Use of experimental data to generate swirl velocity in the FLUENT Fan Model.

Ligong Yang
University of Windsor

Follow this and additional works at: https://scholar.uwindsor.ca/etd

Recommended Citation
https://scholar.uwindsor.ca/etd/4473

This online database contains the full-text of PhD dissertations and Masters' theses of University of Windsor students from 1954 forward. These documents are made available for personal study and research purposes only, in accordance with the Canadian Copyright Act and the Creative Commons license—CC BY-NC-ND (Attribution, Non-Commercial, No Derivative Works). Under this license, works must always be attributed to the copyright holder (original author), cannot be used for any commercial purposes, and may not be altered. Any other use would require the permission of the copyright holder. Students may inquire about withdrawing their dissertation and/or thesis from this database. For additional inquiries, please contact the repository administrator via email (scholarship@uwindsor.ca) or by telephone at 519-253-3000 ext. 3208.
INFORMATION TO USERS

This manuscript has been reproduced from the microfilm master. UMI films the text directly from the original or copy submitted. Thus, some thesis and dissertation copies are in typewriter face, while others may be from any type of computer printer.

The quality of this reproduction is dependent upon the quality of the copy submitted. Broken or indistinct print, colored or poor quality illustrations and photographs, print bleedthrough, substandard margins, and improper alignment can adversely affect reproduction.

In the unlikely event that the author did not send UMI a complete manuscript and there are missing pages, these will be noted. Also, if unauthorized copyright material had to be removed, a note will indicate the deletion.

Oversize materials (e.g., maps, drawings, charts) are reproduced by sectioning the original, beginning at the upper left-hand corner and continuing from left to right in equal sections with small overlaps.

ProQuest Information and Learning
300 North Zeeb Road, Ann Arbor, MI 48106-1346 USA
800-521-0600

UMI
USE OF EXPERIMENTAL DATA TO GENERATE SWIRL VELOCITY IN THE FLUENT FAN MODEL

By

Ligong Yang

A Thesis

Submitted to the Faculty of Graduate Studies and Research through Mechanical, Automotive and Materials Engineering in Partial Fulfillment of the Requirements for the Degree of Master of Applied Science at the University of Windsor

Windsor, Ontario, Canada
2002
The author has granted a non-exclusive licence allowing the National Library of Canada to reproduce, loan, distribute or sell copies of this thesis in microform, paper or electronic formats.

L’auteur a accordé une licence non exclusive permettant à la Bibliothèque nationale du Canada de reproduire, prêter, distribuer ou vendre des copies de cette thèse sous la forme de microfiche/film, de reproduction sur papier ou sur format électronique.

The author retains ownership of the copyright in this thesis. Neither the thesis nor substantial extracts from it may be printed or otherwise reproduced without the author’s permission.

L’auteur conserve la propriété du droit d’auteur qui protège cette thèse. Ni la thèse ni des extraits substantiels de celle-ci ne doivent être imprimés ou autrement reproduits sans son autorisation.
ABSTRACT

The University of Windsor/DaimlerChrysler Fan Test Facility was used to measure the pressure rise of the fan and the detailed velocity field downstream of the fan. These measurements were made for six different conditions, comprised of two fan rotating speeds and three flow rates. At each condition, pressure rise and three components of velocity were measured on two downstream planes, at 25 mm and 100 mm below the base of the hub.

In order to numerically simulate the wake of an automotive cooling fan, the FLUENT Fan Model was adopted as a boundary condition in the simulation domain. The parameters required by the fan model were generated from the experimental data. The experimental pressure rise and corresponding flow rates were used to derive the relation between the pressure rise and the average fluid velocity magnitude normal to the fan. This polynomial relation was applied in the fan model. In order to develop the relation for the radial and tangential velocity components as function of radial distance, needed in the fan model to simulate the swirl, the measurements taken on 25 mm planes were assumed to be taken at the bottom of the fan blade. Thus at a specific flow condition, the radial and tangential velocity polynomials, to be applied in the fan model, were generated from the experimental radial and tangential velocity data on the 25 mm plane.

Two types of simulations were carried out, one using only the pressure rise fan boundary condition and the other using the pressure rise and swirl velocity fan boundary conditions. The effect of applying the radial and tangential velocity boundary conditions has been investigated. Comparisons between the computational pressure rise and the experimental pressure rise was made. Comparisons were also made between the CFD results and the experimental data for the velocity components on the 100 mm downstream plane.
To my parents
ACKNOWLEDGEMENTS

The author would like to give his deep and sincere gratitude to Dr. R. M. Barron for his guidance, unceasing help and generous aid during this study.

Thanks also go to Dr. G. W. Rankin for his valuable opinions and continuous support. The author wishes to express his acknowledgements to Mr. J. Billington of DaimlerChrysler, for arranging access to the computer facilities at the University of Windsor/DaimlerChrysler Automotive Research and Development Centre and for reading the thesis. Mr. P. Seguin's assistance with the electronic equipment used in this study is greatly appreciated. Thanks are also due to Dr. O. P. Chandna for evaluating this study as a committee member.

Finally, the author would like to acknowledge DaimlerChrysler Canada Inc., Natural Science and Engineering Research Council and the University of Windsor for their financial support.
# TABLE OF CONTENTS

ABSTRACT .................................................................................................................. iv

ACKNOWLEDGEMENTS ............................................................................................... vi

TABLE OF CONTENTS ................................................................................................. vii

LIST OF TABLES ........................................................................................................... ix

LIST OF FIGURES ......................................................................................................... x

NOMENCLATURE .......................................................................................................... xvi

CHAPTER 1  INTRODUCTION ........................................................................................ 1

CHAPTER 2  LITERATURE REVIEW .............................................................................. 3

2.1 Fan Experiments .................................................................................................... 3
2.2 Fan CFD Simulation ............................................................................................... 4
2.3 Cooling Module Models ......................................................................................... 6
2.4 Summary ................................................................................................................ 7

CHAPTER 3  EXPERIMENTAL SIMULATIONS ............................................................. 8

3.1 Experimental Facility ............................................................................................. 8
3.2 Experimental Conditions ....................................................................................... 9
3.3 Outline of Experimental Procedures .................................................................. 10
3.4 Velocity Contours and Pressure Rise ................................................................. 10

CHAPTER 4  CFD SIMULATIONS ............................................................................... 39

4.1 Actuator Disc Theory ............................................................................................ 39
4.2 FLUENT Fan Model .............................................................................................. 39
4.3 Producing the Polynomials of the Fan Model .................................................... 41
   4.3.1 Pressure Rise ................................................................................................. 41
   4.3.2 Velocity Components ................................................................................. 43
4.4 The Computational Domain .................................................................................. 45
CHAPTER 5 RESULTS AND DISCUSSION

5.1 Experimental Results and Analysis
5.1.1 Axial Velocity
5.1.2 Radial Velocity
5.1.3 Tangential Velocity
5.1.4 Velocity Magnitude

5.2 CFD Results and Analysis
5.2.1 Pressure Rise
5.2.2 Axial Velocity
5.2.3 Radial Velocity
5.2.4 Tangential Velocity
5.2.5 Velocity Magnitude
5.2.6 Average Axial Velocity and Velocity Magnitude

CHAPTER 6 CONCLUSIONS AND RECOMMENDATIONS

6.1 Conclusions
6.2 Recommendations for Future Work

REFERENCES

APPENDIX A RESIDUE COMPARISON

VITA AUCTORIS
LIST OF TABLES

Chapter 3
Table 3.1 Fan Test Experiment Conditions .................................................. 9
Table 3.2 Pressure Rise vs. Flow Rate at 2300 rpm .................................. 11
Table 3.3 Pressure Rise vs. Flow Rate at 2700 rpm .................................. 12

Chapter 4
Table 4.1 Pressure Rise vs. Velocity Magnitude Normal to the Fan at 2300 rpm..... 42
Table 4.2 Pressure Rise vs. Velocity Magnitude Normal to the Fan at 2700 rpm..... 42
Table 4.3 Circumferential Averaged Radial Velocity Curve Fit Polynomial

Coefficients (Data from 25 mm Downstream Plane) ..................................... 44
Table 4.4 Circumferential Averaged Tangential Velocity Curve Fit Polynomial

Coefficients (Data from 25 mm Downstream Plane) ..................................... 45
Table 4.5 Inlet Velocities .................................................................................. 46

Chapter 5
Table 5.1 Experimental Axial Velocity .......................................................... 58
Table 5.2 Experimental Radial Velocity ......................................................... 59
Table 5.3 Experimental Tangential Velocity ................................................... 60
Table 5.4 Experimental Velocity Magnitude .................................................. 61
Table 5.5 Iteration Count .............................................................................. 62
Table 5.6 Comparison of Pressure Rise ......................................................... 63
Table 5.7 Average Axial Velocity & Velocity Magnitude ................................ 65

Appendix A
Table A.1 Sum of Squares of Residues for Radial Velocity Polynomial Curve Fit..... 83
Table A.2 Sum of Squares of Residues for Tangential Velocity Polynomial Curve Fit
...................................................................................................................... 83
LIST OF FIGURES

Chapter 3

Figure 3.1  Perspective View of Fan Test Facility ........................................ 13
Figure 3.2  Cutaway View of Fan Test Facility ........................................ 13
Figure 3.3  Automotive Cooling Fan .......................................................... 14
Figure 3.4  Measurement Grid ..................................................................... 14
Figure 3.5  Contours of Axial Velocity at 2300 rpm, 635 cfm, 25 mm Downstream .. 15
Figure 3.6  Contours of Axial Velocity at 2300 rpm, 635 cfm, 100 mm Downstream 15
Figure 3.7  Contours of Radial Velocity at 2300 rpm, 635 cfm, 25 mm Downstream .. 16
Figure 3.8  Contours of Radial Velocity at 2300 rpm, 635 cfm, 100 mm Downstream 16
Figure 3.9  Contours of Tangential Velocity at 2300 rpm, 635 cfm, 25 mm Downstream ................................................................. 17
Figure 3.10  Contours of Tangential Velocity at 2300 rpm, 635 cfm, 100 mm Downstream ........................................................................ 17
Figure 3.11  Contours of Axial Velocity at 2300 rpm, 800 cfm, 25 mm Downstream .... 18
Figure 3.12  Contours of Axial Velocity at 2300 rpm, 800 cfm, 100 mm Downstream 18
Figure 3.13  Contours of Radial Velocity at 2300 rpm, 800 cfm, 25 mm Downstream .. 19
Figure 3.14  Contours of Radial Velocity at 2300 rpm, 800 cfm, 100 mm Downstream ................................................................. 19
Figure 3.15  Contours of Tangential Velocity at 2300 rpm, 800 cfm, 25 mm Downstream ........................................................................ 20
Figure 3.16  Contours of Tangential Velocity at 2300 rpm, 800 cfm, 100 mm Downstream ................................................................. 20
Figure 3.17  Contours of Axial Velocity at 2300 rpm, 1123 cfm, 25 mm Downstream 21
Figure 3.18  Contours of Axial Velocity at 2300 rpm, 1123 cfm, 100 mm Downstream ........................................................................ 21
Figure 3.19  Contours of Radial Velocity at 2300 rpm, 1123 cfm, 25 mm Downstream ........................................................................ 22
Figure 3.20  Contours of Radial Velocity at 2300 rpm, 1123 cfm, 100 mm Downstream ........................................................................ 22
Figure 3.21  Contours of Tangential Velocity at 2300 rpm, 1123 cfm, 25 mm
Downstream ................................................................. 23

Figure 3.22  Contours of Tangential Velocity at 2300 rpm, 1123 cfm, 100 mm
Downstream ................................................................. 23

Figure 3.23  Contours of Axial Velocity at 2700 rpm, 737 cfm, 25 mm Downstream . 24
Figure 3.24  Contours of Axial Velocity at 2700 rpm, 737 cfm, 100 mm Downstream 24
Figure 3.25  Contours of Radial Velocity at 2700 rpm, 737 cfm, 25 mm Downstream 25
Figure 3.26  Contours of Radial Velocity at 2700 rpm, 737 cfm, 100 mm Downstream ................................................................. 25

Figure 3.27  Contours of Tangential Velocity at 2700 rpm, 737 cfm, 25 mm
Downstream ................................................................. 26

Figure 3.28  Contours of Tangential Velocity at 2700 rpm, 737 cfm, 100 mm
Downstream ................................................................. 26

Figure 3.29  Contours of Axial Velocity at 2700 rpm, 800 cfm, 25 mm Downstream . 27
Figure 3.30  Contours of Axial Velocity at 2700 rpm, 800 cfm, 100 mm Downstream 27
Figure 3.31  Contours of Radial Velocity at 2700 rpm, 800 cfm, 25 mm Downstream 28
Figure 3.32  Contours of Radial Velocity at 2700 rpm, 800 cfm, 100 mm Downstream ................................................................. 28

Figure 3.33  Contours of Tangential Velocity at 2700 rpm, 800 cfm, 25 mm
Downstream ................................................................. 29

Figure 3.34  Contours of Tangential Velocity at 2700 rpm, 800 cfm, 100 mm
Downstream ................................................................. 29

Figure 3.35  Contours of Axial Velocity at 2700 rpm, 1327 cfm, 25 mm Downstream30
Figure 3.36  Contours of Axial Velocity at 2700 rpm, 1327 cfm, 100 mm Downstream ................................................................. 30

Figure 3.37  Contours of Radial Velocity at 2700 rpm, 1327 cfm, 25 mm Downstream ................................................................. 31

Figure 3.38  Contours of Radial Velocity at 2700 rpm, 1327 cfm, 100 mm Downstream ................................................................. 31

Figure 3.39  Contours of Tangential Velocity at 2700 rpm, 1327 cfm, 25 mm
Downstream ................................................................. 32
Figure 3.40  Contours of Tangential Velocity at 2700 rpm, 1327 cfm, 100 mm  
Downstream ................................................................................................................. 32

Figure 3.41  Contours of Velocity Magnitude at 2300 rpm, 635 cfm, 25 mm  
Downstream ................................................................................................................. 33

Figure 3.42  Contours of Velocity Magnitude at 2300 rpm, 635 cfm, 100 mm  
Downstream ................................................................................................................. 33

Figure 3.43  Contours of Velocity Magnitude at 2300 rpm, 800 cfm, 25 mm  
Downstream ................................................................................................................. 34

Figure 3.44  Contours of Velocity Magnitude at 2300 rpm, 800 cfm, 100 mm  
Downstream ................................................................................................................. 34

Figure 3.45  Contours of Velocity Magnitude at 2300 rpm, 1123 cfm, 25 mm  
Downstream ................................................................................................................. 35

Figure 3.46  Contours of Velocity Magnitude at 2300 rpm, 1123 cfm, 100 mm  
Downstream ................................................................................................................. 35

Figure 3.47  Contours of Velocity Magnitude at 2700 rpm, 737 cfm, 25 mm  
Downstream ................................................................................................................. 36

Figure 3.48  Contours of Velocity Magnitude at 2700 rpm, 737 cfm, 100 mm  
Downstream ................................................................................................................. 36

Figure 3.49  Contours of Velocity Magnitude at 2700 rpm, 800 cfm, 25 mm  
Downstream ................................................................................................................. 37

Figure 3.50  Contours of Velocity Magnitude at 2700 rpm, 800 cfm, 100 mm  
Downstream ................................................................................................................. 37

Figure 3.51  Contours of Velocity Magnitude at 2700 rpm, 1327 cfm, 25 mm  
Downstream ................................................................................................................. 38

Figure 3.52  Contours of Velocity Magnitude at 2700 rpm, 1327 cfm, 100 mm  
Downstream ................................................................................................................. 38

Chapter 4

Figure 4.1  Pressure Rise and Linear Curve Fit ................................................................. 47

Figure 4.2  Non-dimensional Pressure Rise and Linear Curve Fit ................................. 48
Figure 4.3  Radial Velocity Polynomial Curve Fit at 2300 rpm, 800 cfm, 25 mm
  Downstream (20 and 21 points) ................................................................................. 48
Figure 4.4  Tangential Velocity Polynomial Curve Fit at 2300 rpm, 800 cfm, 25 mm
  Downstream (20 and 21 points) ................................................................................. 49
Figure 4.5  Radial Velocity Polynomial Curve Fit at 2300 rpm, 635 cfm, 25 mm
  Downstream (21 and 23 points) ................................................................................. 49
Figure 4.6  Radial Velocity Polynomial Curve Fit at 2300 rpm, 635 cfm, 25 mm
  Downstream ................................................................................................................. 50
Figure 4.7  Tangential Velocity Polynomial Curve Fit at 2300 rpm, 635 cfm, 25 mm
  Downstream ................................................................................................................. 50
Figure 4.8  Radial Velocity Polynomial Curve Fit at 2300 rpm, 800 cfm, 25 mm
  Downstream ................................................................................................................. 51
Figure 4.9  Tangential Velocity Polynomial Curve Fit at 2300 rpm, 800 cfm, 25 mm
  Downstream ................................................................................................................. 51
Figure 4.10  Radial Velocity Polynomial Curve Fit at 2300 rpm, 1123 cfm, 25 mm
  Downstream ................................................................................................................. 52
Figure 4.11  Tangential Velocity Polynomial Curve Fit at 2300 rpm, 1123 cfm, 25 mm
  Downstream ................................................................................................................. 52
Figure 4.12  Radial Velocity Polynomial Curve Fit at 2700 rpm, 737 cfm, 25 mm
  Downstream ................................................................................................................. 53
Figure 4.13  Tangential Velocity Polynomial Curve Fit at 2700 rpm, 737 cfm, 25 mm
  Downstream ................................................................................................................. 53
Figure 4.14  Radial Velocity Polynomial Curve Fit at 2700 rpm, 800 cfm, 25 mm
  Downstream ................................................................................................................. 54
Figure 4.15  Tangential Velocity Polynomial Curve Fit at 2700 rpm, 800 cfm, 25 mm
  Downstream ................................................................................................................. 54
Figure 4.16  Radial Velocity Polynomial Curve Fit at 2700 rpm, 1327 cfm, 25 mm
  Downstream ................................................................................................................. 55
Figure 4.17  Tangential Velocity Polynomial Curve Fit at 2700 rpm, 1327 cfm, 25 mm
  Downstream ................................................................................................................. 55
Figure 4.18  Computational Domain ................................................................................ 56
Figure 5.14  Circumferential Averaged Radial Velocity at 2700 rpm, 737 cfm, 100 mm Downstream ................................................................. 72

Figure 5.15  Circumferential Averaged Tangential Velocity at 2700 rpm, 737 cfm, 100 mm Downstream ................................................................. 73

Figure 5.16  Circumferential Averaged Velocity Magnitude at 2700 rpm, 737 cfm, 100 mm Downstream ................................................................. 73

Figure 5.17  Circumferential Averaged Axial Velocity at 2700 rpm, 800 cfm, 100 mm Downstream ................................................................. 74

Figure 5.18  Circumferential Averaged Radial Velocity at 2700 rpm, 800 cfm, 100 mm Downstream ................................................................. 74

Figure 5.19  Circumferential Averaged Tangential Velocity at 2700 rpm, 800 cfm, 100 mm Downstream ................................................................. 75

Figure 5.20  Circumferential Averaged Velocity Magnitude at 2700 rpm, 800 cfm, 100 mm Downstream ................................................................. 75

Figure 5.21  Circumferential Averaged Axial Velocity at 2700 rpm, 1327 cfm, 100 mm Downstream ................................................................. 76

Figure 5.22  Circumferential Averaged Radial Velocity at 2700 rpm, 1327 cfm, 100 mm Downstream ................................................................. 76

Figure 5.23  Circumferential Averaged Tangential Velocity at 2700 rpm, 1327 cfm, 100 mm Downstream ................................................................. 77

Figure 5.24  Circumferential Averaged Velocity Magnitude at 2700 rpm, 1327 cfm, 100 mm Downstream ................................................................. 77

Appendix A

Figure A.1  Radial Velocity Polynomial Curve Fit at 2300 rpm, 635 cfm, 25 mm Downstream (N = 3) ................................................................. 85

Figure A.2  Radial Velocity Polynomial Curve Fit at 2300 rpm, 635 cfm, 25 mm Downstream (N = 4) ................................................................. 85

Figure A.3  Radial Velocity Polynomial Curve Fit at 2300 rpm, 635 cfm, 25 mm Downstream (N = 5) ................................................................. 86
NOMENCLATURE

\( p \)  Static pressure
\( u \)  Velocity normal to the fan
\( u_1 \)  Far upstream velocity
\( u_2 \)  Far downstream velocity
\( u_i \)  Velocity in \( i \) direction
\( u_j \)  Velocity in \( j \) direction
\( \tau_{ij} \)  Stress tensor
\( x_i \)  \( i \) axis
\( x_j \)  \( j \) axis
\( F_i \)  External body force
\( r \)  Radial distance from the axis of the hub
\( u_r \)  Radial velocity
\( u_\theta \)  Tangential velocity
\( f_n \)  Coefficients of pressure rise polynomial
\( g_n \)  Coefficients of radial velocity polynomial
\( h_n \)  Coefficients of tangential velocity polynomial
\( \omega \)  Fan rotating speed
\( r_{hub} \)  Radius of the fan hub
\( Q \)  Flow rate
\( r_{inlet} \)  Radius of the computational domain inlet
\( f \)  Polynomial function
\( v_i \)  Experimental radial or tangential velocity
CHAPTER 1 INTRODUCTION

The automotive industry requirements of increasing the interior passenger space, noise reduction, fuel efficiency and styling of the vehicle has led to a reduction of available underhood space. The smaller engine compartment creates higher underhood temperature and lower air cooling flow, making the underhood thermal management problem more demanding. A high performance cooling fan is required to alleviate this problem. In order to make design decisions, engineers traditionally had to package underhood components in the vehicle design phase based on experience, build a prototype, test it, analyze the test results and make any necessary changes. A new prototype would then be built and the cycle was repeated until the acceptable design was achieved. Meanwhile, the testing methods can only obtain the data at the exact location where measurement is taken. For example, hot wire can only measure the flow velocities where the wire is located. The high cost of designing and conducting experiments and the need for more detailed flow field information motivates the Computational Fluid Dynamics (CFD) simulation for the study of the underhood airflow.

In underhood thermal management CFD simulation, the simulation of the fan is one of the most challenging problems. Full 3D simulation of an automotive cooling fan requires very fine and complicated grid generation to simulate the actual fan geometry, which is labour and computationally intensive. Furthermore, it seems that accurate CFD fan simulation requires the implementation of a transient solver with sliding mesh capability, since simpler models such as “mixing plane” and “multiple reference frame” have not been shown to consistently predict the correct flow field in the wake region of the fan. Due to the significant CPU time required for a transient calculation, it is not feasible to incorporate this type of fan calculation into an overall underhood simulation. The industry requires a fan simulation which is relatively fast. A simple fan model that is accurate enough to predict pressure rise and flow velocities at different radial and axial locations is needed.

Commercial CFD codes used by the automotive industry have simple fan models available to the user. In particular, FLUENT implements a “pressure jump” fan model through the source term approach. This fan model is essentially an extension of the well
known "actuator disc" model for purely axial fans. For fans which generate significant radial and tangential flow, FLUENT also allows the user to input velocity field data immediately behind the fan, as boundary conditions on the non-axial components of velocity. It appears that this feature has been of very little use to date, probably due to the lack of available data on the swirl components behind the fan blades.

The overall objective of the present study is to evaluate the fan model in the CFD code FLUENT. Pressure rise and velocities at 25 mm and 100 mm downstream planes of the fan were measured, using the Fan Test Facility at the University of Windsor. The FLUENT Fan Model requires the input of a pressure rise polynomial, and also allows radial velocity and tangential velocity polynomials to be prescribed on the fan face. Assuming the velocities measured on the 25 mm downstream plane were measured at the fan, velocity polynomials were obtained by curve fitting the circumferential averaged experimental velocity data as a function of radial location. The pressure rise polynomials were also derived by curve fitting pressure rise as a polynomial function of air velocity normal to the fan. The CFD prediction and experimental results are compared for the pressure rise and circumferential averaged velocities at different radial locations on the 100 mm downstream plane of the fan, over a range of operating conditions (i.e., specified fan rotating speed and flow rate).

This thesis is organized as follows: A survey of the relevant literature is presented in Chapter 2. The experimental facility and the procedures for acquiring pressure and velocity data are described in Chapter 3. This chapter also contains plots of the experimental velocity contours and the pressure rise versus flow rate data. Chapter 4 is devoted to the CFD simulation using the FLUENT fan model. The polynomials required in the model are derived and the computational setup is described. The results obtained from the experiments and the numerical simulations are presented and discussed in Chapter 5. Finally, some conclusions and recommendations are suggested in Chapter 6.
CHAPTER 2  LITERATURE REVIEW

In order to evaluate the CFD simulation of the automotive fan, CFD results should be compared with the experimental data. Thus, some researchers have set up test facilities to fulfill the experiment. Others have developed CFD models to simulate the fan. Some researchers have also developed the cooling models comprised of the fan and other pertinent underhood compartments.

2.1 Fan Experiments

The automotive fan literature contains a considerable number of papers concerning experimental measurements of overall integral quantities such as volume flow rate, pressure rise, power and efficiency (cf., e.g. [1]). Some researchers have also considered the effect that the installation or type of system has on the performance (cf., e.g. [2]). Flow field details are difficult to measure because of the flow unsteadiness and relatively high level of turbulence. Recently, Tillman et al [3] reported on measurements in the near-field three-dimensional wake of a low speed axial fan using hot wire anemometry.

Morris et al [4] used a hot wire to measure axial, radial and tangential velocity components in the wake of an automotive cooling fan. The volume flow rate through the system was measured using a unique moment-of-momentum flux device. Time-averaged velocity vector directions were approximated by suspending and photographing a tuft in the fan wake. Digital images of the tuft and reference angles were acquired to locate the mean flow angles at 40 positions for four flow rate conditions. Hot wires in an X configuration were aligned with these mean flow angles. The time series data acquired at these locations were phase averaged with respect to a fixed reference point on the rotating fan. An ensemble of 900 realizations provided the sample population for the statistical quantities. These data were acquired such that 370 samples per revolution could be averaged.

Nourse [5] developed a Fan Test Facility and procedures to measure the detailed velocity field downstream of the fan, using a method similar to Morris. Three-dimensional velocity vector information can be acquired at planes downstream of the fan. The fan is attached to a vertical rotating shaft. The "ram air" effect is simulated by
drawing additional air through the fan using a blower on the exhaust from the test chamber. An orifice plate is used to determine the overall flow rate through the facility.

2.2 Fan CFD Simulation

Base [6] developed a model to predict axial fan performance using a 3D boundary element method and lifting line theory. He obtained good agreement with steady flow experiments for volume flow rate and pressure rise but could not evaluate the drag force since the theory is based on potential flow. A "total thrust" formula was used to estimate real flow forces, i.e., drag, in addition to lift. No evaluation of the flow details was presented.

Capdevila and Pharaoh [7] used the CFD software TASCflow3D to simulate the flow of axial automotive fans. Three fan geometries were analyzed. Comparisons were made with the measured axial fan static pressure. Good agreement with experiments was observed for the static pressure gradients at design and high flow rates. The CFD simulation for two of the geometries predicted a significant amount of radial flow along the blade leading edge. For the fan with low levels of radial and recirculating flow at the design flow rate, three computational grids were applied. Low flow rate predictions were sensitive to grid quality and optimization. They did not simulate actual laboratory conditions such as the large inlet region and the sudden expansion after the fan.

Reister and Ross [8] used a commercial CFD code to analyse the flow through an axial fan including system effects. In the calculations, a multiple reference frame approach, a transient moving mesh and a body force model were investigated and compared. A comparison of experimental and computed discharge curves showed fairly good agreement.

Tests and computation were performed by Anderson [9] to study the system effects on cooling fan air flow. The fan flow field was measured with an LDA and investigated with CFD using the streamline curvature method.

Foss et al [10] evaluated CFD models of axial fans by comparison with experimental data. In order to improve the reliability of fan design and the prediction of underhood engine cooling based on CFD, Valeo Motors and Michigan State University set up fan test facilities separately. The agreement of the pressure rise collected independently from
the two facilities for an identical fan/shroud system indicated the reliability of the experiment. At the high flow rate, both experimental pressure rises agree well with the simulation. For a mid-sized automotive fan (380 mm radius), the simulation data deviated from the experimental pressure rises at low flow rate. Detailed downstream velocity data collected in the Michigan State University facility showed good agreement with simulation of the axial and tangential velocity components. However, the radial velocity map in the detailed CFD facility simulations showed significantly lower magnitudes than the experiment.

Barron et al [11] considered three different configurations to simulate the Fan Test Facility developed by Nourse [5]. FLUENT was used to solve the 3D Navier-Stokes equations, using the standard k – ε turbulence model and its multiple reference frame capability. Three configurations of computational domain were investigated. The first configuration had larger upstream (approximately 5 fan diameters) than downstream (approximately 3 fan diameters) radial extensions. The radial extensions for the second one were opposite to the first one, i.e. the upstream was smaller than the downstream. The third configuration had upstream and downstream of equal size, approximately 3 fan diameters. All configurations had identical mesh in the core region and the same axial lengths, about 55 blade axial lengths upstream and 80 downstream of the fan. The simulations indicated that the first and third configurations provided better agreement with the experimental data. Details can also be found from [12], [13] and [14]. There was good agreement between the experimental and numerical velocity magnitude contours on planes close to the fan. On planes far from the fan, the numerical contours expand more widely than the experimental contours.

Tzanos and Chien [15] presented a simple fan model based on the “actuator disc” approximation, and the blade element and vortex theory of a propeller. Assuming the dominant body forces generated by the fan are in the axial and circumferential directions, they derived a set of equations comprised of the fan rotational speed, geometric fan data, lift and drag coefficients of the blades, and the axial and circumferential body force. These equations were solved iteratively to obtain the axial and circumferential body forces. These forces were then used as momentum source terms in the CFD code STAR-CD to represent the fan as a source of axial and circumferential body forces. Predictions
were generated for axial, radial and tangential velocities at different radial positions and at different planes downstream of the fan. Comparisons between the CFD results and the experimental data showed good agreement. Comparisons were made with predictions of a rotating reference frame model, showing that the accuracy of this simple model was comparable.

2.3 Cooling Module Models

Research efforts to develop a model for the complete cooling module have recently been reported.

Chang et al [16,17] proposed both incompressible and compressible one-dimensional models for the radiator-shroud-fan system for heavy-duty trucks. For incompressible flow, they obtained a closed form equation for the transient velocity, which is useful for engineers in evaluating effects of the relevant factors on the vehicle engine cooling air flow. For compressible flow, similarity theory was used to identify the important dimensionless parameters and the correlation of the non-dimensional mass flow rate to the dimensionless parameters was established. These one-dimensional models do not provide any information about the flow field behind the fan, which is critical to underhood flow analysis.

Habchi et al [18] developed a numerical model using computational fluid dynamics techniques for analysis of engine cooling systems for large slow moving vehicles. They developed physical models for predicting pressure losses due to screens and grills, for approximating forces exerted by the fan on the flow, and for calculating the heat transfer within the radiator. These models were incorporated into a CFD code and validated against experimental data. The CFD model accurately predicted system resistance for various components and fan performance for free inlet/free outlet conditions. However, the model over-predicted the system air flow for the engine installed fan configuration. They recommended more test measurements for comprehensive validation and more fan modeling researches, particularly in the area of fan resistance and radial flow.

Henon [19] developed a model to predict the interaction of the fan system and the heat exchangers of the engine cooling system. An experiment was designed to measure the local velocity field induced by the fan on the heat exchanger. This information was
then input into a local thermal computation model. They concluded that a better understanding of the fan-radiator interactions was needed, requiring detailed hot wire measurements and CFD calculations.

Nobel and Jain [20] used a complete 3D CAD model of all pertinent underhood components of a heavy-duty truck. The airflow and heat transfer analysis was performed using FLUENT. The heat exchangers were modeled using an approach that divided the heat exchanger core into cell zones and computed heat rejection cumulatively from zone to zone. Since the cooling fan performance has a significant influence on airflow across the heat exchangers, the detailed geometry of the blades were considered in the fan model. The simulations were performed using the multiple reference frame approach. These models were coupled together to provide an integrated methodology to access the underhood airflow and heat transfer characteristics. The CFD results showed the higher temperature areas downstream of the heat exchangers and, in particular, the non-uniform or swirling flow in the area of the engine. The CFD results also matched well with test results on coolant heat rejection, charge air cooler outlet temperature and heat exchanger pressure drop. The comparison of cooling fan performance (pressure drop vs. flow rate) between CFD results and test results showed good correlation. This methodology is flexible to allow for relatively rapid analysis of modest configuration changes while providing directional data to guide truck design engineers when packaging underhood components. However, the authors did not discuss the detailed CFD results of the 3D airflow velocity field.

2.4 Summary

Analysis of the literature indicates that there continues to be a need for development of simplified fan models which can be incorporated into large scale underhood flow simulations. The work of Tzanos and Chien [15] appears to be very promising. An alternative avenue for investigation, proposed in this thesis, is based on fully exploiting flow field data collected from fan test facilities, such as that developed by Nourse [5] and Morris et al [4].
CHAPTER 3 EXPERIMENTAL SIMULATIONS

In this chapter, a brief description of the experimental facility, experimental conditions and experimental procedures are presented. More complete details can be found in [5] and [21]. At the end of this chapter, velocity contours and pressure rises are presented.

3.1 Experimental Facility

The University of Windsor/DaimlerChrysler Fan Test Facility (Fig. 3.1) developed by Nourse, as described in [5], is part of a research project to develop a simplified model for predicting the behaviour of automotive cooling fans. The facility can be used to measure automotive fan performance and the flow field upstream and downstream of the fan. The facility is designed to acquire data that can be used to validate commercially available CFD software and guide in the development of new CFD fan models.

The facility consists of a 3m x 3m x 3m test chamber, which contains a fan and shroud combination located in the ceiling. Figure 3.1 shows a perspective view of the facility with the area of the ceiling around the fan, as well as the facility door, removed. The numbers in Fig 3.1 indicate the flow path through the facility. At a large distance from the fan (1), the air is stationary. The flow accelerates through the fan (2) and into the upper section of the facility. The facility was designed to be large enough that the flow at the inner side walls of the facility would be negligible. The fan shaft and drive mechanisms are housed at a location downstream of the fan such that the fan performance and measured velocity field are not affected. The flow is kept approximately symmetrical by the use of grating in the four corners of the facility. Elsewhere the upper and lower sections of the facility are separated by plywood. A rectangular diffuser is located at the centre of the lower section of the facility. Air is drawn through the grating separating the upper and lower sections and into the diffuser with the use of a mechanical blower (3). The “ram air” effect, which is caused by the vehicle motion, is simulated by drawing additional air through the fan into the measuring section, using the blower on the exhaust from the test chamber. The air then flows through the duct (4) and through an orifice plate (5), which is located at a sufficient distance from an elbow in the duct to produce
fully developed pipe flow. This orifice plate is used to measure the flow rate and provide a signal for its control. The flow then returns to the laboratory at a large distance from the facility (6). With this facility, the fan speed and flow rate through the fan are independently controlled. Figure 3.2 shows a cutaway view of the Fan Test Facility.

For the purpose of validating CFD models, three-dimensional velocity vector information is required on planes at specified axial locations downstream of the fan. The automotive cooling fan (Fig. 3.3) is attached to the vertical rotating shaft, which is driven by a variable speed DC motor arrangement. An X-array hot wire probe can be positioned at different downstream locations to measure the three components of velocity over a plane perpendicular to the fan axis. The pressure rise across the fan is measured with pressure transducers.

The outputs from the hot wire and the pressure and flow measurement devices are sent to a special data acquisition card with sample and hold capability installed in a Pentium microcomputer. From this data, phase averaged velocity maps can be made on the measurement planes.

### 3.2 Experimental Conditions

Measurements were taken on a five blade automotive cooling fan, as shown in Fig. 3.3. The experimental data were obtained at the following six conditions:

<table>
<thead>
<tr>
<th>Fan Rotating Speed (rpm)</th>
<th>Flow Rate (cfm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>2300</td>
<td>635</td>
</tr>
<tr>
<td></td>
<td>800</td>
</tr>
<tr>
<td></td>
<td>1123</td>
</tr>
<tr>
<td>2700</td>
<td>737</td>
</tr>
<tr>
<td></td>
<td>800</td>
</tr>
<tr>
<td></td>
<td>1327</td>
</tr>
</tbody>
</table>

Table 3.1 Fan Test Experiment Conditions

In each condition, three-dimensional velocity vectors are measured at two downstream planes, i.e., 25 mm plane and 100 mm plane. On each plane, measurements were taken at a total of 20 radial locations, from 15 mm to 95.75 mm away from the hub. The distance between radial positions is 4.25 mm. At each radial location, measurements
were taken every 2°. Thus, measurements were taken at the intersection points on a grid as shown in Fig. 3.4.

If velocities were measured at the two planes one after the other, the whole experiment would last about 20 hours. The change of environmental conditions also made the initial calibration of the hot wire invalid. Thus the measurement for one condition was usually taken on two separate days, one for 25 mm plane, and the other for 100 mm plane. For each day, the hot wire must be calibrated before taking measurements.

3.3 Outline of Experimental Procedures

There are several steps involved in setting up and taking measurements, summarized below.

1. Balance the bridge

In order to prepare the probe for calibration, the bridge must be adjusted to result in a probe resistance overheat ratio of approximately 0.8. This will prevent the wire temperature from becoming too high to burn out the wire.

2. Calibrate the hot wire

In order to compensate for contamination and changes in room temperature, the hot wire must be calibrated before taking measurements.

3. Take measurements

The hot wire probe is placed in the support, the wire is moved to the required downstream plane of the fan, the probe is aligned in the mean velocity direction of the flow and then measurements are taken.

Step 1 is required only if a new wire is used. More complete details can be found in [5] and [21].

3.4 Velocity Contours and Pressure Rise

Figures 3.5 - 3.40 show the three-dimensional velocity contours for all the conditions. Figures 3.41 - 3.52 show the velocity magnitude contours. The radius of the contours ranges from 0.07375 m to 0.1545 m, corresponding to the measurement locations of the hot wire. The uncertainty of the velocity data is ±0.2 m/s. Details
regarding the uncertainty can be found in [5]. These contour plots were generated with the FIELDVIEW software program [22, 23], which is commercially available from Intelligent Light. The direction of the positive axial velocities in the contours is downward, i.e., from upstream to downstream. The positive radial velocities point from the root to the tip of the blade, i.e., away from the hub. From a point above the fan, the fan rotates in the clockwise direction but the clockwise tangential velocity is defined as negative.

Table 3.2 shows the pressure rise vs. flow rate at 2300 rpm. Table 3.3 shows the pressure rise vs. flow rate at 2700 rpm. These experimental results are discussed in Chapter 5.

<table>
<thead>
<tr>
<th>Flow Rate (cfm)</th>
<th>Pressure Rise (Pa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>600</td>
<td>108</td>
</tr>
<tr>
<td>650</td>
<td>96</td>
</tr>
<tr>
<td>700</td>
<td>86</td>
</tr>
<tr>
<td>750</td>
<td>75</td>
</tr>
<tr>
<td>800</td>
<td>67</td>
</tr>
<tr>
<td>850</td>
<td>57</td>
</tr>
<tr>
<td>900</td>
<td>49</td>
</tr>
<tr>
<td>950</td>
<td>40</td>
</tr>
<tr>
<td>1000</td>
<td>30</td>
</tr>
<tr>
<td>1050</td>
<td>22</td>
</tr>
<tr>
<td>1100</td>
<td>15</td>
</tr>
<tr>
<td>1150</td>
<td>0</td>
</tr>
</tbody>
</table>

Table 3.2  Pressure Rise vs. Flow Rate at 2300 rpm
<table>
<thead>
<tr>
<th>Flow Rate (cfm)</th>
<th>Pressure Rise (Pa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>600</td>
<td>160</td>
</tr>
<tr>
<td>650</td>
<td>150</td>
</tr>
<tr>
<td>700</td>
<td>140</td>
</tr>
<tr>
<td>750</td>
<td>130</td>
</tr>
<tr>
<td>800</td>
<td>120</td>
</tr>
<tr>
<td>850</td>
<td>110</td>
</tr>
<tr>
<td>900</td>
<td>98</td>
</tr>
<tr>
<td>950</td>
<td>85</td>
</tr>
<tr>
<td>1000</td>
<td>75</td>
</tr>
<tr>
<td>1050</td>
<td>65</td>
</tr>
<tr>
<td>1100</td>
<td>55</td>
</tr>
<tr>
<td>1150</td>
<td>47</td>
</tr>
<tr>
<td>1200</td>
<td>35</td>
</tr>
<tr>
<td>1250</td>
<td>24</td>
</tr>
<tr>
<td>1300</td>
<td>20</td>
</tr>
<tr>
<td>1350</td>
<td>10</td>
</tr>
<tr>
<td>1375</td>
<td>0</td>
</tr>
</tbody>
</table>

Table 3.3  Pressure Rise vs. Flow Rate at 2700 rpm
Figure 3.1  Perspective View of Fan Test Facility

Figure 3.2  Cutaway View of Fan Test Facility
Figure 3.3  Automotive Cooling Fan

Figure 3.4  Measurement Grid
Figure 3.5  Contours of Axial Velocity at 2300 rpm, 635 cfm, 25 mm Downstream

Figure 3.6  Contours of Axial Velocity at 2300 rpm, 635 cfm, 100 mm Downstream
Figure 3.7 Contours of Radial Velocity at 2300 rpm, 635 cfm, 25 mm Downstream

Figure 3.8 Contours of Radial Velocity at 2300 rpm, 635 cfm, 100 mm Downstream
Figure 3.9  Contours of Tangential Velocity at 2300 rpm, 635 cfm, 25 mm Downstream

Figure 3.10  Contours of Tangential Velocity at 2300 rpm, 635 cfm, 100 mm Downstream
Figure 3.11  Contours of Axial Velocity at 2300 rpm, 800 cfm, 25 mm Downstream

Figure 3.12  Contours of Axial Velocity at 2300 rpm, 800 cfm, 100 mm Downstream
Figure 3.13  Contours of Radial Velocity at 2300 rpm, 800 cfm, 25 mm Downstream

Figure 3.14  Contours of Radial Velocity at 2300 rpm, 800 cfm, 100 mm Downstream
Figure 3.15  Contours of Tangential Velocity at 2300 rpm, 800 cfm, 25 mm Downstream

Figure 3.16  Contours of Tangential Velocity at 2300 rpm, 800 cfm, 100 mm Downstream
Figure 3.17  Contours of Axial Velocity at 2300 rpm, 1123 cfm, 25 mm Downstream

Figure 3.18  Contours of Axial Velocity at 2300 rpm, 1123 cfm, 100 mm Downstream
Figure 3.19  Contours of Radial Velocity at 2300 rpm, 1123 cfm, 25 mm Downstream

Figure 3.20  Contours of Radial Velocity at 2300 rpm, 1123 cfm, 100 mm Downstream
Figure 3.21  Contours of Tangential Velocity at 2300 rpm, 1123 cfm, 25 mm Downstream

Figure 3.22  Contours of Tangential Velocity at 2300 rpm, 1123 cfm, 100 mm Downstream
Figure 3.23  Contours of Axial Velocity at 2700 rpm, 737 cfm, 25 mm Downstream

Figure 3.24  Contours of Axial Velocity at 2700 rpm, 737 cfm, 100 mm Downstream
Figure 3.25  Contours of Radial Velocity at 2700 rpm, 737 cfm, 25 mm Downstream

Figure 3.26  Contours of Radial Velocity at 2700 rpm, 737 cfm, 100 mm Downstream
Figure 3.27  Contours of Tangential Velocity at 2700 rpm, 737 cfm, 25 mm Downstream

Figure 3.28  Contours of Tangential Velocity at 2700 rpm, 737 cfm, 100 mm Downstream
Figure 3.29  Contours of Axial Velocity at 2700 rpm, 800 cfm, 25 mm Downstream

Figure 3.30  Contours of Axial Velocity at 2700 rpm, 800 cfm, 100 mm Downstream
Figure 3.31  Contours of Radial Velocity at 2700 rpm, 800 cfm, 25 mm Downstream

Figure 3.32  Contours of Radial Velocity at 2700 rpm, 800 cfm, 100 mm Downstream
Figure 3.33  Contours of Tangential Velocity at 2700 rpm, 800 cfm, 25 mm Downstream

Figure 3.34  Contours of Tangential Velocity at 2700 rpm, 800 cfm, 100 mm Downstream
Figure 3.35  Contours of Axial Velocity at 2700 rpm, 1327 cfm, 25 mm Downstream

Figure 3.36  Contours of Axial Velocity at 2700 rpm, 1327 cfm, 100 mm Downstream
Figure 3.37  Contours of Radial Velocity at 2700 rpm, 1327 cfm, 25 mm Downstream

Figure 3.38  Contours of Radial Velocity at 2700 rpm, 1327 cfm, 100 mm Downstream
Figure 3.39  Contours of Tangential Velocity at 2700 rpm, 1327 cfm, 25 mm Downstream

Figure 3.40  Contours of Tangential Velocity at 2700 rpm, 1327 cfm, 100 mm Downstream
Figure 3.41  Contours of Velocity Magnitude at 2300 rpm, 635 cfm, 25 mm Downstream

Figure 3.42  Contours of Velocity Magnitude at 2300 rpm, 635 cfm, 100 mm Downstream
Figure 3.43  Contours of Velocity Magnitude at 2300 rpm, 800 cfm, 25 mm Downstream

Figure 3.44  Contours of Velocity Magnitude at 2300 rpm, 800 cfm, 100 mm Downstream
Figure 3.45  Contours of Velocity Magnitude at 2300 rpm, 1123 cfm, 25 mm Downstream

Figure 3.46  Contours of Velocity Magnitude at 2300 rpm, 1123 cfm, 100 mm Downstream
Figure 3.47  Contours of Velocity Magnitude at 2700 rpm, 737 cfm, 25 mm Downstream

Figure 3.48  Contours of Velocity Magnitude at 2700 rpm, 737 cfm, 100 mm Downstream
Figure 3.49  Contours of Velocity Magnitude at 2700 rpm, 800 cfm, 25 mm Downstream

Figure 3.50  Contours of Velocity Magnitude at 2700 rpm, 800 cfm, 100 mm Downstream
Figure 3.51  Contours of Velocity Magnitude at 2700 rpm, 1327 cfm, 25 mm Downstream

Figure 3.52  Contours of Velocity Magnitude at 2700 rpm, 1327 cfm, 100 mm Downstream
CHAPTER 4  CFD SIMULATIONS

If a full 3D geometry model (including the shape of the blades) is used in the numerical simulation, the model will be very complicated and very fine grids are required for the CFD simulation. Once the geometry of the fan is changed, the model must also be changed. This is very labour intensive and places considerable demand on computing resources. Meanwhile, industry is mostly interested in the pressure rise across the fan and the circumferential averaged velocities downstream of the fan, which are important factors for designing the fan and the underhood components.

4.1 Actuator Disc Theory

The pressure rise can be predicted using the “actuator disc” theory developed by Rankine in 1865 for propellers and by Betz in 1920 for windmills [24]. In the “actuator disc” model, the fan is represented by a disc face with the same diameter as the fan but with negligible thickness in the axial direction. All elements of fluid passing through the disc undergo an equal increase of pressure. Radial and circumferential velocities are neglected in this model. In this theory the pressure rise $\Delta p$ across the disc is given by

$$\Delta p = \frac{1}{2} \rho (u_2^2 - u_1^2)$$  \hspace{1cm} (4.1)

where $u_2$ is the far downstream velocity, $u_1$ is the far upstream velocity, and $\rho$ is the fluid density. The “actuator disc” theory does not impart any rotational or radial motion to the flow.

4.2 FLUENT Fan Model

The CFD software FLUENT [25] contains a simple fan model. In the FLUENT fan model, a fan is considered to be infinitely thin, and the discontinuous pressure rise across it is specified as a function of the normal velocity through the fan. The relationship is assumed to be a polynomial of the form

$$\Delta p = \sum_{n=1}^{N} f_n u^{n-1}, \hspace{1cm} 1 \leq N \leq 7$$  \hspace{1cm} (4.2)
where $\Delta p$ is the pressure jump in Pa, $f_n$ are the pressure jump polynomial coefficients, and $u$ is the magnitude of the local fluid velocity normal to the fan. This model may be regarded as an extension of the "actuator disc" model.

FLUENT implements this fan model by using the Source Term Method. To illustrate this method, consider the conservation of momentum in the $ith$ direction in an inertial reference frame,

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + \rho g_i + F_i \tag{4.3}$$

where $p$ is the static pressure, $\tau_{ij}$ is the stress tensor, and $g_i$ and $F_i$ are the gravitational acceleration and external body forces in the $ith$ direction respectively. For a fan simulation, if $i$ is the axial direction, $F_i$ is set to zero out of the fan zone. In the fan zone,

$$F_i = \frac{1}{\Delta x_i} (\Delta p)$$

where $\Delta p$ is given by Eq.(4.2). Here the fan zone is referred to as the source zone and $F_i$ is the source term. This model works well for flows that are primarily axial, i.e., with small radial and tangential velocity components.

For flows with significant three-dimensional characteristics, the values of the tangential and radial velocity fields can be imposed on the fan surface in FLUENT to generate swirl. These velocities can be specified as polynomial functions of the radial distance from the fan centre. The radial velocity can be specified as:

$$u_r = \sum_{n=-1}^{N} g_n r^n; \quad -1 \leq N \leq 6. \tag{4.4}$$

The tangential velocity can be specified as:

$$u_\theta = \sum_{n=-1}^{N} h_n r^n; \quad -1 \leq N \leq 6 \tag{4.5}$$

where $u_r$ and $u_\theta$ are the radial and tangential velocities, with polynomial coefficients $g_n$ and $h_n$ respectively, and $r$ is the distance from the fan centre.

The manufacturer of the fan can provide the fan performance through the relation between pressure rise ($\Delta p$) and volume flow rate ($Q$), from which the coefficients in Eq.
(4.2) can be deduced. But, normally, no data on the radial and tangential velocity are provided by the fan manufacturer. However, the radial and tangential velocity components obtained from our experiments can be used to deduce the coefficients in Eq. (4.4) and (4.5).

4.3 Producing the Polynomials of the Fan Model

In order to apply the fan model, pressure rise and swirl velocity polynomials were produced with the Least Squares Curve Fit method.

4.3.1 Pressure Rise

The pressure rise polynomials can be produced from the flow rates and pressure rises of the experiments, using the least squares curve fit method. The tip radius of the fan blade is 0.15145 m and the radius of the hub is 0.0575 m. The flow rate $Q$, usually given in cubic feet per minute (cfm) must be converted to cubic metres per second ($m^3/s$). The relation between the average velocity normal to the fan $u$ (in $m/s$) and the flow rate $Q$ (in $cfm$) is:

$$u = \frac{Q(12 \times 0.0254)^3}{60\pi(0.15145^2 - 0.0575^2)} = 7.65 \times 10^{-3} Q.$$  \hspace{1cm} (4.6)

It is also convenient to introduce a non-dimensional form of pressure rise $\Delta \bar{p}$ and velocity magnitude normal to the fan $\bar{u}$, defined by:

$$\Delta \bar{p} = \frac{\Delta p}{\rho \omega^2 R^2},$$  \hspace{1cm} (4.7)

$$\bar{u} = \frac{u}{\omega R}$$  \hspace{1cm} (4.8)

where $\rho$ is the air density ($kg/m^3$), $\omega$ is fan rotating speed (rad/s) and $R$ is the radius of the fan blade (m). Based on the experiments described in Chapter 3, we take $\rho = 1.18$ kg/m$^3$. Table 4.1 shows the pressure rise versus velocity magnitude normal to the fan at 2300 rpm. Table 4.2 shows the pressure rise versus velocity magnitude normal to the fan at 2700 rpm.
<table>
<thead>
<tr>
<th>Flow Rate (cfm)</th>
<th>Velocity (m/s)</th>
<th>Non-dimensional Velocity</th>
<th>Pressure Rise (Pa)</th>
<th>Non-dimensional Pressure Rise</th>
</tr>
</thead>
<tbody>
<tr>
<td>600</td>
<td>4.592</td>
<td>0.124</td>
<td>108</td>
<td>0.066</td>
</tr>
<tr>
<td>650</td>
<td>4.974</td>
<td>0.134</td>
<td>96</td>
<td>0.059</td>
</tr>
<tr>
<td>700</td>
<td>5.357</td>
<td>0.145</td>
<td>86</td>
<td>0.053</td>
</tr>
<tr>
<td>750</td>
<td>5.739</td>
<td>0.155</td>
<td>75</td>
<td>0.046</td>
</tr>
<tr>
<td>800</td>
<td>6.122</td>
<td>0.165</td>
<td>67</td>
<td>0.041</td>
</tr>
<tr>
<td>850</td>
<td>6.505</td>
<td>0.176</td>
<td>57</td>
<td>0.035</td>
</tr>
<tr>
<td>900</td>
<td>6.887</td>
<td>0.186</td>
<td>49</td>
<td>0.03</td>
</tr>
<tr>
<td>950</td>
<td>7.27</td>
<td>0.196</td>
<td>40</td>
<td>0.024</td>
</tr>
<tr>
<td>1000</td>
<td>7.653</td>
<td>0.207</td>
<td>30</td>
<td>0.018</td>
</tr>
<tr>
<td>1050</td>
<td>8.035</td>
<td>0.217</td>
<td>22</td>
<td>0.013</td>
</tr>
<tr>
<td>1100</td>
<td>8.418</td>
<td>0.227</td>
<td>15</td>
<td>9.18E-3</td>
</tr>
<tr>
<td>1150</td>
<td>8.8</td>
<td>0.238</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

Table 4.1  Pressure Rise vs. Velocity Magnitude Normal to the Fan at 2300 rpm

<table>
<thead>
<tr>
<th>Flow Rate (cfm)</th>
<th>Velocity (m/s)</th>
<th>Non-dimensional Velocity</th>
<th>Pressure Rise (Pa)</th>
<th>Non-dimensional Pressure Rise</th>
</tr>
</thead>
<tbody>
<tr>
<td>600</td>
<td>4.592</td>
<td>0.106</td>
<td>160</td>
<td>0.071</td>
</tr>
<tr>
<td>650</td>
<td>4.974</td>
<td>0.114</td>
<td>150</td>
<td>0.067</td>
</tr>
<tr>
<td>700</td>
<td>5.357</td>
<td>0.123</td>
<td>140</td>
<td>0.062</td>
</tr>
<tr>
<td>750</td>
<td>5.739</td>
<td>0.132</td>
<td>130</td>
<td>0.058</td>
</tr>
<tr>
<td>800</td>
<td>6.122</td>
<td>0.141</td>
<td>120</td>
<td>0.053</td>
</tr>
<tr>
<td>850</td>
<td>6.505</td>
<td>0.15</td>
<td>110</td>
<td>0.049</td>
</tr>
<tr>
<td>900</td>
<td>6.887</td>
<td>0.158</td>
<td>98</td>
<td>0.044</td>
</tr>
<tr>
<td>950</td>
<td>7.27</td>
<td>0.167</td>
<td>85</td>
<td>0.038</td>
</tr>
<tr>
<td>1000</td>
<td>7.653</td>
<td>0.176</td>
<td>75</td>
<td>0.033</td>
</tr>
<tr>
<td>1050</td>
<td>8.035</td>
<td>0.185</td>
<td>65</td>
<td>0.029</td>
</tr>
<tr>
<td>1100</td>
<td>8.418</td>
<td>0.194</td>
<td>55</td>
<td>0.024</td>
</tr>
<tr>
<td>1150</td>
<td>8.8</td>
<td>0.202</td>
<td>47</td>
<td>0.021</td>
</tr>
<tr>
<td>1200</td>
<td>9.183</td>
<td>0.211</td>
<td>35</td>
<td>0.016</td>
</tr>
<tr>
<td>1250</td>
<td>9.566</td>
<td>0.22</td>
<td>24</td>
<td>0.011</td>
</tr>
<tr>
<td>1300</td>
<td>9.948</td>
<td>0.229</td>
<td>20</td>
<td>8.882E-3</td>
</tr>
<tr>
<td>1350</td>
<td>10.33</td>
<td>0.238</td>
<td>10</td>
<td>4.441E-3</td>
</tr>
<tr>
<td>1375</td>
<td>10.52</td>
<td>0.242</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

Table 4.2  Pressure Rise vs. Velocity Magnitude Normal to the Fan at 2700 rpm
The FLUENT fan model allows one to choose up to a sixth degree polynomial for the pressure jump (see Eq. (4.2)). The experimental data collected in the Fan Test Facility, plotted in Fig 4.1, shows that the relationship between $\Delta p$ and $u$ is essentially linear. Hence, in Eq. (4.2), we take $N = 2$. Using the least squares curve fit method, the linear relationships between $\Delta p$ and $u$, shown in Fig. 4.1, are:

$$\Delta p = 217.67 - 24.481u \quad (2300 \text{ rpm}),$$  \hspace{1cm} (4.7)  

$$\Delta p = 282.76 - 26.812u \quad (2700 \text{ rpm}).$$  \hspace{1cm} (4.8)

Alternatively, one can use the non-dimensional form. As seen from Fig. 4.2, the experimental data collapses to a single performance curve, independent of the fan rotating speed. Thus the data in Table 4.1 and 4.2 can be used to generate a single linear relationship between $\bar{\Delta p}$ and $\bar{u}$, which can then be used to determine the fan model pressure rise coefficients for any rotating speed. Figure 4.2 shows the linear curve fit for the non-dimensional pressure rise versus velocity magnitude normal to the fan.

4.3.2 Velocity Components

For a specified fan rotating speed and flow rate, three components of velocities have been measured at two different downstream planes, as described in Chapter 3. One plane is located at 25 mm downstream of the fan and the other is at 100 mm downstream. The downstream location is actually measured from the base of the hub. Hence, the 25 mm measurement plane actually lies about 11 mm below the lowest point on the fan blade. Assuming the measurements taken on the 25 mm plane were taken at the bottom of fan blade, swirl velocity polynomials can be produced from this data using the least squares curve fit method.

First, the radial and tangential velocity components on the 25 mm downstream plane are circumferentially averaged. It is not possible to take measurements too close to the hub, so the smallest radial position at which the data has been collected is $r = 0.07375$ m. However, it is reasonable to apply the no slip boundary condition at the hub. In fact, if the no slip boundary condition at the hub is not used, the radial velocity polynomial tends to $\pm \infty$ as the hub is approached, rather than zero. Also, the tangential velocity

43
polynomial predicts that \( u_\theta \) will not become its true value, i.e., \( \omega r_{hub} \), where \( r_{hub} \) is the hub radius, at the hub, as seen in Figs. 4.3 and 4.4. Hence, at the surface of the hub, we specify the averaged radial velocity as zero and the averaged tangential velocity as \( \omega r_{hub} \). Furthermore, as seen in Fig. 4.3, the radial velocity polynomial drops well below the experimental values before recovering to reach zero at the hub. A similar behaviour is seen at other operating conditions. In fact, at 635 cfm, 2300 rpm, the radial velocity become positive, then negative, within a very small distance from the hub. This behaviour is contrary to intuition and also conflicts with that seen from the results of full 3D CFD simulations [14]. For this reason, it was decided to add two more points to the data, between the hub and the first measurement point. These two points were added by fitting a quadratic between the hub and the first two measured values. A typical configuration of the polynomial is shown in Fig. 4.5. Then, the coefficients in Eqs. (4.4) and (4.5) are obtained by curve fitting these circumferentially averaged values from the 25 mm plane. The polynomial degree \( N \) in Eqs. (4.4) and (4.5) has been chosen as six, which is the maximum degree available in the FLUENT fan model, because the sum of the squares of the residuals of the sixth degree polynomial curve fit is much smaller than other lesser degree polynomial curve fits (see Appendix A for details). The polynomial coefficients for \( u_r \) and \( u_\theta \) are shown in Tables 4.3 and 4.4 respectively. The resulting polynomials are shown in Figs. 4.6 - 4.17.

<table>
<thead>
<tr>
<th>Coefficient</th>
<th>2300 rpm</th>
<th>2700 rpm</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>635 cfm</td>
<td>800 cfm</td>
</tr>
<tr>
<td>( g_{-1} )</td>
<td>201.2</td>
<td>153.6</td>
</tr>
<tr>
<td>( g_0 )</td>
<td>-1.516E4</td>
<td>-1.154E4</td>
</tr>
<tr>
<td>( g_1 )</td>
<td>4.783E5</td>
<td>3.657E5</td>
</tr>
<tr>
<td>( g_2 )</td>
<td>-8.19E6</td>
<td>-6.329E6</td>
</tr>
<tr>
<td>( g_3 )</td>
<td>8.224E7</td>
<td>6.453E7</td>
</tr>
<tr>
<td>( g_4 )</td>
<td>-4.848E8</td>
<td>-3.872E8</td>
</tr>
<tr>
<td>( g_5 )</td>
<td>1.556E9</td>
<td>1.267E9</td>
</tr>
<tr>
<td>( g_6 )</td>
<td>-2.099E9</td>
<td>-1.743E9</td>
</tr>
</tbody>
</table>

Table 4.3  Circumferential Averaged Radial Velocity Curve Fit Polynomial Coefficients (Data from 25 mm Downstream Plane)
<table>
<thead>
<tr>
<th>Coefficient</th>
<th>2300 rpm</th>
<th>2700 rpm</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>635 cfm</td>
<td>800 cfm</td>
</tr>
<tr>
<td>$h_1$</td>
<td>-301.7</td>
<td>-235.4</td>
</tr>
<tr>
<td>$h_2$</td>
<td>2.042E4</td>
<td>1.535E4</td>
</tr>
<tr>
<td>$h_3$</td>
<td>-5.865E5</td>
<td>-4.245E5</td>
</tr>
<tr>
<td>$h_5$</td>
<td>-8.665E7</td>
<td>-5.833E7</td>
</tr>
<tr>
<td>$h_6$</td>
<td>4.808E8</td>
<td>3.128E8</td>
</tr>
</tbody>
</table>

Table 4.4  Circumferential Averaged Tangential Velocity Curve Fit Polynomial Coefficients (Data from 25 mm Downstream Plane)

4.4 The Computational Domain

According to Barron et al [11], it is preferable to adopt a configuration with large upstream region (Fig. 4.18). The radius of the upstream part of the domain is 0.76 m, and its axial length is 0.954 m. The radial and axial length of the downstream portion are 0.46 m and 1.49 m respectively. The fan zone is a disc with zero thickness and 0.15145 m radial extension. The fan hub extends along the downstream part from the fan to the outlet with a radius of 0.0576 m. The radius of the inlet is 0.76 m. The radius of the outlet is equal to the fan radius, i.e., 0.15145 m. The mesh is block-structured and created using ICEM HEXA [26]. There are 117421 cells in the mesh and 1305 cells in the fan zone. The mesh distribution is illustrated in Fig. 4.19. Figure 4.20 shows the mesh in the core region around the fan zone.

Because the domain is cylindrical and the flow is circumferentially “mixed out”, a slice of the domain was used for the computation, specifying periodic boundary condition on both sides of the slice. Thus the number of the nodes is decreased, as well as the time for computation. The full 3D geometry simulations [11] used one fifth of the domain for computation because the fan has five blades. The mesh used in this study was also constructed on one fifth of the domain.
4.5 Numerical Algorithm

FLUENT has been used to solve the 3D Navier-Stokes equations, using Finite Volume Method. Both structured and unstructured cells can be applied in FLUENT. In this study, structured hexahedral cells, cell-centre scheme, segregated solver, quick scheme discretization, standard $k-\varepsilon$ turbulence model and standard wall functions are used. Airflow is assumed isothermal and steady. FLUENT solves the unsteady equations, marching to the steady state. The velocity magnitudes illustrated in Figs. 3.41 - 3.52 indicated that the Mach number for these flows is small, less than 0.3. In this case, the flow can be considered as incompressible. The velocity magnitude range of the CFD simulations also validates this assumption.

A uniform axial velocity is specified on the inlet plane, corresponding to a given flow rate, i.e.

$$u = \frac{Q(12 \times 0.0254)^3}{60\pi (r_{inlet}^2)} = 2.6 \times 10^{-4} Q$$

(4.9)

where $u$ is the axial velocity at the inlet and $r_{inlet}$ is the radius of the inlet, i.e., 0.76 m. Table 4.5 shows the inlet velocity for each condition.

<table>
<thead>
<tr>
<th>Fan Rotating Speed (rpm)</th>
<th>Flow Rate (cfm)</th>
<th>Velocity at Inlet (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>2300</td>
<td>635</td>
<td>0.16515</td>
</tr>
<tr>
<td></td>
<td>800</td>
<td>0.20807</td>
</tr>
<tr>
<td></td>
<td>1123</td>
<td>0.29208</td>
</tr>
<tr>
<td>2700</td>
<td>737</td>
<td>0.19168</td>
</tr>
<tr>
<td></td>
<td>800</td>
<td>0.20807</td>
</tr>
<tr>
<td></td>
<td>1327</td>
<td>0.34513</td>
</tr>
</tbody>
</table>

Table 4.5 Inlet Velocities

A pressure outlet boundary condition with "Radial Equilibrium Pressure Distribution" is specified at the outlet. When this "Radial Equilibrium Pressure Distribution" feature is active, the specified gauge pressure applies only to the position of minimum radius (relative to the axis of the rotation) at the boundary. The static pressure on the rest of the outlet zone is calculated from the assumption that radial velocity is negligible, so that the pressure gradient is given by
\[
\frac{\partial p}{\partial r} = \frac{\rho u_0^2}{r}.
\] (4.10)

The pressure jump polynomial coefficients in Eqs. (4.7) and (4.8), and the radial and tangential velocity polynomial coefficients in Eqs. (4.4) and (4.5) and Tables 4.3 and 4.4, were applied at the fan boundary.

In order to determine the significance of applying the swirl velocities, two implementations of the FLUENT fan model were tested. In one case, the performance polynomial, and the radial and tangential polynomials were all applied in the fan model. This is called “swirl” simulation. The other implementation only applied the performance polynomial. This is referred to as “no swirl” simulation.

![Graph showing pressure rise and linear curve fit](image)

**Figure 4.1** Pressure Rise and Linear Curve Fit
Figure 4.2  Non-dimensional Pressure Rise and Linear Curve Fit

Figure 4.3  Radial Velocity Polynomial Curve Fit at 2300 rpm, 800 cfm, 25 mm Downstream (20 and 21 points)
Figure 4.4  Tangential Velocity Polynomial Curve Fit at 2300 rpm, 800 cfm, 25 mm
Downstream (20 and 21 points)

Figure 4.5  Radial Velocity Polynomial Curve Fit at 2300 rpm, 635 cfm, 25 mm
Downstream (21 and 23 points)
Figure 4.6  Radial Velocity Polynomial Curve Fit at 2300 rpm, 635 cfm, 25 mm Downstream

Figure 4.7  Tangential Velocity Polynomial Curve Fit at 2300 rpm, 635 cfm, 25 mm Downstream
Figure 4.8  Radial Velocity Polynomial Curve Fit at 2300 rpm, 800 cfm, 25 mm Downstream

Figure 4.9  Tangential Velocity Polynomial Curve Fit at 2300 rpm, 800 cfm, 25 mm Downstream
Figure 4.10  Radial Velocity Polynomial Curve Fit at 2300 rpm, 1123 cfm, 25 mm Downstream

Figure 4.11  Tangential Velocity Polynomial Curve Fit at 2300 rpm, 1123 cfm, 25 mm Downstream
Figure 4.12  Radial Velocity Polynomial Curve Fit at 2700 rpm, 737 cfm, 25 mm Downstream

Figure 4.13  Tangential Velocity Polynomial Curve Fit at 2700 rpm, 737 cfm, 25 mm Downstream
Figure 4.14  Radial Velocity Polynomial Curve Fit at 2700 rpm, 800 cfm, 25 mm Downstream

Figure 4.15  Tangential Velocity Polynomial Curve Fit at 2700 rpm, 800 cfm, 25 mm Downstream
Figure 4.16  Radial Velocity Polynomial Curve Fit at 2700 rpm, 1327 cfm, 25 mm Downstream

Figure 4.17  Tangential Velocity Polynomial Curve Fit at 2700 rpm, 1327 cfm, 25 mm Downstream
Figure 4.18  Computational Domain
Figure 4.19  Computational Mesh Distribution

Figure 4.20  Core Region Mesh Distribution
CHAPTER 5 RESULTS AND DISCUSSION

The experimental and CFD results are presented and analysed in this chapter.

5.1 Experimental Results and Analysis

The analysis of axial, radial and tangential velocity, and velocity magnitude are included in this section, based on the contours in Chapter 3.

5.1.1 Axial Velocity

Table 5.1 shows the maximum and minimum axial velocity for each condition. The contours of axial velocity are plotted in Figs. 3.5, 3.11, 3.17, 3.23, 3.29 and 3.35 for the plane located 25 mm downstream of the fan, and in Figs. 3.6, 3.12, 3.18, 3.24, 3.30 and 3.36 for the 100 mm downstream plane.

<table>
<thead>
<tr>
<th>Fan Rotating Speed (rpm)</th>
<th>Flow Rate (cfm)</th>
<th>Axial Velocity (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>25 mm Downstream</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Min.</td>
</tr>
<tr>
<td>2300</td>
<td>635</td>
<td>6.8</td>
</tr>
<tr>
<td></td>
<td>800</td>
<td>5.2</td>
</tr>
<tr>
<td></td>
<td>1123</td>
<td>3.3</td>
</tr>
<tr>
<td>2700</td>
<td>737</td>
<td>8.9</td>
</tr>
<tr>
<td></td>
<td>800</td>
<td>6.0</td>
</tr>
<tr>
<td></td>
<td>1327</td>
<td>3.7</td>
</tr>
</tbody>
</table>

Table 5.1 Experimental Axial Velocity

At 25 mm downstream, the contours plots indicate that axial flow is always lower around the ring. There are high axial velocity spots at about 80% of the span. Large axial flow regions migrate towards the hub as the flow rate decreases. Figures 3.5 and 3.23 show that there are some large axial velocity spots near the hub at low flow rate. High axial flow regions are concentrated in five strips with sharp transition between high and low flow regions, corresponding to the wake of the five blades.

The data in Table 5.1 does not display any particular trends in the maximum values of axial velocity on the 25 mm plane. However, the minimum axial velocity is seen to decrease with increasing flow rate for both rpms.
At 100 mm downstream, the ranges between the maximum and minimum axial velocity are small. At low flow rate, the axial velocity is fairly uniform, as illustrated in Figs. 3.6 and 3.24. The non-uniformity increases with the increase of the flow rate. At 2300 \( \text{rpm} \), the minimum axial velocity increases slowly with flow rate, whereas it is essentially constant at 2700 \( \text{rpm} \). The maximum axial velocity increases with flow rate for both rpms.

Compared with the velocity on the 25 mm downstream plane, the velocity on the 100 mm downstream plane is less circumferentially dependent. At high flow rate, axial velocity is smallest around the ring. In general, axial velocity is smaller on the 100 mm plane than on the 25 mm plane, which indicates the flow expands radially after the fan.

### 5.1.2 Radial Velocity

Table 5.2 shows the maximum and minimum radial velocity for each condition. The contours of the radial velocity are plotted in Figs. 3.7, 3.13, 3.19, 3.25, 3.31 and 3.37 for the plane located 25 mm downstream of the fan, and in Figs. 3.8, 3.14, 3.20, 3.26, 3.32 and 3.38 for the 100 mm downstream plane.

<table>
<thead>
<tr>
<th>Fan Rotating Speed (rpm)</th>
<th>Flow Rate (cfm)</th>
<th>Radial Velocity (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>25 mm Downstream</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Min.</td>
</tr>
<tr>
<td>2300</td>
<td>635</td>
<td>-3.3</td>
</tr>
<tr>
<td></td>
<td>800</td>
<td>-3.0</td>
</tr>
<tr>
<td></td>
<td>1123</td>
<td>-1.1</td>
</tr>
<tr>
<td>2700</td>
<td>737</td>
<td>-4.2</td>
</tr>
<tr>
<td></td>
<td>800</td>
<td>-2.8</td>
</tr>
<tr>
<td></td>
<td>1327</td>
<td>-2.3</td>
</tr>
</tbody>
</table>

Table 5.2 Experimental Radial Velocity

On the 25 mm downstream plane, the maximum and the minimum radial velocities both increase with the increase of the flow rate. The range of the radial velocity is always small. The negative value of radial velocity represent the flow direction is inward, i.e., towards the hub. Figures 3.7, 3.13, 3.25 and 3.31 demonstrated that there is strong inward
flow at low flow rate. Good mixing occurs in the circumferential direction at low flow rate. As the flow rate increases, the flow near the ring changes from inward to outward.

At 100 mm downstream, the minimum radial velocity decreases as the flow rate increases. The radial flow is very small for large flow rate, which indicates there is not much expansion downstream of the fan. The radial flow has weak dependence on the circumferential direction, especially at the low flow rates, as seen from Figs 3.8, 3.14, 3.26 and 3.32.

At low flow rates, the strong inward flow at 25 mm plane becomes strong outward flow at 100 mm. In general, radial velocity contributes more to the overall velocity vector, i.e., velocity magnitude, at the 100 mm plane than at the 25 mm plane.

5.1.3 Tangential Velocity

Table 5.3 shows the maximum and minimum tangential velocity for each condition. The contours of tangential velocity are plotted in Figs. 3.9, 3.15, 3.21, 3.27, 3.33 and 3.39 for the plane located 25 mm downstream of the fan, and in Figs. 3.10, 3.16, 3.22, 3.28, 3.34 and 3.40 for the 100 mm downstream plane.

<table>
<thead>
<tr>
<th>Fan Rotating Speed (rpm)</th>
<th>Flow Rate (cfm)</th>
<th>Tangential Velocity (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>25 mm Downstream</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Min.</td>
</tr>
<tr>
<td>2300</td>
<td>635</td>
<td>-1.5</td>
</tr>
<tr>
<td></td>
<td>800</td>
<td>-3.0</td>
</tr>
<tr>
<td></td>
<td>1123</td>
<td>-2.5</td>
</tr>
<tr>
<td>2700</td>
<td>737</td>
<td>-2.8</td>
</tr>
<tr>
<td></td>
<td>800</td>
<td>-2.7</td>
</tr>
<tr>
<td></td>
<td>1327</td>
<td>-3.0</td>
</tr>
</tbody>
</table>

Table 5.3 Experimental Tangential Velocity

The negative value of the tangential velocity represents the flow in the direction of fan rotation. The magnitude of the tangential velocity is small, both at the 25 mm plane and 100 mm plane. The range of the tangential velocity is also small. At both 25 mm and 100 mm planes, the tangential velocity contributes less to the overall velocity than the axial and radial components.
5.1.4 Velocity Magnitude

Table 5.4 shows the maximum and minimum velocity magnitude for each condition. The contours of velocity magnitude are plotted in Figs. 3.41, 3.43, 3.45, 3.47, 3.49 and 3.51 for the plane located 25 mm downstream of the fan, and in Figs. 3.42, 3.44, 3.46, 3.48, 3.50 and 3.52 for the 100 mm downstream plane.

<table>
<thead>
<tr>
<th>Fan Rotating Speed (rpm)</th>
<th>Flow Rate (cfm)</th>
<th>Velocity Magnitude (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>25 mm Downstream</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Min.</td>
</tr>
<tr>
<td>2300</td>
<td>635</td>
<td>6.8</td>
</tr>
<tr>
<td></td>
<td>800</td>
<td>5.2</td>
</tr>
<tr>
<td></td>
<td>1123</td>
<td>3.3</td>
</tr>
<tr>
<td>2700</td>
<td>737</td>
<td>9.6</td>
</tr>
<tr>
<td></td>
<td>800</td>
<td>6.0</td>
</tr>
<tr>
<td></td>
<td>1327</td>
<td>3.7</td>
</tr>
</tbody>
</table>

Table 5.4 Experimental Velocity Magnitude

At 25 mm downstream, the minimum velocity magnitude decreases as the flow rate increases. There is no trend for the maximum velocity. Since the radial and tangential velocities contribute less, the velocity magnitude contours are similar to the axial velocity contours, i.e., flow is primarily axial close to the fan, as seen by comparison of Figs. 3.5 and 3.41, or Figs 3.11 and 3.43, for example.

At 100 mm downstream, the minimum velocity magnitude also decreases as the flow rate increases. There is no trend for the maximum velocity magnitude. The velocity magnitude range is smaller at 100 mm than at 25 mm. The flow is more mixed on the 100 mm plane. Contributions from radial and tangential velocity are greater than at 25 mm planes. The lowest speed is always at the ring. The highest speed lies at around 40 ~ 60 % span.

5.2 CFD Results and Analysis

The number of iterations for each case is shown in Table 5.5. The solution was considered to have converged when the residuals fell to less than $1 \times 10^{-3}$. The residual
computed by FLUENT's segregated solver is the imbalance in the discretized equation for \( \phi \), summed over all the computational cells, where \( \phi \) is one of the velocity components, or \( k \) or \( \varepsilon \). FLUENT scales the residual using a scaling factor representative of the flow rate of \( \phi \) through the domain. The segregated solver's scaled residual for the continuity equation is defined as the rate of mass creation in a cell, summed over all cells, divided by the largest absolute value of the continuity residual in the first five iterations. The details of the definition of the residual for the segregated solver can be found in [25].

<table>
<thead>
<tr>
<th>Fan Rotating Speed (rpm)</th>
<th>Flow Rate (cfm)</th>
<th>Number of Iterations</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Swirl</td>
<td>No swirl</td>
</tr>
<tr>
<td>2300</td>
<td>635</td>
<td>1345</td>
</tr>
<tr>
<td></td>
<td>800</td>
<td>985</td>
</tr>
<tr>
<td></td>
<td>1123</td>
<td>970</td>
</tr>
<tr>
<td>2700</td>
<td>737</td>
<td>1228</td>
</tr>
<tr>
<td></td>
<td>800</td>
<td>1095</td>
</tr>
<tr>
<td></td>
<td>1327</td>
<td>620</td>
</tr>
</tbody>
</table>

Table 5.5 Iteration Count

Table 5.5 shows that neither "swirl" simulation nor "no swirl" simulation has superiority on the convergence rate.

After the solution is converged, the velocities at 100 mm downstream of the fan and the pressure rise are obtained and compared with the experimental data. The pressure rise is displayed in Table 5.6. The comparisons between the experimental velocities and CFD velocities at 100 mm downstream of the fan are shown in Figs. 5.1 to 5.24.

5.2.1 Pressure Rise

Table 5.6 shows the comparison of pressure rise for different cases. The experimental pressure rise was obtained by interpolation from Eqs. (4.7) and (4.8). CFD pressure rise is the difference between the area weighted average static pressure of the 5mm downstream plane and the 5mm upstream plane of the fan. Both planes have the same radius as the fan.
<table>
<thead>
<tr>
<th>Fan Rotating Speed (rpm)</th>
<th>Flow Rate (cfm)</th>
<th>Pressure Rise (Pa)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>Exp.</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Swirl</td>
</tr>
<tr>
<td>2300</td>
<td>635</td>
<td>98.708</td>
</tr>
<tr>
<td></td>
<td>800</td>
<td>67.797</td>
</tr>
<tr>
<td></td>
<td>1123</td>
<td>7.286</td>
</tr>
<tr>
<td>2700</td>
<td>737</td>
<td>131.543</td>
</tr>
<tr>
<td></td>
<td>800</td>
<td>118.616</td>
</tr>
<tr>
<td></td>
<td>1327</td>
<td>10.487</td>
</tr>
</tbody>
</table>

Table 5.6 Comparison of Pressure Rise

At the lower flow rates, both simulations fit the experimental data very well. At the higher flow rate, both simulations deviate from the experimental data, with the "no swirl" simulation giving slightly better prediction than the "swirl" simulation.

5.2.2 Axial Velocity

The experimental and CFD axial velocity are compared in Figs. 5.1, 5.5, 5.9, 5.13, 5.17 and 5.21. The CFD results predict that the axial velocity dominates the flow. This is consistent with the experimental results, although the CFD predicts much less contribution from the other velocity components than the experiments indicate, especially at the low flow rate. Compared with the CFD curves, the experimental axial velocity curves appear more flat. The experimental axial velocity is locally maximum near the hub and at 60 ~ 75% of the blade span. The CFD maximum axial velocity is located near the root of the blade. The CFD axial velocity decreases sharply from the maximum value location to the tip of the blade. There is very little difference between the "swirl" and "no swirl" solutions for the low flow rate. Both simulations are reasonable over 70% of the blade span from the hub outward. Both severely underpredict the axial velocity near the outer ring radius. At high flow rates, neither "swirl" nor "no swirl" simulation is very good, but "swirl" is slightly better. This is seen from the figures as well as Table 5.7, in which the average axial velocities are compared.

5.2.3 Radial Velocity

The radial velocity component is illustrated in Figs 5.2, 5.6, 5.10, 5.14, 5.18 and 5.22. Both "swirl" and "no swirl" models predict almost negligible radial velocity,
compared with the measured data. The "swirl" simulation generates a small amount of radial velocity at high flow rates, but still falls far below the experimental values, as seen in Figs 5.10 and 5.22.

5.2.4 Tangential Velocity

Figures 5.3, 5.7, 5.11, 5.15, 5.19 and 5.23 illustrate the circumferential averaged tangential velocity. There is significant difference between the "swirl" and the "no swirl" CFD tangential velocities. The "no swirl" tangential velocity is almost zero everywhere (except at the hub), while the "swirl" tangential velocity is always larger than the "no swirl" velocity. Nevertheless, for low flow rates, Figs 5.3 and 5.15 indicate that even the "swirl" simulation predicts much smaller tangential velocity than the experimental tangential velocity. The "swirl" model gives good prediction of tangential velocity at intermediate flow rates and excellent agreement with experimental data at high flow rates.

5.2.5 Velocity Magnitude

Predicted and experimental velocity magnitudes are shown in Figs. 5.4, 5.8, 5.12, 5.16, 5.20 and 5.24. There is very little difference between the two models at low flow rate. At high flow rate, the "swirl" model is a little better. Both models underpredict at low flow rate. Since radial and tangential velocities are poorly predicted, at the low flow rates, the velocity magnitude prediction is very similar to the axial velocity prediction. At the higher flow rates, where the "swirl" simulation predicts stronger tangential velocity, the CFD results and the experimental results are in reasonable agreement.

5.2.6 Average Axial Velocity and Velocity Magnitude

Table 5.7 shows that the average axial and velocity magnitude is better predicted by the "swirl" computation than by the "no swirl" computation. It can also be seen that the "no swirl" simulation gives identical values for the average axial velocity and velocity magnitude, suggesting that the model will only be adequate for flows which are strongly axial. As the flow rate increases, the "swirl" model gives a better estimate of the
tangential velocity component, leading to a better prediction of the average velocity magnitude.

<table>
<thead>
<tr>
<th>Fan Rotating Speed (rpm)</th>
<th>Flow Rate (cfm)</th>
<th>Velocity (m/s)</th>
<th>Axial CFD</th>
<th>Magnitude CFD</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td>Exp. CFD</td>
<td>Exp. CFD</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Swirl</td>
<td>No swirl</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Swirl</td>
<td>No swirl</td>
</tr>
<tr>
<td>2300</td>
<td>635</td>
<td>5.8</td>
<td>5.6</td>
<td>5.6</td>
</tr>
<tr>
<td></td>
<td>800</td>
<td>7.3</td>
<td>7.0</td>
<td>7.0</td>
</tr>
<tr>
<td></td>
<td>1123</td>
<td>8.3</td>
<td>9.3</td>
<td>9.9</td>
</tr>
<tr>
<td>2700</td>
<td>737</td>
<td>5.8</td>
<td>6.7</td>
<td>6.5</td>
</tr>
<tr>
<td></td>
<td>800</td>
<td>7.3</td>
<td>6.9</td>
<td>7.1</td>
</tr>
<tr>
<td></td>
<td>1327</td>
<td>8.3</td>
<td>11.2</td>
<td>11.8</td>
</tr>
</tbody>
</table>

Table 5.7  Average Axial Velocity & Velocity Magnitude
Figure 5.1  Circumferential Averaged Axial Velocity at 2300 rpm, 635 cfm, 100 mm Downstream

Figure 5.2  Circumferential Averaged Radial Velocity at 2300 rpm, 635 cfm, 100 mm Downstream
Figure 5.3  Circumferential Averaged Tangential Velocity at 2300 rpm, 635 cfm, 100 mm Downstream

Figure 5.4  Circumferential Averaged Velocity Magnitude at 2300 rpm, 635 cfm, 100 mm Downstream
Figure 5.5  Circumferential Averaged Axial Velocity at 2300 rpm, 800 cfm, 100 mm Downstream

Figure 5.6  Circumferential Averaged Radial Velocity at 2300 rpm, 800 cfm, 100 mm Downstream
Figure 5.7  Circumferential Averaged Tangential Velocity at 2300 rpm, 800 cfm, 100 mm Downstream

Figure 5.8  Circumferential Averaged Velocity Magnitude at 2300 rpm, 800 cfm, 100 mm Downstream
Figure 5.9  Circumferential Averaged Axial Velocity at 2300 rpm, 1123 cfm, 100 mm Downstream

Figure 5.10  Circumferential Averaged Radial Velocity at 2300 rpm, 1123 cfm, 100 mm Downstream
Figure 5.11  Circumferential Averaged Tangential Velocity at 2300 rpm, 1123 cfm, 100 mm Downstream

Figure 5.12  Circumferential Averaged Velocity Magnitude at 2300 rpm, 1123 cfm, 100 mm Downstream
Figure 5.13  Circumferential Averaged Axial Velocity at 2700 rpm, 737 cfm, 100 mm Downstream

Figure 5.14  Circumferential Averaged Radial Velocity at 2700 rpm, 737 cfm, 100 mm Downstream
Figure 5.15  Circumferential Averaged Tangential Velocity at 2700 rpm, 737 cfm, 100 mm Downstream

Figure 5.16  Circumferential Averaged Velocity Magnitude at 2700 rpm, 737 cfm, 100 mm Downstream
Figure 5.17  Circumferential Averaged Axial Velocity at 2700 rpm, 800 cfm, 100 mm Downstream

Figure 5.18  Circumferential Averaged Radial Velocity at 2700 rpm, 800 cfm, 100 mm Downstream
Figure 5.19  Circumferential Averaged Tangential Velocity at 2700 rpm, 800 cfm, 100 mm Downstream

Figure 5.20  Circumferential Averaged Velocity Magnitude at 2700 rpm, 800 cfm, 100 mm Downstream
Figure 5.21  Circumferential Averaged Axial Velocity at 2700 rpm, 1327 cfm, 100 mm Downstream

Figure 5.22  Circumferential Averaged Radial Velocity at 2700 rpm, 1327 cfm, 100 mm Downstream
Figure 5.23  Circumferential Averaged Tangential Velocity at 2700 rpm, 1327 cfm, 100 mm Downstream

Figure 5.24  Circumferential Averaged Velocity Magnitude at 2700 rpm, 1327 cfm, 100 mm Downstream
CHAPTER 6  CONCLUSIONS AND RECOMMENDATIONS

At the end of this project, some conclusions and recommendations can be presented.

6.1 Conclusions

Based on the objectives of this project as well as the results and discussions, the following conclusions can be made:

- An X-array hot wire can be utilized to obtain measurements of the three components of the phase averaged velocity downstream of an automotive fan.
- The convergence of the two fan performance curves in Fig. 4.2 indicates that the correlation between pressure rise and the average axial velocity at the fan for any fan rotating speed can be derived if performance curve for one rotating speed is available.
- The circumferential "mixing" of the flow, indicated by the annular velocity contour patterns at 100 mm downstream, suggests that applying the circumferential averaged velocity in the fan model is reasonable.
- From Figs. 5.1 to 5.24 and Table 5.7, we can conclude that the FLUENT fan model predicts the average axial velocity and the velocity magnitude in the wake of the fan reasonably well. It significantly underpredicts the tangential and radial velocity. For low flow rates, if tangential velocity coefficients are applied on the fan boundary, the resulting tangential velocity curve is, although still far away from the experimental tangential velocity curve, much better than the tangential curve without applying tangential velocity coefficients on the fan boundary. At intermediate and high flow rates, the fan model with swirl coefficients give good agreement with the experimental data.
- The CFD radial velocity curves remain poor regardless of whether the radial velocity coefficients are applied or not in the fan model.
- The FLUENT fan model, either with or without swirl added, gives good prediction of the fan performance curve.
• The FLUENT fan model, without swirl added, predicts the axial velocity downstream of the fan reasonably well, but not the radial and tangential components of velocity.

• The FLUENT fan model, with swirl added, gives good results for the axial and tangential velocity components, but severely underestimates the radial component.

6.2 Recommendations for Future Work

As an extension of the work reported in this thesis, the following questions should be investigated:

• Is it possible to acquire velocity field data on planes close to the fan? Would this data provide more accurate radial and tangential components for the fan model polynomials?

• FLUENT also has a “velocity profile” option which can be used instead of the “polynomial” option for the swirl components. This option uses the local (pointwise) values of the radial and tangential velocity in the fan model rather than the circumferentially averaged values used in the “polynomial” option. Would the “velocity profile” option yield a more accurate prediction of the wake region?

• What are the effects of changes to the computational domain and the mesh topology and distribution?

• Can one obtain a correlation between the minimum, maximum or average velocity on downstream planes and the prescribed flow rate, for a given fan?

• Can measurements taken on several planes be interpolated to other planes?
REFERENCES


[26] ICEM CFD Engineering, Berkeley, CA, USA.
APPENDIX A  RESIDUE COMPARISON

In order to determine the appropriate $N$ value in Eqs. (4.4) and (4.5), the sum of squares of residues (SSR) were compared. The definition of the sum of squares of residues is:

$$SSR = \sum_{i=1}^{M} (f(r_i) - v_i)^2$$  \hspace{1cm} (A.1)

where $M$ is the total number of radial positions, $f$ is the curve fit polynomial function, $r_i$ is the radial distance from the axis of the hub and $v_i$ is the experimental radial or tangential velocity. The $Nth$ degree polynomial curve fit that has the smallest sum of squares of residues should be used to generate the polynomials for the fan model. Tables A.1 and A.2 show the sum of squares of residues for radial velocity and tangential velocity respectively, from $N = 3$ to $N = 6$.

<table>
<thead>
<tr>
<th>$N$</th>
<th>2300 rpm</th>
<th>2700 rpm</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>635 cfm</td>
<td>800 cfm</td>
</tr>
<tr>
<td>3</td>
<td>1.091</td>
<td>0.875</td>
</tr>
<tr>
<td>4</td>
<td>0.915</td>
<td>0.839</td>
</tr>
<tr>
<td>5</td>
<td>0.833</td>
<td>0.826</td>
</tr>
<tr>
<td>6</td>
<td>0.332</td>
<td>0.481</td>
</tr>
</tbody>
</table>

Table A.1  Sum of Squares of Residues for Radial Velocity Polynomial Curve Fit

<table>
<thead>
<tr>
<th>$N$</th>
<th>2300 rpm</th>
<th>2700 rpm</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>635 cfm</td>
<td>800 cfm</td>
</tr>
<tr>
<td>3</td>
<td>1.162</td>
<td>1.436</td>
</tr>
<tr>
<td>4</td>
<td>0.498</td>
<td>0.576</td>
</tr>
<tr>
<td>5</td>
<td>0.315</td>
<td>0.282</td>
</tr>
<tr>
<td>6</td>
<td>0.18</td>
<td>0.232</td>
</tr>
</tbody>
</table>

Table A.2  Sum of Squares of Residues for Tangential Velocity Polynomial Curve Fit

From the values in these tables, we can conclude that in all cases, for both the radial and tangential velocities, the sixth degree polynomial curve fit results in the smallest sum of squares of residues. This can be further confirmed by examining the polynomials obtained for various $N$. For example, Figs. A.1 to A.3 show the radial velocity
polynomial curve fit at 2300 rpm, 635 cfm and 25 mm downstream, for \( N = 3 \) to \( N = 5 \) (\( N = 6 \) is shown in Fig. 4.6). The comparison of these figures clearly indicates that the sixth degree polynomial curve fit is the most appropriate.
Figure A.1  Radial Velocity Polynomial Curve Fit at 2300 rpm, 635 cfm, 25 mm Downstream \((N = 3)\)

![Graph 1](image)

Figure A.2  Radial Velocity Polynomial Curve Fit at 2300 rpm, 635 cfm, 25 mm Downstream \((N = 4)\)

![Graph 2](image)
Figure A.3  Radial Velocity Polynomial Curve Fit at 2300 rpm, 635 cfm, 25 mm Downstream (N = 5)
VITA AUCTORIS

Ligong Yang was born in 1972 in Mianchi, China. He graduated from No.1 High School of Dengzhou in 1989. From there he went to Nanjing University of Science and Technology where he obtained a B. Sc. in Mechanical Engineering in 1993. He is currently a candidate for the Master of Applied Science degree in Mechanical Engineering at the University of Windsor, Windsor, Ontario.