DEVELOPMENT OF SUBSONIC CURVED DIFFUSER PERFORMANCE CORRELATIONS INTEGRATED ANGLE OF TURNS USING ASYMPTOTIC COMPUTATIONAL FLUID DYNAMICS

SHAMSURI BIN MOHAMED RASIDI

A thesis submitted in
fulfilment of the requirement for the award of the
Degree of Master of Mechanical Engineering

Faculty of Mechanical and Manufacturing Engineering
Universiti Tun Hussein Onn Malaysia

June 2022

In the name of God, The Most Gracious, The Most Merciful

For my beloved parents, Mr Mohamed Rasidi Bin Abdullah and Mrs Faridah Bt. Abu Bakar.
My dear siblings, Nur Fadilah Binti Mohamed Rasidi, Mohd Azim Bin Mohamed Rasidi, Nur Syamimi Binti Mohamed Rasidi.

My Fiance, Nur Amirah Binti Madri Saleh.

My friends, Muhammad Zahid Firdaus Shariff, Muhammad Syamir Abu Bakar, Akmal Azfar, Solehan Khairulfuad, Razizy Fauzi, Nur Farahain, Nurzaha Hassan, Nur Syazwani, Muhd Ridzuan, Zhairul Iqumal, Nur Farahalya, & Nur Amani.

My respected supervisor Dr Normayati Binti Nordin, and laboratory instructor En. Zaimal Abidin Bin Alias.

Thank you for all your support along the way.

ACKNOWLEDGEMENTS

The author wishes to express his heartfelt gratitude to his supervisor, Dr Normayati Binti Nordin, for her guidance, assistance, compassion, and encouragement to overcome the various barriers and problems encountered throughout this research.

Additionally, the author wishes to convey his appreciation to his parents, and for their support throughout his two and a half years of study at Universiti Tun Hussein Onn Malaysia. The author wishes to express his gratitude and appreciation to everyone who assisted him in any way during the process.

Insha'Allah, Allah S.W.T. will reward those directly or indirectly engaged in this research, and may Allah shower us with blessings.

ABSTRACT

Numerous studies on the performance of the curved diffuser have been made on either 2-D or 3-D expansion type with various working geometrical and operating parameters. Most researchers are focusing in the existence of flow separation phenomenon and secondary flow vortices that often disturb the recovery of pressure and uniformity of flow. On top of that, the existing guideline have just integrated the geometrical and operating effects in a low range. Therefore, the current work focussing on studying the effects of a wider turning angles in a range of 30° to 180° integrated with various operating condition by experiment and numerical method. The experimental rig was built at Aerodynamics Laboratory, UTHM. The blower speed was set in range of 9RPM-25RPM and tested for 30°, 90° and 180° curved diffusers. A profound set of Re_{in} were obtained for three test made where $6.149\mathrm{x}10^4$-$1.828\mathrm{x}10^5$ for 30°, $6.080\mathrm{x}10^4$-$1.820\mathrm{x}10^5$ for 90°, and $5.784\mathrm{x}10^4$-$1.607\mathrm{x}10^5$ for 180°. The result of C_p obtained indicates that 30° curved diffuser have the best C_p performance from all curved diffusers tested. In validating the results, three solver models are tested which are standard k-ε adopting enhanced wall treatment of $y^+ \approx 1.1$-1.8 was the best operated model to validate the experiment results against numerical. Two sets of correlations that integrate the performance of $\mathrm{C}_p$and σ_{out} by using Asymptotic Computational Fluid Dynamics (ACFD) method is made. This correlation indicated the novelty of the new correlation made in curved diffuser research.

ABSTRAK

Pelbagai kajian tentang prestasi peresap melengkung telah dibuat pada sama ada jenis pengembangan 2-D atau 3-D dengan pelbagai parameter geometri dan operasi yang berfungsi. Kebanyakan penyelidik memfokuskan kepada kewujudan fenomena pemisahan aliran dan pusaran aliran sekunder yang sering mengganggu pemulihan tekanan dan keseragaman aliran. Selain itu, garis panduan sedia ada baru sahaja menyepadukan kesan geometri dan operasi dalam julat yang rendah. Oleh itu, kerja semasa memfokuskan pada mengkaji kesan sudut pusingan yang lebih luas dalam julat 30° hingga 180° disepadukan dengan pelbagai keadaan operasi melalui kaedah eksperimen dan berangka. Rig eksperimen telah dibina di Makmal Aerodinamik, UTHM. Kelajuan blower ditetapkan dalam julat 9RPM-25RPM dan diuji untuk peresap melengkung 30°, 90° and 180°. Re_{in} diperolehi untuk tiga ujian yang dibuat di mana $6.149\mathrm{x}10^4$-$1.828\mathrm{x}10^5$ untuk 30°, $6.080\mathrm{x}10^4$-$1.820\mathrm{x}10^5$ untuk 90° dan $5.784\mathrm{x}10^4$-$1.607\mathrm{x}10^5$ untuk 180°. Hasil C_p terhadap ketiga-tiga kajian menunjukkan peresap lengkup 30° adalah tertinggi. Untuk kaedah numerik, ANSYS 19.2 (Fluent) dengan pelbagai model penyelesai ditambah dengan rawatan dekat dinding telah digunakan untuk pengesahan dan simulasi intensif dalam prosedur berangka. Antara ketiga-tiga model tersebut, k-ε standard yang mengguna pakai rawatan dinding dipertingkatkan $y^+ \approx 1.1$-1.8 adalah model kendalian terbaik untuk mengesahkan keputusan eksperimen terhadap berangka. Dua set korelasi yang menyepadukan indeks prestasi dengan menggunakan kaedah Asymptotic Computational Fluid Dynamics (ACFD). Korelasi ini menunjukkan kebaharuan korelasi baharu yang dibuat dalam penyelidikan peresap melengkung.

CONTENTS

APPENDICES

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS & ABBREVIATIONS

2-D	Two dimensional
3-D	Three dimensional
ACFD	Asymptotic computational fluid dynamics
AR	Area ratio
CFD	Computational fluid dynamics
Gr	Grashof number
HVAC	Heating, ventilation and air-conditioning
Nu	Nusselt number
Pr	Prant number
PIV	Particle Image Velocimetry
PRESTO	Pressure discretization scheme
QUICK	Quadratic upwind interpolation
Ra	Rayleigh number
Re	Reynolds number
Re_{in}	Inflow Reynolds number
Rke	Realizable k-ε turbulence model
Rngke	Renormalization k-ε turbulence model
RANS	Reynolds average navier stokes
RPM	Rotation per minute
RSM	Reynolds stress model
SIMPLE	Pressure-velocity coupling scheme
SIMPLEC	Pressure velocity coupling scheme
Ske	Standard k-ε turbulence model

A	Cross-sectional area $[m^2]$
A_{in}	Inlet cross-sectional area $[m^2]$
A_{out}	Outlet cross-sectional area$[m^2]$
$\boldsymbol{C}$	Contraction ratio
C_p	Outlet pressure recovery coefficient
$C_{p\ ideal}$	Ideal outlet pressure recovery coefficient
C_f	Wall skin friction coefficient
D_h	Hydraulic diameter [m]
E	Empirical constant (= 9.793)
F	Friction factor
H	Height of diffuser [m]
I	Turbulent intensity [%]
K	Overall system pressure loss coefficient
K_f	Fittings pressure loss coefficient
$L_{diffuser}$	Length of diffuser [m]
L_{ax}	Axial length [m]
L_c	Contraction cone length [m]
$L_{h,turb}$	Turbulent hydrodynamic length [m]
L_{in}	Inner wall length [m]
L_m	Centre curve length [m]
L_{out}	Outer wall length [m]
N	Number of measurement points
$\boldsymbol{P}$	Perimeter [m]
P_{dyn}	Dynamic pressure [Pa]
P_{dync}	Central dynamic pressure [Pa]
P_{in}	Average static pressure at the inlet [Pa]
P_{out}	Average static pressure at the outlet [Pa]
Q	Volumetric flow rate $[m^3 s^{-1}]$
R	Intermediate radius [m]
r_{in}	Inner wall radius [m]
r_m	Centre curve radius [m]
r_p	Measurement point from the wall

R	Pipe radius [m]
S	Separation point
S_{out}	Secondary flow index
T	Operating temperature [ºC]
U	Mean velocity [ms^{-1}]]
U^*	Dimensionaless velocity
u'	Fluctuating *x*-velocity component [ms^{-1}]
u_*	Friction velocity [ms^{-1}]
U_O	Centreline velocity [ms^{-1}]
v'	Fluctuating y-velocity component [ms^{-1}]
V_i	Local outlet air velocity [ms^{-1}]
$V_{i\,max}$	Maximum local outlet air velocity [ms^{-1}]
V_{in}	Mean inlet air velocity [ms^{-1}]
$V_{in\,max}$	Maximum inlet air velocity [ms^{-1}]
V_{out}	Mean outlet air velocity [ms^{-1}]
V_x	x- velocity component [ms^{-1}]
V_y	y- velocity component [ms^{-1}]
$V_{y\,ps}$	y- velocity component obtained by pitot static probe [ms^{-1}]
V_z	z- velocity component [ms^{-1}]
W	Width of duct [m]
w'	Fluctuating *z*-velocity component [ms^{-1}]
W_1	Inlet throat width at y-axis direction [m]
W_2	Outlet throat width at z-axis direction[m]
w_{Pn}	Local inlet air velocity at point *n*, [ms^{-1}]
$w_{Pn\,ps}$	Local inlet air velocity at point *n* obtained by Pitot static probe [ms^{-1}]
$w_{Pn\,theo}$	Local inlet air velocity at point *n* obtained by theory [ms^{-1}]
X_1	Inlet throat width at x-axis direction [m]
X_2	Outlet throat width at x-axis direction [m]
y^+	The first grid point off the wall
y_*	Dimensionaless distance from the wall

σ_{out}	Flow uniformity index [ms^{-1}]
σ_y	Core flow index [ms^{-1}]
σ_k	Turbulent Prandtl number for k
σ_ε	Turbulent Prandtl number for ε
$\phi_{1,2,3\ldots}$	Dimensionless groups of independent variables
$\Delta\phi$	Turning angle [o]
θ	Single divergence angle [o]
Δt	Time between pulses [s]
α	Total divergence angle, 2θ [o]
ρ	Air density [kgm^{-3}]
ε	Total energy dissipation rate [m^2s^{-3}]
δ	Boundary layer thickness [m]
μ	Dynamic viscosity [kgm-1 s-1]
μ_t	Turbulent or eddy viscosity [kg$m^{-1}s^{-1}$]]
υ	Kinematic viscosity [m^2s^{-2}]
υ_t	Kinematic turbulent or eddy viscosity m^2s^{-1}]
ξ	Effectiveness of diffuser
κ	Von Karman constant (= 0.4187)

CHAPTER 1

INTRODUCTION

1.1 Research Background

Diffusers has been used widely in engineering systems to decelerate the fluid’s flow via the changing phase of the diffuser’s shapes and sizes based on its application. Referring to Nordin et al. [1], a straight diffuser is a diverging duct with zero angle of turn, whereas a curved diffuser refers to a diverging duct with a certain angle of turn. A curved diffuser can be assembled by diverging it into either two or three dimensions.

In recent years, various types of curved diffusers have been used simply based on their applicability throughout recent years. A closed-circuit wind tunnel is customarily constructed with the principle used of 90° or 180° curved diffusers [2]. In general, actual work or applications of a diffuser is always centred on a settlement between pressure recovery and flow uniformity [3]. Other than a simple turning diffuser, S-shaped and Y-shaped (fish-tail) diffusers were also found applicable for almost all modern combat aircraft that used fuselage-mounted intake [4,5]. The nature of the geometry of the curved diffuser has led to the determination of pressure recovery, losses, and non-uniformity of flow that usually present at the exit of flow as the effects of flow separation and dispersion of core flow [6]. The existence of secondary flow in turning diffuser has drawn massive interest in studying its performance, especially on the pressure recovery coefficient and flow uniformity index. Moreover, various geometrical aspects and operating parameters have been applied in consideration of the best turnout model of performance for a turning diffuser.

In this present work, the performance of curved diffuser is in concern with varying geometrical and operating parameters, namely turning angle ($\Delta\emptyset$), inner wall length to inlet throat width ratio (L_{in}/W_1) area ratio (AR), outlet-inlet configurations (W_2 /W_1, X_2 /X_1) and inflow Reynolds number (Rein) were numerically and experimentally investigated. Integrated performance correlations of a curved diffuser were ultimately developed using Asymptotic Computational Fluid Dynamics.

1.2 Problem Statement

Curved diffuser is applied widely in industrial flow to conserve energy by converting the kinetic energy to pressure energy. Nevertheless, flow performance is often disturbed, especially on a complex flow due to its nature of geometry such as the turning angle, aspect ratio, or even inlet flow velocity by the existence of flow separation and dispersion of core. The practical application always seeks a compromise between maximum permissible pressure recovery and flow uniformity, which can be achieved by optimally setting the geometrical and operating parameters.

The performance of various applications of fluid machinery is often disturbed by the existence of flow separation [7]. Flow separation problem is common when using a diffuser; the cross-sectional area increases and is vulnerable to separation. It is the primary cause of pressure drop. The lower Reynolds number applied also causes the problem of separation phenomenon in a passage flow diffuser [8]. This circulation or rotating flow phenomenon appears for a fluid with non-uniform velocity distribution that passes around a bend. Besides the flow speed, the stagnant fluids in the wall boundary layer as the effect of the centrifugal pressure gradient is also the attribution of secondary flow or flow separation [9].

There are various literature available for a diffuser that has been made, particularly for 2-D curved diffusers. Fox and Kline [7] have established a guideline in choosing the optimum geometries of a 2-D curved diffuser free from the stall. Nordin et al. [3] recently have developed mathematical correlations to quantitatively evaluate pressure recovery and flow quality of 90° curved diffusers. Nevertheless, these available guidelines could still not comprehensively represent the performance of curved diffuser, particularly when the 3-D expansion of various angle of turns are considered.

1.3 Objectives of research

The objectives of this research are specified as follows:

1) To numerically and experimentally investigate the effects of varying geometrical and operating parameters on the performance of curved diffusers.
2) To develop performance correlations of curved diffusers by integrating angle of turn via the Asymptotic Computational Fluid Dynamics (ACFD) technique.
3) To propose optimum configuration of geometrical and operating parameters for a curved diffuser.

1.4 Scope of Research

The research scopes are as follows:

1) Curved diffusers with identical inlet conditions were considered in range of area ratio 1.0 to 4.0 as in guideline.
2) Performance of curved diffusers was evaluated primarily in terms of pressure recovery coefficient, Cp and flow uniformity index, σ_{out}.
3) The angles of turn varied between 30° to 180° representing common turning angles applied in HVAC and wind tunnel systems.
4) ANSYS 19.2 Fluent is used for CFD simulation, including research and data management (Workbench), modelling (DesignModeler), grid generation (ICEM CFD) and flow analysis (Fluent).
5) Asymptotic computational fluid dynamics (ACFD) developed the performance correlations (C_p and σ_{out}) as a function of geometrical and operating parameters (L_{in}/W_1, W_2/W_1, X_2/X_1, Re_{in}).

1.5 Significant of Research

The following were the significant contributions of the research to the body of knowledge:

1. The prospective performance of 30° to 180° of 2-D and 3-D curved diffusers has been scientifically assessed.
2. The performance correlations representing the effects of both geometrical and operating parameters for 2-D and 3-D turning diffusers have been developed. These correlations may be utilised to evaluate the performance of 2-D and 3-D curved diffusers without running the full simulations or experiments.

REFERENCES

1. Nordin, N., Abdul Karim, Z. A., Othman, S., & Raghavan, V. R. (2013). Design and development of low subsonic wind tunnel for turning diffuser application. In *Advanced Materials Research* (Vol. 614, pp. 586-591). Trans Tech Publications Ltd.

2. Chong, T. P., Joseph, P. F., & Davies, P. O. A. L. (2008). A parametric study of passive flow control for a short, high area ratio 90deg curved diffuser. *Journal of Fluids Engineering*, *130*(11).

3. Nordin, N., Karim, Z. A. A., Othman, S., Raghavan, V. R., Batcha, M. F. M., Hariri, A., & Basharie, S. M. (2017, April). Flow characteristics of 3-D turning diffuser using particle image velocimetry. In *AIP Conference Proceedings* (Vol. 1831, No. 1, p. 020021). AIP Publishing LLC.

4. Gopaliya, M. K., & Chaudhary, K. K. (2010). CFD analysis of performance characteristics of Y-shaped diffuser with combined horizontal and vertical offsets. *Aerospace Science and Technology*, *14*(5), 338-347.

5. Saha, K., Singh, S. N., Seshadri, V., & Mukhopadhyay, S. (2007). Computational analysis on flow through transition S-diffusers: Effect of inlet shape. *Journal of aircraft*, *44*(1), 187-193.

6. Majumdar, B., Mohan, R., Singh, S. N., & Agrawal, D. P. (1998). Experimental study of flow in a high aspect ratio 90 deg curved diffuser.

7. Fox, R. W., & Kline, S. J. (1962). Flow regimes in curved subsonic diffusers.

8. Senoo, Y., & Nishi, M. (1977). Prediction of flow separation in a diffuser by a boundary layer calculation.

9. Sparrow, E. M., Abraham, J. P., & Minkowycz, W. J. (2009). Flow separation in a diverging conical duct: Effect of Reynolds number and divergence angle. *International Journal of Heat and Mass Transfer*, *52*(13-14), 3079-3083.

10. Hawthorne, W. R. (1951). Secondary circulation in fluid flow. *Proceedings of the Royal Society of London. Series A. Mathematical and Physical Sciences*, *206*(1086), 374-387.

11. Djebedjian, B. (2001). Numerical and experimental investigations of turbulent flow in a 180◦ curved diffuser. In *ASME Division of Fluid Dynamics Summer Meeting*.

12. Calautit, J. K., Chaudhry, H. N., Hughes, B. R., & Sim, L. F. (2014). A validated design methodology for a closed-loop subsonic wind tunnel. *Journal of Wind Engineering and Industrial Aerodynamics*, *125*, 180-194.

13. Nordin, N., Karim, Z. A. A., Othman, S., Raghavan, V. R., Batcha, M. F. M., Hariri, A., & Basharie, S. M. (2017, April). Flow characteristics of 3-D turning diffuser using particle image velocimetry. In *AIP Conference Proceedings* (Vol. 1831, No. 1, p. 020021). AIP Publishing LLC.

14. Sagi, C. J., & Johnston, J. P. (1967). The Design and Performance of Two-Dimensional, Curved Diffusers: Part I—Exposition of Method and Part II—Experiment, Evaluation of Method, and Conclusions.

15. Majumdar, B., Singh, S. N., & Agrawal, D. P. (1996). Flow characteristics in a large area ratio curved diffuser. *Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering*, *210*(1), 65-75.

16. Hamada, K. I. (2009). NUMERICAL VALIDATION ON THE PERFORMANCE OF A TWO DIMENSIONAL CURVED DIFFUSER. *Tikrit Journal of Engineering Sciences*, *16*(1), 105-120.

17. Parsons, D. J., & Hill, P. G. (1973). Effects of curvature on two-dimensional diffuser flow.

18. Nguyen, C. K., Ngo, T. D., Mendis, P. A., & Cheung, J. C. (2006). A flow analysis for a turning rapid diffuser using CFD.

19. Gan, G., & Riffat, S. B. (1996). Measurement and computational fluid dynamics prediction of diffuser pressure-loss coefficient. *Applied energy*, *54*(2), 181-195.

20. Shimizu, Y., Nagafusa, M., & Kuzuhara, S. (1982). Effects of approaching flow types on the performances of straight conical diffusers. *Bulletin of JSME*, *25*(208), 1506-1512.

21. Kumaraswamy, R., Natarajan, K., & Anand, R. B. (2021). CFD analysis of flow and performance characteristics of a 90 curved rectangular diffuser: effects of aspect ratio and Reynolds number. *International Journal of Turbo & Jet-Engines*, *38*(4), 451-463.

22. Majumdar, B., Singh, S. N., & Agrawal, D. P. (1996). Flow characteristics in a large area ratio curved diffuser. *Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering*, *210*(1), 65-75.

23. Nordin, N., Othman, S., Raghavan, V. R., & Karim, Z. A. A. (2012). Verification of 3-D stereoscopic PIV operation and procedures. *International Journal Engineering and Technology IJET/IJENS*, *12*(4), 19-26.

24. Taylor, A. M. K. P., Whitelaw, J. H., & Yianneskis, M. (1982). Curved ducts with strong secondary motion: velocity measurements of developing laminar and turbulent flow.

25. Namet-Allah, A., & Birk, A. M. (2014, June). Numerical and Experimental Study of Swirling Flow in a Short Annular to Round Diffuser/Nozzle. In *Turbo Expo: Power for Land, Sea, and Air* (Vol. 45578, p. V01AT01A034). American Society of Mechanical Engineers.

26. Sinha, P. K., Biswas, A. K., Mullick, A. N., & Majumdar, B. (2017). Flow development through a duct and a diffuser using CFD. *Int J Eng Res Appl*, *7*, 46-54.

27. Herwig, H., & Schäfer, P. (1992). Influence of variable properties on the stability of two-dimensional boundary layers. *Journal of Fluid Mechanics*, *243,* 1-14

28. Balaji, C., & Herwig, H. (2003). The use of ACFD approach problems involving surface radiation and free convection. *International communications in heat and mass transfer*, *30*(2), 251-259.

29. HEYWOOD, F. (1925). The Flow of Water in Pipes and Channels. In *Minutes of the Proceedings of the Institution of Civil Engineers* (Vol. 219, No. 1925, pp. 174-189). Thomas Telford-ICE Virtual Library.

30. Subramanian, G., Natarajan, S. K., Adhimoulame, K., & Natarajan, A. (2014). Comparison of numerical and experimental investigations of jet ejector with blower. *International journal of thermal sciences*, *84*, 134-142.

31. Cerantola, D. J., & Birk, A. M. (2013, June). Experimental validation of numerically optimized short annular diffusers. In *Turbo Expo: Power for Land, Sea, and Air* (Vol. 55232, p. V06BT38A005). American Society of Mechanical Engineers.