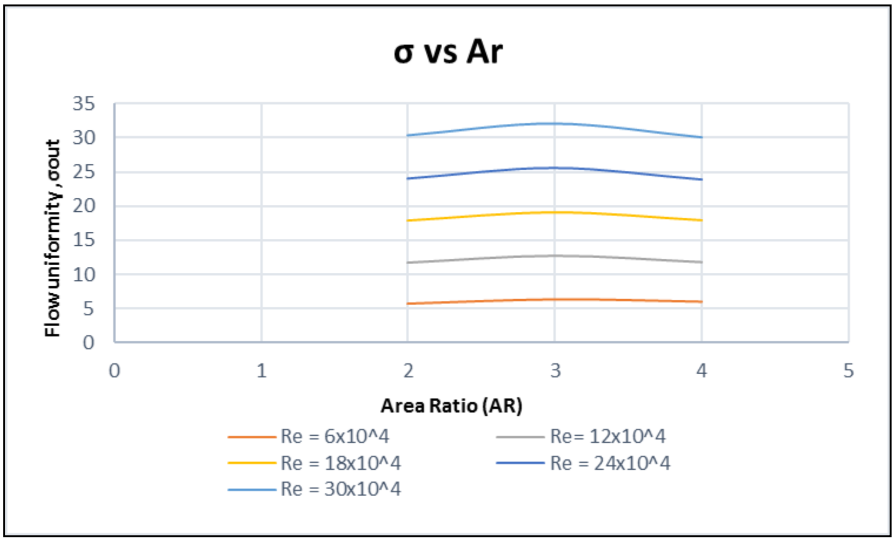

Fig. 7. Graph of flow uniformity σ_{out} vs Reynolds number (Re)

3.4 Pressure and Velocity Contour by CFD

Figure 8 shows the static pressure contour for Re=6x10^4 and AR=2. From the figure, it shows that at the pressure is higher once it reached a bigger area of diameter. This proved that pressure would increase when it in the diverging phase. Figure 9 shows the velocity contour for almost all models for different (AR=2). From the figure, it displayed that the velocity at inlet is bigger or higher than near the outlet. This occur based on the converging phase of the diffuser itself. Converging experimentally proved that the velocity will decrease once the air is going through the increasement of area of diffuser.

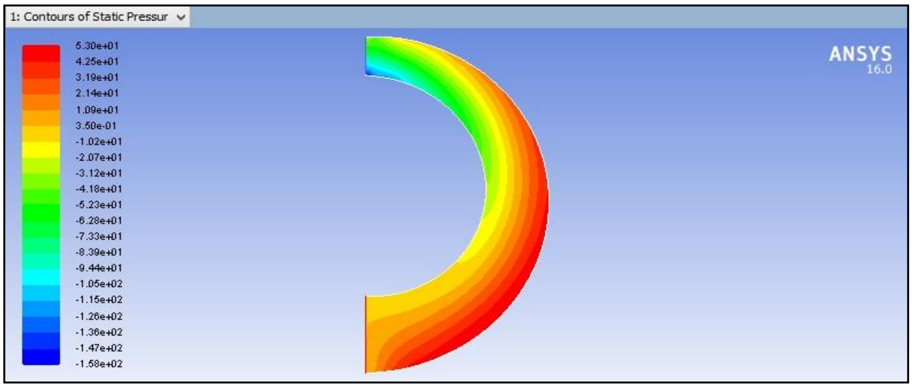

Fig. 8. Pressure contour of diffuser at (AR=2) and (Re=6x10^4)

Fig. 9. Velocity contour of diffuser at (AR=2) and (Re=6x10^4)

4. Conclusions

In conclusion, the research on investigating the performance of 2-D 180° curved diffuser via computational fluid dynamics (CFD) has been successfully carried out. The best pressure recovery performance of 0.445 was obtained at curved diffuser with area ratio of 2 and operated at Reynolds number of 6 x10^4. Meanwhile, the highest flow uniformity index of 32.04 was obtained at curved diffuser with area ratio of 3 and operated at Reynolds number of 30 x10^4. From the results, the pressure recovery performance was higher when the area ratio of curved diffuser is low and operated at the lowest Reynolds number. For flow uniformity index, the uncertainty in the value of results is obtained for different area ratio used but it is proved that the increment for inflow Reynolds number was increased the value of flow uniformity.

Acknowledgement
This experiment was supported in part by Universiti Tun Hussein Onn Malaysia under TIER 1 Vote H172. CFD work was carried out in CFD laboratory, Universiti Tun Hussein Onn Malaysia (UTHM).

References

[1] Kahnert, Michael, T. Nousiainen, and P. Räisänen. "Mie simulations as an error source in mineral aerosol radiative forcing calculations." *Quarterly Journal of the Royal Meteorological Society: A journal of the atmospheric sciences, applied meteorology and physical oceanography* 133, no. 623 (2007): 299-307. https://doi.org/10.1002/qj.40

[2] Saha, K., S. N. Singh, V. Seshadri, and S. Mukhopadhyay. "Computational analysis on flow through transition S-diffusers: Effect of inlet shape." *Journal of aircraft* 44, no. 1 (2007): 187-193. https://doi.org/10.2514/1.22828

[3] Gopaliya, Manoj Kumar, Mahesh Kumar, Shailendra Kumar, and Shiv Manjaree Gopaliya. "Analysis of performance characteristics of S-shaped diffuser with offset." *Aerospace Science and Technology* 11, no. 2-3 (2007): 130-135. https://doi.org/10.1016/j.ast.2006.11.003

[4] Cerantola, D. J., and A. M. Birk. "Experimental Validation of Numerically Optimized Short Annular Diffusers." *Journal of Engineering for Gas Turbines and Power* 137, no. 5 (2015). https://doi.org/10.1115/GT2013-94648

[5] Merkli, Peter E. "Pressure recovery in rectangular constant area supersonic diffusers." *AIAA journal* 14, no. 2 (1976): 168-172. https://doi.org/10.1016/j.ast.2012.02.005

[6] Blevin R . 1984 Applied Fluid Dynamics (New York).

[7] Jakirlić, S., 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* 31, no. 5 (2010): 820-832. https://doi.org/10.1016/j.ijheatfluidflow.2010.05.004

[8] W MALALASEKERA & H V 1995 An introduction to fluid dynamics (Harlow, England: Pearson, Prentice Hall)

[9] Nordin, Normayati, Vijay R. Raghavan, Safiah Othman, and Zainal Ambri Abdul Karim. "Numerical investigation of turning diffuser performance by varying geometric and operating parameters." In *Applied Mechanics and Materials*, vol. 229, pp. 2086-2093. Trans Tech Publications Ltd, 2012. https://doi.org/10.4028/www.scientific.net/AMM.229-231.2086

[10] Calautit, John Kaiser, Hassam Nasarullah Chaudhry, Ben Richard Hughes, and Lik Fang Sim. "A validated design methodology for a closed-loop subsonic wind tunnel." *Journal of Wind Engineering and Industrial Aerodynamics* 125 (2014): 180-194. https://doi.org/10.1016/j.jweia.2013.12.010

[11] Fox, Robert W., and S. J. Kline. "Flow regimes in curved subsonic diffusers." (1962): 303-312. https://doi.org/10.1115/1.3657307

[12] Sudo, K., M. Sumida, and H. Hibara. "Experimental investigation on turbulent flow in a circular-sectioned 90-degree bend." *Experiments in Fluids* 25, no. 1 (1998): 42-49. https://doi.org/10.1007/s003480050206

[13] Nordin, Normayati. "Performance investigation of turning diffusers at various geometrical and operating parameters." PhD diss., Universiti Teknologi PETRONAS (UTP), 2016.

[14] Huang, Lim Gim, Normayati Nordin, Lim Chia Chun, Nur Shafiqah Abdul Rahim, Shamsuri Mohamed Rasidi, and Muhammad Zahid Firdaus Shariff. "Effect of Turbulence Intensity on Turning Diffuser Performance at Various Angle of Turns." *CFD Letters* 12, no. 1 (2020): 48-61.

[15] Liang, Chua Bing, Akmal Nizam Mohammed, Azwan Sapit, Mohd Azahari Razali, Mohd Faisal, Amir Khalid Hushim, and Nurul Farhana Mohd Yusof. "Numerical Simulation of Aerofoil with Flow Injection at the Upper Surface." *CFD Letters* 12, no. 1 (2020): 98-110.

[16] Zuan, A. M. S., A. Ruwaidab, S. Syahrullailc, and M. N. Musad. "The Effect of Adding Diffuser by Experimental." *Journal of Advanced Research in Applied Mechanics* 14, no. 1 (2015): 18-24.