PARAMETRIC STUDY ON THE DESIGN OF BAFFLE FOR THREE-DIMENSIONAL TURNING DIFFUSER

NUR HAZIRAH BINTI NOH @ SETH

A thesis submitted in
fulfillment of the requirement for the award of the
Doctor of Philosophy

Faculty of Mechanical and Manufacturing Engineering
University Tun Hussein Onn Malaysia

NOVEMBER 2016
To my husband Mohd. Salehin Hj. Mazelan,

My better half,

Who helped me through thick and thin in this entire adventure

To my son Muhammad Nur Uwais Mohd Salehin,

My heaven and earth,

Who gave me strength to complete this thesis

To my heartbeat,


Who always pray for my success

Last and foremost,

To my beloved father Allahyarham Noh@Seth Hj. Ismail,

My inspiration
ACKNOWLEDGEMENT

I would like to express my sincere appreciation to my supervisor, Dr. Norasikin Mat Isa for all the guidance and support throughout the entire duration in completing this thesis. Thank you so much.

Credit should also be given to UTHM Aerodynamic Lab technician, Mr. Zainal for his cooperation and Dr. Normayati who taught me on hands on practical of PIV before running the experiment myself. Appreciation also goes to other PhD candidatures who keep on inspiring me; Zila, Najwa, Rasidi, Kak Syikin, Fatimah, Mun and many others. Thanks for the advices and help along the journey. May success be with all of us, all the time, Amiin.

This emotional journey would not be complete without the constant advices and helps from the best of friend; Ayong, Iza, Yuyun, Mira, Shahed, Wani, Nani, Efa, Jah, Sha, Zura and Ecah. Thank you for the constant instant messages regardless of the topic and always brings the best out of me. Knowing all of you is a blessed, but having all of you by my side throughout these years is always a miracle.
ABSTRACT

Secondary flow developed in the inner wall region within a turning diffuser will reduce its performance particularly in terms of both pressure recovery ($C_p$) and flow uniformity ($\sigma_u$). Introduction of baffle is effective in reducing separated flow in turning diffuser, hence enhance its performance. Therefore, flow structure in three-dimensional turning diffuser with baffle was studied and the subsequent impacts towards turning diffuser performance was observed. A parametric study was also conducted on the preliminary design of airfoil in determining the most optimum baffle design. An experiment was conducted with inflow Reynolds number ($Re_{in}$) that was varied between $4.527 \times 10^4$ and $1.263 \times 10^5$. As measured by using pressure tapping that was connected to a digital Manometer, a pressure recovery of $C_p=0.341$ was obtained when the system was operated at Reynolds number $Re_{in}=1.263 \times 10^5$. This result had shown an improvement of up to 43% compared to the previous study with pressure recovery $C_p=0.194$. Similarly, the flow uniformity which was measured by using Particle Image Velocimetry (PIV) had improved up to 33% at $Re_{in}=9.950 \times 10^4$ with $\sigma_u=3.09$ as compared to the previous study, where $\sigma_u=4.64$. A parametric study on the preliminary baffle design was also simulated using ANSYS Fluent, which had been verified and validated according to experimental data. The parametric study involved varying several parameters such as type of baffle, the angle of attack, AOA, thickness to chord ratio ($t/c$ (%)), camber to chord ratio ($f/c$ (%)), and chord length ($c$ (cm)). Simulations of various 23 designs with combination of several parameter changes had discovered an optimum design of airfoil with AOA=16°, $t/c = 7.658\%$, $f/c = 7\%$ and chord length, $c = 5$ cm. In comparison to the preliminary airfoil design, that optimum design for the three-dimensional turning diffuser had achieved 7.202% and 6.164% performance improvement in terms of flow uniformity and pressure recovery, respectively.
**ABSTRAK**

Aliran menengah yang terbentuk di rantau dinding dalam penyerap getaran akan menyebabkan prestasi menurun dari segi liputan tekanan ($C_p$) dan keseragaman aliran ($\sigma_u$). Pengenalan sesekat dapat membantu dalam mengurangkan aliran menengah dan memperbaiki prestasi penyerap getaran. Oleh itu, struktur aliran dalam penyerap getaran 3 dimensi dengan sesekat dikaji dan kesannya terhadap prestasi penyerap getaran disiasat. Eksperimen ke atas sesekat bentuk aerofoil permulaan yang diuji dengan nombor alir masuk Reynolds ($Re_{in}$) di antara 4.527E+04 – 1.263E+05 telah dijalankan, menghasilkan liputan tekanan $C_p=0.341$, dimana ianya diukur dengan menggunakan tekanan menoreh yang disambung ke Manometer digital, dicatatkan apabila sistem beroperasi dengan nombor Reynolds paling tinggi yang iaitu $Re_{in}=1.263E+05$. Keputusan ini menunjukkan peningkatan sehingga 43% daripada kajian lepas iaitu $C_p=0.194$ pada nombor Reynolds yang sama. Keseragaman aliran, yang diukur menggunakan *Particle Image Velocimetry (PIV)* juga menunjukkan peningkatan sebanyak 33% daripada kajian lepas pada $Re_{in}=9.950E+04$ iaitu $\sigma_u=3.09$ jika dibandingkan dengan kajian lepas iaitu $\sigma_u=4.64$.

Kajian parametrik ke atas reka bentuk permulaan sesekat dilakukan menggunakan simulas pada ANSYS Fluent, dimana keputusannya disahkan menggunakan nilai kajian dari eksperimen. Kajian parametrik merangkumi menukar jenis sesekat, sudut serang (AOA), nisbah tebal perentas, $t/c(\%)$, kamber nisbah perentas, $f/c(\%)$ dan panjang perentas, $c(cm)$. Simulasi ke atas 23 rekaan dengan pelbagai perubahan parameter menghasilkan reka bentuk optimum iaitu AOA=16°, t/c=7.658%, f/c=7% dan panjang perentas, c=5 cm. Reka bentuk optimum menyebabkan peningkatan prestasi penyerap getaran 3 dimensi sebanyak 7.202% dari segi keseragaman aliran jika dibandingkan dengan aerofoil permulaan dan peningkatan sebanyak 6.164% dari segi liputan tekanan.
CONTENTS

<table>
<thead>
<tr>
<th>CONTENT</th>
<th>PAGE</th>
</tr>
</thead>
<tbody>
<tr>
<td>TITLE</td>
<td>i</td>
</tr>
<tr>
<td>DECLARATION</td>
<td>ii</td>
</tr>
<tr>
<td>DEDICATION</td>
<td>iii</td>
</tr>
<tr>
<td>ACKNOWLEDGEMENT</td>
<td>iv</td>
</tr>
<tr>
<td>ABSTRACT</td>
<td>v</td>
</tr>
<tr>
<td>ABSTRAK</td>
<td>vi</td>
</tr>
<tr>
<td>CONTENTS</td>
<td>vii</td>
</tr>
<tr>
<td>LIST OF TABLES</td>
<td>xi</td>
</tr>
<tr>
<td>LIST OF FIGURES</td>
<td>xiv</td>
</tr>
<tr>
<td>LIST OF SYMBOLS</td>
<td>xxii</td>
</tr>
<tr>
<td>LIST OF ABBREVIATIONS</td>
<td>xxiv</td>
</tr>
<tr>
<td>LIST OF APPENDICES</td>
<td>xxvi</td>
</tr>
</tbody>
</table>

CHAPTER 1 INTRODUCTION

1.1 Research background                             4
1.2 Problem Statement                                6
1.3 Objectives of study                              7
1.4 Scope of study                                   7
1.5 Significance of study                            8
1.6 Thesis outline                                   9
## CHAPTER 2 LITERATURE REVIEW

2.1 Diffuser applications and turning diffuser theoretical background  
2.2 Experimental rig development on low subsonic wind tunnel feature  
2.3 Various baffles design  
2.4 PIV instrumentations  
2.5 Numerical analysis solutions

## CHAPTER 3 METHODOLOGY: EXPERIMENTAL APPROACH

3.1 Experimental method  
3.1.1 Experimental setup  
3.1.2 Centrifugal blower  
3.1.3 Settling chamber and screens  
3.1.4 Contraction cone  
3.1.5 Rectangular duct  
3.1.6 Turning diffuser with airfoil baffle  
3.1.7 Initial condition measurement  
3.1.8 Pressure recovery and efficiency measurement  
3.2 PIV measurement and instrumentation  
3.2.1 2D PIV setup  
3.2.2 3D PIV setup  
3.2.3 Tracer particle seeding  
3.2.4 Illumination  
3.2.5 Photographing  
3.2.6 Image processing

## CHAPTER 4 METHODOLOGY: NUMERICAL METHOD

4.1 Description of the whole system  
4.2 Mathematical model  
4.3 Computational domain  
4.4 Domain discretization  
4.5 Boundary and operating conditions
4.6 Solver and solution algorithm 82
4.7 Convergence criteria 84

CHAPTER 5 EXPERIMENTAL DATA ANALYSIS 89
5.1 Design of baffles 90
5.2 Measurement of initial condition 92
5.3 Measurement of static pressure recovery 93
5.4 Flow structure in three-dimensional turning diffuser 95
5.5 Outlet flow uniformity, $\sigma_{out}$ measurement 111
5.6 Turning diffuser efficiency 120

CHAPTER 6 NUMERICAL DATA ANALYSIS 123
6.1 Numerical work on 2D turning diffuser with baffles 123
6.1.1 Computational domain 124
6.1.2 Domain discretization 124
6.1.3 Flow structure in two-dimensional turning diffuser with baffles 126
6.2 Numerical work on 3D turning diffuser with baffle 130
6.2.1 Computational domain 130
6.2.2 Domain discretization 131
6.2.3 Flow structure in the three-dimensional turning diffuser with baffle 132
6.2.4 Outlet plane velocity contour 141

CHAPTER 7 VERIFICATION AND VALIDATION 143
7.1 Verification of the two-dimensional turning diffuser solutions 144
7.1.1 Grid independence study on two-dimensional turning diffuser with baffles 144
7.1.2 Turbulence model test on two-dimensional turning diffuser with baffles 147
7.1.3 Numerical scheme test on two-dimensional turning diffuser with baffles 149
7.1.4 Convergence criteria test on two-dimensional turning diffuser with baffles 150
7.2 Validation of the two-dimensional turning diffuser with baffles 152
7.3 Verification of the three-dimensional turning diffuser solutions 155
7.3.1 Grid independence study on three-dimensional turning diffuser with baffle
7.3.2 Turbulence model test on three-dimensional turning diffuser with baffle
7.3.3 Numerical scheme test on three-dimensional turning diffuser with baffle
7.3.4 Convergence criteria test on three-dimensional turning diffuser with baffle
7.4 Validation on three-dimensional turning diffuser with baffle solutions

CHAPTER 8 CFD STUDY ON VARIOUS BAFFLE DESIGNS
8.1 Parametric study on the design of baffle
8.2 Parametric study on airfoil angle of attack (AOA)
8.3 Parametric study on thickness to chord ratio, t/c (%)
8.4 Parametric study on camber to chord ratio, f/c (%)
8.5 Parametric study in chord length, c (cm)

CHAPTER 9 CONCLUSIONS AND FUTURE WORK
9.1 Conclusions
9.2 Contributions
9.3 Recommendations on future work

REFERENCES

LIST OF PUBLICATIONS

APPENDICES
# LIST OF TABLES

2.1 Pressure loss coefficient ($K$) (Nordin et al., 2011) 18  
2.2 $C_p$ measured for each $Re_i$ tested and verification of PIV results for two-dimensional turning diffuser (Nordin et al., 2014a) 20  
2.3 Result comparison between Noh@Seth et al. (2013) and Nordin et al. (2014a) for both $C_p$ and $\sigma_{out}$ 22  
2.4 Mean outlet velocity, $V_{out}$ and flow uniformity comparison, $\sigma_{out}$ (Nordin et al., 2014a) 24  
2.5 Pressure recovery, $C_p$ comparison (Nordin et al., 2014a) 24  
3.1 Input parameters for centrifugal blower 45  
4.1 ANSYS Fluent input parameters 86  
5.1 Preliminary airfoil parameters 91  
5.2 Measurement of inlet conditions 93  
5.3 Measurement of static $P_i$ and $P_o$ 93  
5.4 Comparison of $C_p$ 94  
5.5 Selection of $\Delta t$ 96  
5.6 $U_{max}$ comparison for validation of $\Delta t$. Optimum $\Delta t$ is marked with (*) 97  
5.7 $U_o$ extracted along Line 1 for the selected $\Delta t$ 98  
5.8 Comparison of $U_{in}$ measurement 102  
5.9 $V_o$ comparison for validation of $\Delta t$. Optimum $\Delta t$ were marked with (*) 113  
5.10 $V_o$ extracted at outlet plane for selected $\Delta t$ 114  
5.11 $V_{out}$ (m/s) comparison with previous study 116  
5.12 Calculated value of diffuser efficiency, $\eta$ for the present study 121  
5.13 Calculated value of diffuser efficiency, $\eta$ for Nordin et al. (2014a) 121
6.1 Calculation of first layer thickness for 2-dimensional turning diffuser with baffle 125
6.2 Turbulence intensity measurement at both inlet and outlet 126
6.3 Calculation of first layer thickness for three-dimensional turning diffuser with baffle 131
7.1 Three types of meshes tested for grid independence study 146
7.2 Numerical scheme CPU hours needed to reach convergence 149
7.3 Convergence criteria CPU hours needed to reach convergence 151
7.4 Comparison of $P_t$ 152
7.5 Comparison of $P_o$ 152
7.6 Comparison of $C_p$ 153
7.7 Comparison of $V_{out}$ 153
7.8 Experimental value of the two-dimensional turning diffuser with baffle efficiency, $\eta$ 154
7.9 CFD value of the two-dimensional turning diffuser with baffle efficiency, $\eta$ 154
7.10 Comparison of $\eta$ 155
7.11 Three types of meshes tested in grid independence study 158
7.12 Numerical scheme CPU hours needed to reach convergence 160
7.13 Convergence criteria CPU hours needed to reach convergence 162
7.14 Comparison of $P_t$ 163
7.15 Comparison of $P_o$ 164
7.16 Comparison of $C_p$ 164
7.17 Comparison of $V_{out}$ 165
7.18 CFD value of three-dimensional turning diffuser with baffle efficiency, $\eta$ 165
7.19 Comparison of $\eta$ 165
8.1 Design parameters changes in determining optimum design of baffle 168
8.2 Static pressure comparison between flat plate and airfoil baffle 171
8.3 $C_p$ comparison between flat plate and airfoil baffle 171
8.4 Mean outlet velocity, $V_{out}$ comparison between flat plate and airfoil 172
8.5 $\sigma_{out}$ comparison between both designs of baffle 172
8.6 $\eta$ comparison between both designs of baffle 173
8.7 Turning diffuser performance comparison for airfoil with various AOA tested

8.8 Turning diffuser efficiency, $\eta$ comparison for airfoil with various AOA tested

8.9 Drag coefficient measured for airfoil with various AOA tested

8.10 Turning diffuser performance for each airfoil with various AOA tested

8.11 Turning diffuser efficiency, $\eta$ comparison for airfoils with various AOA tested

8.12 Drag coefficient, $C_d$ measured for each AOA

8.13 Airfoil characteristics tested

8.14 Turning diffuser performance for each airfoil tested

8.15 Turning diffuser efficiency, $\eta$ comparison for each $t/c$ tested

8.16 Drag coefficient measured for each airfoil

8.17 Airfoil tested in changes of $f/c$

8.18 Turning diffuser performance for each airfoil tested

8.19 Turning diffuser efficiency, $\eta$ comparison for each $f/c$ tested

8.20 Drag coefficient measured for each airfoil

8.21 Airfoil properties tested in changes in chord length, $c$

8.22 Turning diffuser performance for each airfoil tested

8.23 Turning diffuser efficiency, $\eta$ comparison for each $c$ tested

8.24 Drag coefficient measured for each airfoil

8.25 Performance comparison between base airfoil and optimum airfoil

9.1 Design parameters of the optimum baffle
## LIST OF FIGURES

1.1 Schematic diagram of experimental circulating fluidised bed including diffuser (Schut et al., 2000)  

1.2 Test rig measurement of diffuser’s pressure loss coefficient in HVAC free discharge duct system (Gan & Riffat, 1996)  

1.3 Closed loop subsonic wind tunnel detailed computer-aided design CAD model (Calautit et al., 2014)  

1.4 Design of the two-dimensional turning diffuser (Nordin et al., 2012)  

1.5 Design of the three-dimensional turning diffuser (Nordin et al., 2012)  

2.1 Two different positions of diffuser in the riser; (a) 550 cm below the exit and (b) 1050 cm below the exit (Schut et al., 2000)  

2.2 Bend-diffuser combination with short spacer shows highly distorted flow at diffuser exit for both studies by; (a) Gan & Riffat (1996), (b) El-Askary & Nasr (2009)  

2.3 (a) Contours of velocity magnitude for wind tunnel before and (b) after installation of guide vanes (Calautit et al., 2014)  

2.4 Location chosen for measurement; S1, S2, S3 (Nordin et al., 2011)  

2.5 Two planes selected for each location measurement (Nordin et al., 2011)  

2.6 Rig development of low subsonic wind tunnel. All dimensions in cm. (Nordin et al., 2013)  

2.7 (a) Five points location at inlet for measurements of fully developed flow and (b) velocity profile measured using Pitot static probe (Nordin et al., 2014b)  

2.8 Flat plate baffle designed by Noh@Seth et al. (2013). All dimensions in cm.
2.9 Outlet velocity contour and vector comparison between (a) Nordin et al. (2014a) and (b) Noh@Seth et al. (2013). Red box indicates inner wall region.

2.10 Schematic diagram of experimental setup using diffusing bend (Friedman & Westphal, 1952)

2.11 Sketch showing airfoil location and cascade end condition (Friedman & Westphal, 1952)

2.12 Airfoil profile (Friedman & Westphal, 1952)

2.13 Computer velocity vectors on sharp 180° bend duct (a) without guide vane, and (b) with one guide vane (Modi & Jayanti, 2004)

2.14 The cascade vane geometry used by Lindgren et al. (1998); \(d=\text{spacing between vanes}, \ c=\text{chord length}, \ h_0=\text{spacing between vanes perpendicular to outflow direction}, \ h_1=\text{spacing between vanes perpendicular to inflow direction}\)

2.15 Solid line guide vane is the optimised guide vane for non-expanding corners. Dashed line is the new vane optimised for expanding corners. (Lindgren & Johansson, 2002)

2.16 Mean velocity contour around airfoil for (a) AOA=0°, (b) AOA=6° and (c) AOA=15°. RA=reattachment, S=separation. (Nakano et al., 2007)

2.17 Geometry of cascades (Schreiber et al., 2004)

2.18 Comparison of leading edge geometry (Schreiber et al., 2004)

2.19 Schematic diagram of the turning vanes in Mathew (2006)

2.20 Schematic diagram of stereoscopic PIV setup taken from Prasad (2000)

2.21 Validation process (Standard A.I.A.A, 1999)

2.22 Verification process (Standard A.I.A.A, 1999)

2.23 (a) Full scale of entire closed circuit wind tunnel; (b) Test section for wind tunnel with inlet and outlet boundary condition (Moonen et al., 2006).

2.24 Instantaneous velocity field obtained by LES (Jakirlić et al., 2010)

3.1 Schematic diagram of the experimental setup (Nordin et al., 2013)

3.2 Actual diagram of the experimental setup consists of; (a) centrifugal blower, settling chamber with screens and...
contraction cone, (b) rectangular duct and turning diffuser with baffle

3.3 Centrifugal blower in Aerodynamic Lab, UTHM

3.4 Settling chamber with 4 units of mesh screen attached to the contraction cone

3.5 200 m long rectangular duct

3.6 Schematic diagram of the three-dimensional turning diffuser with preliminary airfoil baffle. All dimensions are not up to scale.

3.7 Three-dimensional turning diffuser with baffle made up of 3 mm acrylic plate

3.8 Schematic diagram of a Pitot static tube (Elyasi, 2009)

3.9 FLUKE 922 digital air flow meter and its function

3.10 Centre point of turning diffuser inlet, $X_c$

3.11 Pitot tube connection to digital manometer (Appendix C)

3.12 Pitot tube connected to digital manometer (left) and measurement of $\Delta P_{dyn}$ at point $X_c$ at turning diffuser inlet (right)

3.13 (a) Pressure tapping at outlet and (b) inlet connected to digital Manometer using Triple-T piezometer tube

3.14 Basic concept of the 2D PIV system (Adrian, 2005)

3.15 PIV post processing and data analysis technique (Appendix D)

3.16 2D PIV setup

3.17 3D stereoscopic setup

3.18 Local outlet plane of interest points $V_0$, $V_1$, $V_2$, $V_3$ and $V_4$

3.19 Pitot static reading at the outlet for validation with PIV values at $V_0$ and $V_1$

3.20 Medium density tracer particle seeding in PIV frame (Raffel et al., 2013)

3.21 Dantec F2010 Safex fog generator with remote control

3.22 Illumination parts from Dantec; (a) DualPower Laser screw-pumped cooling system, (b) Laser light guiding arm (c) Laser optic and (d) Nd:YAG Laser source

3.23 Calibration plate

3.24 Overlapping camera position for 3D stereoscopic setup
3.25 5 orientations of calibration plate for calibration in 3D stereoscopic PIV; (a) 20° incline to east side, (b) 20° incline to west side, (c) 20° incline to north side, (d) 20° incline to south side and (e) 0° flat position

3.26 Successful calibration (a) pinhole Image Model Fit (IMF) for 2D PIV and (b) Direct Linear Transform (DLT) for 3D PIV

3.27 Turning diffuser divided into 3 planes for 2D PIV setup

3.28 Masked area around turning diffuser for all planes in 2D PIV setup; (a) Plane 1, (b) Plane 2 and (c) Plane 3

3.29 Minor masked area around the outlet for 3D PIV setup from; (a) Camera 1 and (b) Camera 2

3.30 Vector flow in Plane 1 after image processing for 2D PIV setup

3.31 Vector statistic from (a) Camera 1 and (b) Camera 2 combined to produced (c) stereoscopic third velocity component contour, $V_o$

4.1 Extended CFD domain for three-dimensional (a) and two-dimensional (c) turning diffuser and actual geometry in experiment for three-dimensional (b) and two-dimensional (d) turning diffuser. All dimensions are in cm.

4.2 Turbulent flow boundary layer (ANSYS, 2010)

4.3 Value of $y+$ in turbulent flow boundary layers (ANSYS, 2010)

4.4 Enhanced wall treatment in the sub-layer zone (ANSYS, 2010)

4.5 SIMPLE algorithm flow diagram (Versteeg & Malalasekera, 2007)

5.1 Schematic diagram of three-dimensional turning diffuser

5.2 (a) Airfoil parameters and (b) location of airfoil baffle in three-dimensional turning diffuser

5.3 (a) Plane 1 image location and (b) mask image with seeded particle flow

5.4 Point of $X_c$ data acquisition to be compared with Pitot static probe reading and location of Line 1 at Plane 1

5.5 Flow structures on Plane 1 for all $Re_{in}$; (a) 4.527E+04, (b) 5.110E+04, (c) 7.580E+04, (d) 9.950E+04 and (e) 1.263E+05

5.6 Velocity contour at Plane 1 extracted for all $Re_{in}$; (a) 4.527E+04, (b) 5.110E+04, (c) 7.580E+04, (d) 9.950E+04 and (e) 1.263E+05.
5.7 Local inlet velocity, $U_o$ plot along Line 1 together with $U_{max}$ measured by Pitot static reading for comparison

5.8 (a) Original image location of Plane 2; (b) and (c) shows flow with seeded particle image containing reflections due to laser light sheet

5.9 Flow structure at Plane 2 for all $Re_{in}$ tested; (a) 4.527E+04, (b) 5.110E+04, (c) 7.580E+04, (d) 9.950E+04 and (e) 1.263E+05

5.10 Velocity contour at Plane 2 for all $Re_{in}$ tested; (a) 4.527E+04, (b) 5.110E+04, (c) 7.580E+04, (d) 9.950E+04 and (e) 1.263E+05

5.11 (a) Plane 3 desired location and (b) seeded flow after image masking

5.12 Flow structure at Plane 3 from 2D PIV setup for all $Re_{in}$ tested; (a) 4.527E+04, (b) 5.110E+04, (c) 7.580E+04, (d) 9.950E+04 and (e) 1.263E+05

5.13 Flow structure comparison at $Re_{in}=4.527E+04$ between; (a) turning diffuser with baffle and (b) turning diffuser without baffle (Nordin et al., 2014a)

5.14 Flow structure comparison at $Re_{in}=1.263E+05$ between; (a) turning diffuser with baffle and (b) turning diffuser without baffle (Nordin et al., 2014a)

5.15 Both cameras mounted on traverse by using Scheimpflug rule facing outlet plane of interest

5.16 Points taken at the outlet for validation of $\Delta t$ selection

5.17 Local third velocity component, $V_o$ contour comparison between present study and previous study (Nordin et al., 2014a) for; $Re_{in}=4.527E+04$ ((a) and (c) respectively) and $Re_{in}=1.263E+05$ ((b) and (d) respectively).

5.18 Outlet plane of interest and allocated position of Line A and Line B

5.19 Local outlet velocities, $V_0$ plotted graph along Line A; (a) present study, (b) Nordin et al. (2014a)

5.20 Local outlet velocities, $V_0$ plotted graph along Line B; (a) present study, (b) Nordin et al. (2014a)

6.1 Two-dimensional turning diffuser with baffles computational domain

6.2 Two-dimensional turning diffuser with baffle constructed mesh with Enhanced Wall Treatment (EWT) function
6.3 Flow structures on Plane A for all $Re_{in}$; (a) 4.527E+04, (b) 5.110E+04, (c) 7.580E+04, (d) 9.950E+04 and (e) 1.263E+05

6.4 Velocity contour on Plane A for all $Re_{in}$; (a) 4.527E+04, (b) 5.110E+04, (c) 7.580E+04, (d) 9.950E+04 and (e) 1.263E+0

6.5 Three-dimensional turning diffuser with baffle computational domain

6.6 Three-dimensional turning diffuser with baffle constructed mesh with Enhanced Wall Treatment (EWT) function

6.7 Flow structure at Plane 1 for all $Re_{in}$; (a) 4.527E+04, (b) 5.110E+04, (c) 7.580E+04, (d) 9.950E+04 and (e) 1.263E+0

6.8 Velocity contour at Plane 1 for all $Re_{in}$; (a) 4.527E+04, (b) 5.110E+04, (c) 7.580E+04, (d) 9.950E+04 and (e) 1.263E+0

6.9 Flow structure at Plane 2 for all $Re_{in}$; (a) 4.527E+04, (b) 5.110E+04, (c) 7.580E+04, (d) 9.950E+04 and (e) 1.263E+0

6.10 Velocity contour at Plane 2 for all $Re_{in}$; (a) 4.527E+04, (b) 5.110E+04, (c) 7.580E+04, (d) 9.950E+04 and (e) 1.263E+0

6.11 Flow structure at Plane 3 for all $Re_{in}$; (a) 4.527E+04, (b) 5.110E+04, (c) 7.580E+04, (d) 9.950E+04 and (e) 1.263E+0

6.12 Velocity contour at Plane 3 for all $Re_{in}$; (a) 4.527E+04, (b) 5.110E+04, (c) 7.580E+04, (d) 9.950E+04 and (e) 1.263E+0

6.13 Velocity contour at outlet plane for all $Re_{in}$; (a) 4.527E+04, (b) 5.110E+04, (c) 7.580E+04, (d) 9.950E+04 and (e) 1.263E+0

7.1 Mesh 1 unstructured grid for two-dimensional turning diffuser with baffles

7.2 Mesh 2 unstructured grid for two-dimensional turning diffuser with baffles

7.3 Mesh 3 unstructured grid for two-dimensional turning diffuser with baffles

7.4 Local outlet velocity, $V_0$ plotted along Line A for grid independent study test

7.5 Local outlet velocity, $V_0$ plotted along Line A for turbulence model test

7.6 Local outlet velocity, $V_0$ plotted along Line A for numerical scheme test

7.7 Local outlet velocity, $V_0$ plotted along Line A for convergence criteria test

7.8 Mesh 1 unstructured grid for three-dimensional turning diffuser with baffle
7.9 Mesh 2 unstructured grid for three dimensional turning diffuser with baffle
7.10 Mesh 3 unstructured grid for three-dimensional turning diffuser with baffle
7.11 Local outlet velocity, $V_0$ plotted along line A for grid independent study test
7.12 Local outlet velocity, $V_0$ plotted along line A for turbulence model test
7.13 Local outlet velocity, $V_0$ plotted along Line A for numerical scheme test
7.14 Local outlet velocity, $V_0$ plotted along Line A for convergence criteria test
8.1 (a) Location of the leading edge and separation point and (b) preliminary airfoil dimensions used in both the experiment and simulation
8.2 Flat plate baffle design in three-dimensional turning diffuser (all dimensions in cm); (a) side view and (b) isometric view
8.3 Flat plate baffle velocity contour at Plane A for all $Re_{in}$; (a) 4.527E+04, (b) 5.110E+04, (c) 7.580E+04, (d) 9.950E+04 and (e) 1.263E+05
8.4 Airfoil baffle velocity contour at Plane A for all $Re_{in}$; (a) 4.527E+04, (b) 5.110E+04, (c) 7.580E+04, (d) 9.950E+04 and (e) 1.263E+05
8.5 Location for plane of interest, Plane B
8.6 Flat plate baffle velocity contour at Plane B for all $Re_{in}$; (a) 4.527E+04, (b) 5.110E+04, (c) 7.580E+04, (d) 9.950E+04 and (e) 1.263E+05
8.7 Airfoil baffle velocity contour at Plane B for all $Re_{in}$; (a) 4.527E+04, (b) 5.110E+04, (c) 7.580E+04, (d) 9.950E+04 and (e) 1.263E+05
8.8 Local outlet velocity, $V_0$ plot along Line A comparison between flat plate and airfoil baffle for $Re_{in}$=4.527E+04 and $Re_{in}$=1.263E+05
8.9 Local outlet velocity, $V_0$ plot along Line B comparison between flat plate and airfoil baffle for $Re_{in}$=4.527E+04 and $Re_{in}$=1.263E+05
8.10 Allocated position of Line A, Line B and Line C
8.11 Local outlet velocity, \( V_0 \) plot along Line C comparison between flat plate and airfoil baffle for \( Re_{in}=4.527\times10^4 \) and \( Re_{in}=1.263\times10^5 \)

8.12 Velocity contour at outlet plane (flat plate baffle) for all \( Re_{in} \); 
(a) \( 4.527\times10^4 \), (b) \( 5.110\times10^4 \), (c) \( 7.580\times10^4 \), (d) \( 9.950\times10^4 \) and (e) \( 1.263\times10^5 \)

8.13 Different angle of attack tested in simulation

8.14 Location of data for \( C_{pa} \) plot around airfoil

8.15 \( C_{pa} \) plot around airfoil crossing z-axis=0

8.16 Velocity contour at Plane A for airfoil with; (a) AOA=11°, (b) AOA=14°, (c) AOA=17° (preliminary), (d) AOA=20° and (e) AOA=23°

8.17 Velocity contour at Plane B for airfoil with; (a) AOA=11°, (b) AOA=14°, (c) AOA=17° (preliminary), (d) AOA=20° and (e) AOA=23°

8.18 Outlet velocity contour at multiple positions for airfoil with AOA=23°

8.19 Outlet velocity contour at multiple positions for airfoil with AOA=20°

8.20 Outlet velocity contour at multiple positions for airfoil with AOA=17° (preliminary)

8.21 Different angle of attack tested in simulation

8.22 \( C_{pa} \) plot around airfoil across z-axis=0

8.23 Velocity contour at Plane B for airfoil with; (a) AOA=16°, (b) AOA=15°, (c) AOA=14°, (d) AOA=13° and (e) AOA=12°

8.24 Airfoil design with different t/c tested

8.25 \( C_{pa} \) plot around airfoil across z-axis=0

8.26 Velocity contour at Plane B; (a) Airfoil A, (b) Airfoil B, (c) Airfoil Base, (d) Airfoil C, (e) Airfoil D and (f) Airfoil E

8.27 Airfoil designs with different f/c tested

8.28 \( C_{pa} \) plot around airfoil across z-axis=0

8.29 Velocity contour at Plane B for; (a) Airfoil F, (b) Airfoil G, (c) Airfoil Base, (d) Airfoil H and (e) Airfoil I.

8.30 Airfoil designs with different chord length, \( c \) tested

8.31 \( C_{pa} \) plot around airfoil with different chord length, \( c \)

8.32 Velocity contour at Plane B for all airfoil; (a) Airfoil J, (b) Airfoil K, (c) Airfoil Base, (d) Airfoil L and (e) Airfoil M
LIST OF SYMBOLS

A - Current
A1 - Inlet area
A2 - Outlet area
c - Chord length
Cd - Drag coefficient
Cp - Turning diffuser pressure recovery coefficient
Cpa - Airfoil pressure coefficient
Cp_ideal - Pressure recovery for ideal turning diffuser
d - Spacing between vanes in cascade design
Dh - Hydraulic diameter
f/c - Camber-to-chord ratio
h0 - Spacing between vanes perpendicular to outflow direction
h1 - Spacing between vanes perpendicular to inflow direction
I - Turbulence intensity, %
K - Loss coefficient
k - Kinetic energy coefficient
L_in - Inner wall length
L_m - Centreline length
L_m/W_1 - Centreline length to inlet width ratio
L_out - Outer wall length
N - Number of measurement points
P_atm - Atmospheric pressure = 101325 Pa
P_gauge - Gauge pressure
P_i - Inlet average static pressure
P_o - Outlet average static pressure
\( Re_{in} \) - Inlet Reynolds number
\( t \) - Airfoil thickness
\( f \) - Camber thickness
\( t/c \) - Thickness-to-chord ratio
\( U_{\text{max}} \) - Inlet maximum velocity
\( U_{in} \) - Inlet mean velocity
\( U_o \) - Inlet local velocity
\( V_o \) - Local outlet velocity
\( V_{\text{out}} \) - Mean outlet velocity
\( V_p \) - Particle velocity
\( \nu \) - Air kinematic viscosity, 1.60E-05 m\(^2\)/s
\( c_f \) - Skin friction coefficient
\( W_1 \) - Inlet width
\( W_2 \) - Outer width
\( \ell_h \) - Hydrodynamic entrance length
\( y_p \) - First layer thickness
\( y^+ \) - Size of grid cell nearest to the wall
\( \sigma_{\text{out}} \) - Outlet flow uniformity
\( \eta \) - Diffuser efficiency
\( \rho \) - Air density, 1.164 kg/m\(^3\)
\( \mu \) - Air dynamic viscosity, 1.86E-05 kg/ms
\( \lambda \) - Laser wavelength
\( \Delta \phi \) - Turning angle
\( \varepsilon \) - Dissipation rate
\( \Delta P_{\text{dyn}} \) - Dynamic pressure of moving fluid
\( \Delta x \) - Particle displacement between two consecutive images
\( \Delta t \) - Time between two light pulses
**LIST OF ABBREVIATIONS**

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>AR</td>
<td>Area ratio</td>
</tr>
<tr>
<td>AOA</td>
<td>Angle of attack</td>
</tr>
<tr>
<td>CAD</td>
<td>Computer-aided Design</td>
</tr>
<tr>
<td>CCD</td>
<td>Charge-coupled device</td>
</tr>
<tr>
<td>CDA</td>
<td>Controlled diffusion concept</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational fluid dynamic</td>
</tr>
<tr>
<td>DEHS</td>
<td>Di-Ethyl-Hexyl-Sabacar</td>
</tr>
<tr>
<td>DLT</td>
<td>Direct linear transform</td>
</tr>
<tr>
<td>ES</td>
<td>Evolution strategy</td>
</tr>
<tr>
<td>EVM</td>
<td>Eddy viscosity model</td>
</tr>
<tr>
<td>EWT</td>
<td>Enhanced Wall Treatment</td>
</tr>
<tr>
<td>HVAC</td>
<td>Heating, Ventilation and Air Conditioning</td>
</tr>
<tr>
<td>IMF</td>
<td>Image model fit</td>
</tr>
<tr>
<td>LES</td>
<td>Large eddy simulation</td>
</tr>
<tr>
<td>MOGA</td>
<td>Multi objective generic algorithm</td>
</tr>
<tr>
<td>Nd:YAG</td>
<td>Neodym-yttrium-aluminium-garnet</td>
</tr>
<tr>
<td>OGV</td>
<td>Outlet guide vanes</td>
</tr>
<tr>
<td>PIV</td>
<td>Particle Image Velocimetry</td>
</tr>
<tr>
<td>RA</td>
<td>Reattachment point</td>
</tr>
<tr>
<td>RANS</td>
<td>Reynolds-averaged Navier-Stokes</td>
</tr>
<tr>
<td>RNG</td>
<td>Renormalization group turbulence model</td>
</tr>
<tr>
<td>RPM</td>
<td>Revolution per minute</td>
</tr>
<tr>
<td>RSM</td>
<td>Reynolds stress model</td>
</tr>
<tr>
<td>S</td>
<td>Separation point</td>
</tr>
<tr>
<td>SIMPLEC</td>
<td>Semi-Implicit Method for Pressure-Linked Equation</td>
</tr>
</tbody>
</table>
SKE - Standard K-Epsilon turbulence model
RKE - Realizable K-Epsilon turbulence model
SST - Shear stress transport model
UTHM - Universiti Tun Husein Onn Malaysia
# LIST OF APPENDICES

<table>
<thead>
<tr>
<th>APPENDIX</th>
<th>TITLE</th>
<th>PAGE</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>DEHS MSDS</td>
<td>231</td>
</tr>
<tr>
<td>B</td>
<td>Blower Manual</td>
<td>233</td>
</tr>
<tr>
<td>C</td>
<td>Digital Manometer Manual</td>
<td>235</td>
</tr>
<tr>
<td>D</td>
<td>PIV catalogue</td>
<td>239</td>
</tr>
<tr>
<td>E</td>
<td>3D traversing system specifications</td>
<td>240</td>
</tr>
<tr>
<td>F</td>
<td>Fog generator and fog fluid specifications</td>
<td>243</td>
</tr>
<tr>
<td>G</td>
<td>DualPower laser specifications</td>
<td>247</td>
</tr>
<tr>
<td>H</td>
<td>FlowSense 2M specifications</td>
<td>255</td>
</tr>
<tr>
<td>I</td>
<td>Turning diffuser with baffle detail design</td>
<td>257</td>
</tr>
<tr>
<td>J</td>
<td>Stereoscopic PIV contour</td>
<td>261</td>
</tr>
<tr>
<td>K</td>
<td>$V_0$ along Line A, Line B and Line C</td>
<td>264</td>
</tr>
</tbody>
</table>
CHAPTER 1

INTRODUCTION

Investigation of flow patterns and flow characteristics for internal and external fluid flow has been of interest to researchers all around the world. The study of fluid mechanics deals with the action of forces on fluids, which in contrast to solids, can deform and flow under the action of shear stress. Such flows offer a lot of interesting topics to be discussed, especially when considering the vital role fluid mechanics plays in our everyday lives.

The diffuser, for example, is one of the steady flow engineering devices introduced in fluid flow systems, which has the simplest design of an expanding area in the flow direction. By slowing down the flow and consequently resulting in the recovery of static pressure (Ghose, Datta, & Mukhopadhyay, 2013), following the conservation of energy, the diffuser’s basic function is to convert kinetic energy into potential energy (Azad, 1996; Lee et al., 2013).

To minimize the weight and size of the engine, aviation gas turbine, for example, uses dump diffusers in the combustor (Ghose et al., 2013). On the other hand, in circulating fluidized bed application, as the lower section has a smaller cross-section as compared to the upper section, the diffuser is mounted, acting as a connector for both parts as shown in Figure 1.1 (Schut et al., 2000).
Figure 1.1: Schematic diagram of experimental circulating fluidised bed including diffuser (Schut et al., 2000)

Turning diffuser was favourable when involved with space restrictions applications (Gopaliya & Chaudhary, 2010). Intake ducts for aircraft engines use an S-shaped diffuser which also function as an interconnector between components in gas turbine engines (Mohamed, Djebedjian, & Rayan, 2000). In the heating, ventilation and air conditioning (HVAC) duct, the free discharge diffuser was used at the duct outlet system in order to reduce the air velocity when discharged into the atmosphere (Gan & Riffat, 1996) as shown in Figure 1.2. With a proper design compatibility test, both the bend and diffuser can be combined into a turning diffuser, especially when compactness is desired in the system.
The same concept was applied for the closed loop subsonic wind tunnel system. The diffuser in the closed loop wind tunnel was located downstream of the test section. In order to minimize loss of kinetic energy in the flow, the diffuser decelerates the flow after the test section (Calautit et al., 2014). As shown in Figure 1.3, the area covered by the closed loop subsonic wind tunnel can be reduced if both the 90º downstream turn and diffuser were combined into a turning diffuser.
1.1 Research background

There are two types of turning diffusers, namely the two-dimensional turning diffuser and the three-dimensional turning diffuser. Flow structure in the three-dimensional turning diffuser has been proven to be more distorted as compared to the two-dimensional turning diffuser (Nordin et al., 2014a). Consequently, higher pressure loss occurs in the three-dimensional turning diffuser due to curvature effects and diffusing activities. Both types of turning diffusers used in the present study closely resemble the turning diffuser used in a previous study (Chong, Joseph, & Davies, 2008; Nordin et al., 2014a).

The dimensions of the inlet surface area of a diffuser are denoted by $W_1$ and $X_1$ while the outlet dimensions are denoted by $W_2$ and $X_2$. The two-dimensional turning diffuser has expanding cross-section in y-z plane where the length of $X_1$ and $X_2$ remain the same. Figure 1.4 and Figure 1.5 show the design of the two-dimensional turning diffuser and the three-dimensional turning diffuser.

On the other hand, the three-dimensional turning diffuser has different lengths for all $W_1$, $W_2$, $X_1$, and $X_2$. It has expanding cross section in both x-y and y-z planes. Due to this, the flow structure for the three-dimensional turning diffuser is much more complex to be investigated. Other than pressure recovery coefficient $C_p$, turning diffuser performance can be measured by calculating standard deviation of the outlet flow, $\sigma_{out}$. As a square root of variance in probability distribution (Othman, Wahab, & Raghavan, 2012), standard deviation represents variation of local outlet velocity, $V_o$, to the mean outlet velocity, $V_{out}$.

Flow structure in turning diffuser is strongly dependent on turning angle ($\Delta\phi$), area ratio (AR), and inlet Reynolds number ($Re_{in}$). To avoid severe flow separation for both types of turning diffuser, 90° turning angle with AR=2.16 were selected as the optimum parameters of both turning diffusers in the present study (Nordin et al., 2012a, 2012b). As previous studies have proven the effects of varying inlet Reynolds number on turning diffuser performance in various applications (Djebedjian, 2001; Gopaliya & Chaudhary, 2010; Moonen et al., 2006), the present study on turning diffusers was operated within inlet Reynolds number range of 4.570E+04 to 1.122E+05, suitable for low subsonic wind tunnel and HVAC duct system applications.
Figure 1.4: Design of the two-dimensional turning diffuser (Nordin et al., 2012)

Figure 1.5: Design of the three-dimensional turning diffuser (Nordin et al., 2012)
1.2 Problem Statement

Secondary flow (flow separation) occurs mostly in diffuser applications. In circulating fluidized bed riser for example, flow separation leads to recirculation of gas and solids in the diffusers and consequently increases reflux ratio (Schut et al., 2000). In a dump diffuser, a recirculating vortex at the upper corner forms due to flow separation occurring at the outer wall (Ghose et al., 2013). Flow separation and reattachment in engineering situations are believed to contribute to pressure fluctuations, noise and also flow unsteadiness (Park & Sung, 1995).

Flow separation in a diffuser itself is unavoidable due to an adverse pressure gradient in diffuser flow (El-Askary & Nasr, 2009; Moonen et al., 2006; Wang et al., 2009). One of the approaches to reduce such losses in a diffuser is by installing guide vanes (baffles). In a closed loop wind tunnel, for example, in the upstream test section as shown in Figure 1.3 in Calautit et al. (2014), the diffuser was equipped with splitting plates and a 90° bend was installed with guide vanes. These are the approaches taken to reduce flow separation in both parts which can reduce the overall performance of the wind tunnel significantly.

For the two-dimensional 90° turning diffuser with AR=2.16, the approach for installing the baffle has been successfully investigated in a previous study (Noh@seth et al., 2013). By measuring the overall performance of the turning diffuser in terms of pressure recovery, \( C_p \), and flow uniformity, \( \sigma_{\text{out}} \), introduction of three units of flat plate baffles improved the overall performance by 50%. The present study will continue this effort by implementing a numerical approach to the same design of turning diffuser with baffle.

On the other hand, the three-dimensional turning diffuser can be more suitable for certain applications, even though it is proven to have a more complex and distorted flow as compared to the two-dimensional turning diffuser, which was highlighted in a previous study on the three-dimensional 90° turning diffuser with AR=2.16 conducted by Nordin et al. (2014a). The present study will continue the approach by installing the baffle to reduce flow separation, as well as improve the performance of the three-dimensional turning diffuser in terms of pressure recovery and flow uniformity.
It is essential to propose the optimum design of baffle for the three-dimensional turning diffuser application in order to improve its performance in terms of both pressure recovery and flow uniformity. Reducing flow separation will simultaneously reduce pressure fluctuations, noise and flow unsteadiness, as mentioned earlier, especially at the upstream section of turning diffuser in the application of closed loop low speed wind tunnel and HVAC duct system.

1.3 Objectives of study

The objectives of this study are:

1. To investigate the mechanism of flow structure in turning diffuser installed with baffle and studies the effects towards turning diffuser performance.

2. To propose an optimal design of baffles and evaluate the effectiveness of the new baffle design to improve turning diffuser performance.

1.4 Scope of study

The scope of this study covers:

1. A three-dimensional 90° turning diffuser with inlet dimension 13 cm × 5 cm and outlet dimension 19.5 cm × 7.2 cm, giving the area ratio of AR=2.16. The preliminary airfoil installed was optimized Wortmann FX60-100 taken from previous study (Sahlin et al., 1991).

2. The turning diffuser is preceded by a settling chamber and multiple screens, a contraction cone and long duct upstream, which adheres to the turbulent hydrodynamic length, to provide a fully developed flow at the turning diffuser inlet.
3. Inlet operating parameters, \( Re_{in} \) varied within the range of 4.527E+04 (10 m/s) – 1.263E+05 (28 m/s).

4. The turning diffuser performance is evaluated in terms of pressure recovery coefficient \( (C_p) \), which is measured through pressure tapping, and flow uniformity \( (\sigma_{out}) \), which is measured using Particle Image Velocimetry (PIV).

5. Simulations are done on both the two-dimensional and three-dimensional turning diffusers by using ANSYS Fluent. The K-Epsilon turbulence model and boundary conditions were verified and validated using experimental results.

6. The parametric study on the baffle design includes changes on; type of baffle between flat plate and airfoil, angle of attack, AOA ranging from 23° to 11°, thickness-to-chord ratio, \( t/c \) ranging from 5.35% to 13.27%, camber-to-chord ratio, \( f/c \) ranging from 7% to 13% and chord length, \( c \) ranging from 5 cm to 9 cm. Simulations on 23 designs of baffle include the performance comparison in terms of drag coefficient, \( C_d \) and airfoil pressure coefficient, \( C_{pa} \) profile.

1.5 Significance of study

The three-dimensional turning diffuser offers advantages in both applicability and compactness, especially in the HVAC duct system since many large buildings opt for centralized HVAC, which involves installation of the HVAC duct system. The current study focuses on designing new turning baffles to improve the performance of both the two-dimensional and three-dimensional turning diffuser with various inlet conditions. A number of previously available baffle designs are studied and evaluated in the attempt to propose a brand new baffle design with advantageous characteristics. Both experimental and numerical approaches are implemented for this purpose. An optimized baffle design will satisfy the need to achieve high-pressure recovery with less distortion of the outlet flow condition.
1.6 Thesis outline

The remainder of this thesis consists of another 8 chapters.

Chapter 2 presents a review on available literature to date referred to involving diffuser applications and theoretical background as well as development of experimental rig used in present study. Documentations on previous design of baffle which uses similar experimental setup was reviewed in order to propose preliminary airfoil to be installed in three-dimensional turning diffuser. Since present work involve both experimental and numerical approach, instrumentations on PIV sensors and techniques together with turbulence model used to study flow parameters were also reviewed.

Both Chapter 3 and Chapter 4 explain method and tools used in both experimental and numerical approach respectively. Chapter 3 starts with explanations on the overall experimental setup and later segregate each instrument in details. PIV measurement and instrumentation techniques were also discussed in details.

Chapter 4 continues with discussion on CFD modelling techniques which include mathematical model, computational domain, meshing, boundary conditions, solver algorithm and convergence criteria. Summarize input in ANSYS Fluent was also included in this chapter.

Chapter 5 laid out the experimental results from PIV, ranging from pressure recovery measurements data, flow structure from 2D and 3D PIV setup as well as measurement of turning diffuser outlet flow uniformity and efficiency. All data were compared to three-dimensional turning diffuser without baffle taken from previous study.

Chapter 6 continues with numerical results on velocity contour and flow structure for both two-dimensional turning diffuser and three-dimensional turning diffuser taking from ANSYS Fluent.

Chapter 7 focuses on validation and verification of the numerical analysis by using experimental data from Chapter 5. Verification of each CFD building block for both two-dimensional turning diffuser and three-dimensional turning diffuser numerical
analysis were discussed and the numerical results were validated with experimental results presented in Chapter 5.

Chapter 8 presents parametric study conducted on optimized Wortmann FX60-100 airfoil design including changes on design of baffle, AOA, $t/c$, $f/c$ and chord length, $c$. After all parametric study conducted, the optimum design of baffle was proposed in this chapter.

Conclusions are drawn on the present research and contributions towards research society were made. Recommendation on future work was also included in the end of this thesis.
CHAPTER 2

LITERATURE REVIEW

In order to better comprehend most aspects in studying flow in both the two-dimensional and three-dimensional turning diffuser with baffle, a review of some concept and theoretical background on the experimental setup and numerical approach is quite essential. Included in this chapter is review on the basic industrial application of diffuser followed by development of the experimental rig used in the present study. The present study focuses more on improving turning diffuser performance by installing baffles. Thus, a review on various baffle designs from previous studies, taken from different cases, is included in this chapter. As mentioned in the previous chapter, turning diffuser outlet flow uniformity, $\sigma_{out}$ was measured using PIV. Procedures on conducting the experiment using PIV by referring to other studies were also reviewed. Following the experimental procedures is a review on the numerical approach including validation and verification method conducted previously.
2.1 Diffuser applications and turning diffuser theoretical background

In general definition, diffusers are chambers that expand in flow direction, resulting in the decrease of fluid velocities along with increase of fluid pressure (Cermak, 1981). Industrial application, which uses the diffuser is either preceded by a bend or followed by a bend, includes a circulating fluidized bed riser, HVAC duct system as well as closed loop wind tunnel. Schematic experimental diagram of the circulating fluidized bed riser conducted by Schut et al.(2000) as shown in Figure 1.1 in the previous chapter is a clear example of diffuser application in duct system. The location of diffuser within the riser was varied as shown in Figure 2.1 and the effects on reflux ratio concludes that diffuser located 1050 cm below the exit provide better reflux ratio. Reflux ratio in parallel duct is higher when distance below the exit increases. Thus, proposing the use of turning diffuser in this case is rather inappropriate.

Figure 2.1: Two different positions of diffuser in the riser; (a) 550 cm below the exit and (b) 1050 cm below the exit (Schut et al., 2000)
In HVAC ductwork, free-discharge diffuser preceded by a bend was installed at the duct outlet to reduce the air velocity when discharge to atmosphere as part of room air distribution system. Gan and Riffat (1996) concluded in their study that a divergence angle smaller than 10º of pyramidal diffuser should be used to achieve flow regularity and stability discharged air, with the exception of spacer length of twice the hydraulic diameter, $D_h \times (2D_h)$ should be introduced. El-Askary & Nasr (2009) concluded the same issue, where spacer length should be introduced between bend and diffuser which will contribute to loss reduction of the system. However, as shown in Figure 2.2, a highly distorted flow was still recorded. The turning diffuser could be proposed, together with installation of baffle to improve such flaws.

![Figure 2.2: Bend-diffuser combination with short spacer shows highly distorted flow at diffuser exit for both studies by; (a) Gan & Riffat (1996), (b) El-Askary & Nasr (2009)](image)

Diffusers are also commonly used in the wind tunnel system. Studies on subsonic close loop wind tunnel installed with principle components including the contraction cone, test section and diffuser has been conducted previously (Calautit et al., 2014; Gordon & Imbabi, 1998; Moonen, Blocken, & Carmeliet, 2007; Moonen et al., 2006) According to Moonen et al. (2006), flow separation will occur in several sections; entrance and exit of test section, 180º turn and sudden change in cross-sectional area. For the wind tunnel, the main aerodynamic objective is to make sure the flow is steady throughout the test section and has uniform speed (Calautit et al.,
A closed loop wind tunnel has four 90° turn as shown in Figure 1.3 in previous chapter.

For the 90° upstream turn (Section 4), guide vanes were installed to reduce flow separation, whereas for 90° lower upstream turn (Section 8), guide vanes were mounted to direct the flow to be parallel to test section centre line. At the same time, it helped improved flow uniformity just before entering contraction cone. Both 90° turn downstream and upstream of the diffuser’s outlet were also installed with guide vanes, with the same objective to reduce flow separation occurring in the turn (Calautit et al., 2014). As shown in Figure 2.3, significant improvement on velocity contour in closed loop wind tunnel concluded that guide vanes installed in diffuser and 90° turn helps reduce flow separation and improve flow uniformity entering the test section.

However, when space limitations were the factor to be considered in building a closed loop wind tunnel, diffusing and turning activities could be combined as a turning diffuser. Other terms for turning diffuser used in previous studies were expanding corner, diffusing bend and curved diffuser. Studies on turning diffusers were previously conducted, and they highlighted a few subjects to be brought up for discussion (Chong et al., 2008; Djebedjian, 2001; McMillan, 1982; Majumdar et al., 1996, 1998, 1999; Sinha et al., 2010, 2011, 2012).

Flow structure in a curved diffuser depends greatly on centreline length to inlet width ratio ($L_m/W_l$), area ratio (AR), inlet condition ($Re_m$) and turning angle ($\Delta \phi$). Furthermore, higher shear strains near convex curved wall flow structure were initiated when higher inlet Reynolds numbers were introduced (Djebedjian, 2001). According to Chong et al. (2008), centrifugal forces were introduced in curved ducts, which cause deflected core flow to the outer wall and consequently due to adverse pressure gradient reduce the velocity at the outer wall.
Figure 2.3: (a) Contours of velocity magnitude for wind tunnel before and (b) after installation of guide vanes (Calautit et al., 2014)

Majumdar et al. (1996, 1998, and 1999) in all their studies experimentally investigated 90° curved diffuser as well as 180° curved diffuser flow characteristics. Severe flow distortion was observed due to centrifugal force created by the curvature wall. Efforts were done to improve flow characteristic in the curved diffuser was by installing vanes. Other than 90° and 180° curved diffuser, small divergence angle curved diffuser such as 30°, 37.5° and 42° curved annular diffuser were previously studied (Sinha et al., 2010, 2011 and 2012) all resulting in high velocity flow
accumulated and shifted towards the outer (concave) wall especially at the outlet of curved diffuser.

All these studies highlighted critical flow separation due to curvature effects as well as diffusing activities in curved diffuser. Secondary flow cannot be neglected since it contributes to losses in the system. Efforts can be done in improving flow characteristics in curved diffuser, since it offers wide industrial applications especially in restricted space cases. Next section will outline a review on previous research focusing on the two-dimensional and three-dimensional rectangular cross section turning diffuser together with development of experimental rig used in present study.

2.2 Experimental rig development on low subsonic wind tunnel feature

Nordin et al. (2011) started research on performance of a bend-diffuser with baffles installed which was measured and compared to a bend-diffuser without baffles. The tested diffuser has an area ratio (AR) of 7.2 with 13 cm × 13 cm square inlet and axial length of 49 cm. Three locations were chosen to be measured, i.e., before bend (S1), before diffuser (S2) and after diffuser (S3) with two planes (A and B) each using Pitot static probe and digital manometer with accuracy of ±0.1Pa. Macbain’s (MacBain, 2003) patent was selected as baffle design to be adopted in the experiment. Details are shown in Figure 2.4 and Figure 2.5. It was proven that with the installation of baffles in bend-diffuser system, the overall performance improved in terms of pressure loss reduction. As shown in Table 2.1, loss coefficient (K) was reduced for almost all cases except inside the diffuser. This is due to excessive separation in the diffuser itself.

After seeing a promising improvement in overall losses for bend-diffuser, Nordin et al. (2012a) proceed by adopting a turning diffuser in replace of bend-diffuser. A numerical approach was conducted by varying turning diffuser geometric conditions (AR=1.6, 2.0 and 3.0) and operating parameters (Re_m ranging from 23 to 2.123E+05). Simulations on each case were conducted using 3 different turbulence
models, which was Standard K-Epsilon turbulence model (SKE), the Shear Stress Transport model (SST K-Omega) and the Reynolds Stress Model (RSM).

Figure 2.4: Location chosen for measurement; S1, S2, S3 (Nordin et al., 2011)

Figure 2.5: Two planes selected for each location measurement (Nordin et al., 2011)
Table 2.1: Pressure loss coefficient \((K)\) (Nordin et al., 2011)

<table>
<thead>
<tr>
<th>Part</th>
<th>Loss Coefficient ((K)) Without baffles</th>
<th>With Baffles</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bend (a)</td>
<td>1.249</td>
<td>0.227</td>
</tr>
<tr>
<td>Bend (b)</td>
<td>1.145</td>
<td>-0.351</td>
</tr>
<tr>
<td>Diffuser (a)</td>
<td>1.290</td>
<td>2.899</td>
</tr>
<tr>
<td>Diffuser (b)</td>
<td>0.578</td>
<td>1.275</td>
</tr>
<tr>
<td>System (a)</td>
<td>1.306</td>
<td>0.573</td>
</tr>
<tr>
<td>System (b)</td>
<td>1.746</td>
<td>-0.134</td>
</tr>
</tbody>
</table>

Turning diffuser performances were measured in terms of pressure recovery \((C_p)\) and outlet’s flow uniformity \((\sigma_{out})\). Higher value of \(C_p\) represents high pressure recovery, whereas lower value of \(\sigma_{out}\) represents high flow uniformity. From the simulation at specific \(Re_{in}\), pressure recovery increases with increasing AR. Conversely, flow uniformity decreases with increasing \(Re_{in}\). On the other hand, at specific AR, while pressure recovery increase with increasing \(Re_{in}\), flow uniformity decrease with increasing \(Re_{in}\). After all, the increase of AR yields smaller effects on the flow uniformity as compared to the effects by increasing \(Re_{in}\).

Thus, Nordin et al. (2012a) carried out more intensive studies on varying \(Re_{in}\) to find its effects on the flow uniformity. Consequently, an optimum geometric configuration of turning diffuser was proposed; AR=1.6 running at \(Re_{in}=2.653E+04\), which corresponded to performance value of \(C_p=0.320\) and \(\sigma_{out}=1.620\). However, results between simulation and experimental data deviates up to 34.1%, concluding that further improvement on the existing rig need to be implemented.

Nordin et al. (2013) then developed a low subsonic wind tunnel for turning diffuser application to ensure flow at the inlet of turning diffuser need to be steady, uniform and fully developed. Even sufficient hydrodynamic entrance length was introduced, poor joining of duct and abrupt change of cross sectional area between the blower and the duct might be the cause of 34.1% deviation between the numerical and experimental results in previous study (Nordin et al., 2012a). Several improvements were done to the system as shown in Figure 2.6.
To develop steady flow, a centrifugal blower with 3-phase inverter controller was used. Settling chamber and multiple screens made of metal wire interwoven were installed to improve the mean flow uniformity and reduce oncoming turbulence. The contraction cone will help to accelerate flow from the settling chamber, and it is expected to have steady, uniform and free separation out-going flow. Hydrodynamic length was introduced earlier on before connected to the turning diffuser’s inlet. Thus, at the turning diffuser’s inlet, the flow is believed to be steady, uniform and fully developed.

Nordin et al. (2014b) then verified the fully developed flow entering turning diffuser using Pitot static probe at 5 different points. Flow entering the turning diffuser was proved to be fully developed based on the velocity profile which resembles the boundary layer of a turbulent fully developed flow as shown in Figure 2.7. The outlet local velocity was also obtained using Particle Image Velocimetry (PIV) at 5 different points. Small average differences of 0.8%-1.2% between PIV result and Pitot static probe offered promising PIV measurement and rig implementation.
Figure 2.7: (a) Five points location at inlet for measurements of fully developed flow and (b) velocity profile measured using Pitot static probe (Nordin et al., 2014b)

After a strong verification of the rig and the whole system, Nordin et al. (2014a) continues the experimental investigation on two-dimensional turning diffuser by varying inflow Reynolds number. Pressure recovery was measured using pressure tapping at both inlet and outlet of the turning diffuser connected via triple-T piezometer and measured using a digital Manometer, whereas the outlet flow uniformity was measured using PIV.

5 different values of outlet flow velocity were measured. Verification of PIV result was obtained by comparing manual measurement of the local outlet velocity, $V_o$ using Pitot static probe with PIV measurement. Table 2.2 shows the deviation between both approaches. These outputs were used by Noh@Seth et al. (2013) as reference in the study of improving flow uniformity and pressure recovery of the two-dimensional turning diffuser by means of installing baffles.

Table 2.2: $C_p$ measured for each $Re_{in}$ tested and verification of PIV results for two-dimensional turning diffuser (Nordin et al., 2014a)

<table>
<thead>
<tr>
<th>$Re_{in}$</th>
<th>$C_p$</th>
<th>$V_o$ Pitot</th>
<th>$V_o$ PIV</th>
<th>Deviation (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.786E+04</td>
<td>0.191</td>
<td>4.98</td>
<td>4.92</td>
<td>1.2</td>
</tr>
<tr>
<td>6.382E+04</td>
<td>0.209</td>
<td>5.92</td>
<td>5.87</td>
<td>0.8</td>
</tr>
<tr>
<td>1.027E+05</td>
<td>0.216</td>
<td>11.05</td>
<td>10.64</td>
<td>3.7</td>
</tr>
<tr>
<td>1.397E+05</td>
<td>0.221</td>
<td>15.45</td>
<td>15.34</td>
<td>0.7</td>
</tr>
<tr>
<td>1.775E+05</td>
<td>0.239</td>
<td>19.75</td>
<td>19.05</td>
<td>3.5</td>
</tr>
</tbody>
</table>
Noh@Seth et al. (2013) continued the effort to improve flow uniformity and pressure recovery by installing flat plate baffles. 3 units of flat plate baffles were designed, acting as a small turning diffuser in the existing turning diffuser in order to avoid flow abruption. Figure 2.8 shows the design of flat plate baffles in two-dimensional turning diffuser.

Using the same experimental rig, an improvement of 54.6% on pressure recovery was proven after installing the two-dimensional turning diffuser with baffles. Best produced pressure recovery of $C_p=0.526$ was recorded as compared to $C_p=0.239$ for two-dimensional turning diffuser without baffle at the highest Reynolds number tested. As for the flow uniformity, the best $\sigma_{out}$ was $\sigma_{out}=3.235$ at the highest Reynolds number tested, with an improvement of 47.1%. Table 2.3 shows the resulting output from the experiment done by Noh@Seth et al. (2013).

![Figure 2.8: Flat plate baffle designed by Noh@Seth et al. (2013). All dimensions in cm.](image-url)
<table>
<thead>
<tr>
<th>$Re_{in}$</th>
<th>$\sigma_{out}$ (Noh@seth et al., 2013)</th>
<th>$\sigma_{out}$ (Nordin et al., 2014a)</th>
<th>Improvement (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.786E+04</td>
<td>0.719</td>
<td>1.755</td>
<td>58.864</td>
</tr>
<tr>
<td>6.382E+04</td>
<td>0.683</td>
<td>1.852</td>
<td>63.032</td>
</tr>
<tr>
<td>1.027E+05</td>
<td>2.437</td>
<td>2.910</td>
<td>16.240</td>
</tr>
<tr>
<td>1.397E+05</td>
<td>2.621</td>
<td>4.947</td>
<td>46.492</td>
</tr>
<tr>
<td>1.775E+05</td>
<td>3.235</td>
<td>6.128</td>
<td>47.127</td>
</tr>
</tbody>
</table>

Other than flow uniformity ($\sigma_{out}$), velocity contour at the outlet produced by PIV was also compared. Noh@Seth et al. (2013) successfully improve and direct the deflected flow more towards the inner wall as compared to Nordin et al. (2014a). For reference, Figure 2.9 shows velocity contour with flow vector comparison for the highest $Re_{in}$ tested in both experiments. In other words, flow separation at the inner wall region has been successfully reduced by installing baffle which correlate with smaller value of $\sigma_{out}$ measured.
Study on turning diffuser can be widely enhanced to various dimensions of turning diffuser. Since the two-dimensional turning diffuser offered extensive improvement on replacing bend-diffuser, especially with the installation of baffle, Nordin et al. (2012b) extended their studies by varying the area ratios of the three-dimensional turning diffuser. Generally, the three-dimensional turning diffuser has more complex flow as compared to the two-dimensional turning diffuser as prescribed in previous chapter; hence offer wider discussion on flow characteristics and turning diffuser performance.

Nordin et al. (2012b) investigated three different cases; the two-dimensional turning diffuser (Case A), three-dimensional turning diffuser with AR=2.0 (Case B)
and three-dimensional turning diffuser with AR=4.0 (Case C). All cases were compared and concluded that pressure recovery and flow uniformity for Case B is lower than Case A, due to more complex flow and diffusing activities for three-dimensional turning diffuser.

Latest research done by Nordin et al. (2014a) was on the performance of the three-dimensional turning diffuser at various inlet conditions as compared to the two-dimensional turning diffuser by using similar experimental setup and rig. Table 2.4 and 2.5 shows the comparison of both experiments. The research proposed for inflow $Re_{in}=1.027E+05-1.775E+05$, the three-dimensional turning diffuser is more reliable and as for $Re_{in}=5.786E+04-6.382E+04$, the two-dimensional turning diffuser is much more favourable. This is only if flow uniformity is of interest to subject. On the other hand, if pressure recovery is becoming the concern, the three-dimensional turning diffuser performed better within $Re_{in}=5.786E+04-6.382E+04$ and $Re_{in}=1.027E+05-1.775E+05$ for the two-dimensional turning diffuser.

Table 2.4: Mean outlet velocity, $V_{out}$ and flow uniformity comparison, $\sigma_{out}$ (Nordin et al., 2014a)

<table>
<thead>
<tr>
<th>$Re_{in}$</th>
<th>2-D Turning Diffuser</th>
<th>3-D Turning Diffuser</th>
</tr>
</thead>
<tbody>
<tr>
<td>$V_{out}$ (m/s)</td>
<td>$\sigma_{out}$ (m/s)</td>
<td>$V_{out}$ (m/s)</td>
</tr>
<tr>
<td>5.786E+04</td>
<td>1.57</td>
<td>1.75</td>
</tr>
<tr>
<td>6.382E+04</td>
<td>1.61</td>
<td>1.85</td>
</tr>
<tr>
<td>1.027E+05</td>
<td>2.31</td>
<td>2.91</td>
</tr>
<tr>
<td>1.397E+05</td>
<td>4.85</td>
<td>4.90</td>
</tr>
<tr>
<td>1.775E+05</td>
<td>5.75</td>
<td>6.12</td>
</tr>
</tbody>
</table>

Table 2.5: Pressure recovery, $C_p$ comparison (Nordin et al., 2014a)

<table>
<thead>
<tr>
<th>$Re_{in}$</th>
<th>2-D Turning Diffuser</th>
<th>3-D Turning Diffuser</th>
</tr>
</thead>
<tbody>
<tr>
<td>$C_p$</td>
<td>$C_p$</td>
<td></td>
</tr>
<tr>
<td>5.786E+04</td>
<td>0.191</td>
<td>0.210</td>
</tr>
<tr>
<td>6.382E+04</td>
<td>0.209</td>
<td>0.217</td>
</tr>
<tr>
<td>1.027E+05</td>
<td>0.216</td>
<td>0.203</td>
</tr>
<tr>
<td>1.397E+05</td>
<td>0.221</td>
<td>0.219</td>
</tr>
<tr>
<td>1.775E+05</td>
<td>0.239</td>
<td>0.194</td>
</tr>
</tbody>
</table>
REFERENCES


