1

PM-1 3 1/2 x 4 PHOTOGRAPHIC MICROCOPY TARGET
NBS 1010a ANSI/ISO #2 EQUIVALENT

PRECISIONSM RESOLUTION TARGETS

**Computer Prediction of Squirrel-Cage Type
Centrifugal Fan Performance and
Behaviour**

by

James Clark Thompson, B.Eng. (Aerospace)

A thesis submitted to the
Faculty of Graduate Studies and Research
in partial fulfilment of
the requirements for the degree of

Master of Engineering (Aerospace)

Ottawa-Carleton Institute for
Mechanical & Aerospace Engineering
Carleton University
Ottawa, Ontario
Canada

© copyright 1996, J. Thompson

National Library
of Canada

Acquisitions and
Bibliographic Services Branch

395, Queen Street
Ottawa, Ontario
K1A 0N4

Bibliothèque nationale
du Canada

Direction des acquisitions et
des services bibliographiques

395, rue Wellington
Ottawa (Ontario)
K1A 0N4

0-612-19508-2

0-612-19508-2

The author has granted an irrevocable non-exclusive licence allowing the National Library of Canada to reproduce, loan, distribute or sell copies of his/her thesis by any means and in any form or format, making this thesis available to interested persons.

L'auteur a accordé une licence irrévocable et non exclusive permettant à la Bibliothèque nationale du Canada de reproduire, prêter, distribuer ou vendre des copies de sa thèse de quelque manière et sous quelque forme que ce soit pour mettre des exemplaires de cette thèse à la disposition des personnes intéressées.

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

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

ISBN 0-612-19508-2

Canada

Name James Thompson

Dissertation Abstracts International is arranged by broad, general subject categories. Please select the one subject which most nearly describes the content of your dissertation. Enter the corresponding four-digit code in the spaces provided

Aerospace

SUBJECT TERM

U.M.I.

SUBJECT CODE

Subject Categories

THE HUMANITIES AND SOCIAL SCIENCES

COMMUNICATIONS AND THE ARTS

Architecture 0729
Art History 0377
Cinema 0900
Dance 0370
Fine Arts 0357
Information Science 0723
Journalism 0391
Library Science 0399
Mass Communications 0708
Music 0413
Speech Communication 0459
Theater 0465

EDUCATION

General 0515
Administration 0514
Adult and Continuing 0516
Agricultural 0517
Art 0273
Bilingual and Multicultural 0282
Business 0688
Community College 0275
Curriculum and Instruction 0727
Early Childhood 0518
Elementary 0524
Finance 0277
Guidance and Counseling 0519
Health 0680
Higher 0745
History of 0520
Home Economics 0278
Industrial 0521
Language and Literature 0279
Mathematics 0280
Music 0522
Philosophy of 0998
Physical 0523

Psychology 0525
Reading 0535
Religious 0527
Sciences 0714
Secondary 0533
Social Sciences 0534
Sociology of 0340
Special 0529
Teacher Training 0530
Technology 0710
Tests and Measurements 0288
Vocational 0747

LANGUAGE, LITERATURE AND LINGUISTICS

Language 0679
General 0289
Ancient 0290
Linguistics 0291
Modern 0401
Literature 0294
General 0295
Classical 0297
Comparative 0298
Medieval 0316
Modern 0591
African 0305
American 0352
Asian 0355
Canadian (English) 0593
Canadian (French) 0311
English 0312
Germanic 0315
Latin American 0313
Middle Eastern 0314
Romance 0370
Slavic and East European 0372

PHILOSOPHY, RELIGION AND THEOLOGY

Philosophy 0422
Religion 0318
General 0321
Biblical Studies 0319
Clergy 0320
History of 0322
Philosophy of 0469
Theology 0323

SOCIAL SCIENCES

American Studies 0323
Anthropology 0324
Archaeology 0326
Cultural 0327
Physical 0310
Business Administration 0272
General 0770
Accounting 0454
Banking 0338
Management 0385
Marketing 0501
Canadian Studies 0503
Economics 0505
General 0508
Agricultural 0509
Commerce-Business 0510
Finance 0511
History 0358
Labor 0366
Theory 0351
Folklore 0578
Geography 0579
Gerontology 0580
History 0581

Ancient 0579
Medieval 0581
Modern 0582
Black 0328
African 0331
Asia, Australia and Oceania 0332
Canadian 0334
European 0335
Latin American 0336
Middle Eastern 0337
United States 0585
History of Science 0398
Law 0615
Political Science 0616
General 0617
International Law and Relations 0618
Public Administration 0814
Recreation 0452
Social Work 0626
Sociology 0627
Criminology and Penology 0938
Demography 0631
Ethnic and Racial Studies 0628
Individual and Family Studies 0629
Industrial and Labor Relations 0630
Public and Social Welfare 0700
Social Structure and Development 0344
Theory and Methods 0709
Transportation 0999
Urban and Regional Planning 0451
Women's Studies

THE SCIENCES AND ENGINEERING

BIOLOGICAL SCIENCES

Agriculture 0473
General 0285
Agronomy 0475
Animal Culture and Nutrition 0476
Animal Pathology 0359
Food Science and Technology 0478
Forestry and Wildlife 0479
Plant Culture 0480
Plant Pathology 0817
Plant Physiology 0777
Range Management 0746
Wood Technology 0306
Biology 0267
General 0308
Anatomy 0309
Biostatistics 0379
Botany 0329
Cell 0353
Ecology 0369
Entomology 0793
Genetics 0410
Limnology 0307
Microbiology 0317
Molecular 0416
Neuroscience 0433
Oceanography 0821
Physiology 0778
Radiation 0472
Veterinary Science 0786
Zoology 0760

EARTH SCIENCES

Biogeochemistry 0425
Geochemistry 0996

Geodesy 0370
Geology 0372
Geophysics 0373
Hydrology 0388
Mineralogy 0411
Paleobotany 0345
Paleoecology 0426
Paleontology 0418
Paleozoology 0985
Polymology 0427
Physical Geography 0368
Physical Oceanography 0415

HEALTH AND ENVIRONMENTAL SCIENCES

Environmental Sciences 0768
Health Sciences 0566
General 0300
Audiology 0567
Chemotherapy 0350
Dentistry 0769
Education 0758
Hospital Management 0982
Human Development 0564
Immunology 0347
Medicine and Surgery 0569
Mental Health 0570
Nursing 0380
Nutrition 0354
Obstetrics and Gynecology 0381
Occupational Health and Therapy 0571
Ophthalmology 0419
Pathology 0572
Pharmacology 0382
Pharmacy 0573
Physical Therapy 0574
Public Health 0575
Radiology 0576
Recreation 0577

Speech Pathology 0460
Toxicology 0383
Home Economics 0386

PHYSICAL SCIENCES

Pure Sciences 0485
Chemistry 0749
General 0486
Agricultural 0487
Analytical 0488
Biochemistry 0738
Inorganic 0490
Nuclear 0491
Organic 0494
Pharmaceutical 0495
Physical 0754
Polymer 0405
Radiation 0605
Mathematics 0986
Physics 0606
General 0608
Acoustics 0748
Astronomy and Astrophysics 0607
Atmospheric Science 0798
Atomic 0759
Electronics and Electricity 0609
Elementary Particles and High Energy 0610
Fluid and Plasma 0752
Molecular 0756
Nuclear 0611
Optics 0463
Radiation 0346
Solid State 0984
Statistics

Engineering 0537
General 0538
Aerospace 0539
Agricultural 0540
Automotive 0541
Biomedical 0542
Chemical 0543
Civil 0544
Electronics and Electrical 0348
Heat and Thermodynamics 0545
Hydraulic 0546
Industrial 0547
Marine 0794
Materials Science 0548
Mechanical 0743
Metallurgy 0551
Mining 0552
Nuclear 0549
Packaging 0765
Petroleum 0554
Sanitary and Municipal System Science 0796
Geotechnology 0428
Operations Research 0796
Plastics Technology 0795
Textile Technology 0994

PSYCHOLOGY

General 0621
Behavioral 0384
Clinical 0622
Developmental 0620
Experimental 0623
Industrial 0624
Personality 0625
Physiological 0989
Psychobiology 0349
Psychometrics 0632
Social 0451

The undersigned hereby recommend to the
Faculty of Graduate Studies and Research
acceptance of the thesis,

**Computer Prediction of Squirrel-Cage Type
Centrifugal Fan Performance and
Behaviour**

submitted by

James Clark Thompson, B.Eng. (Aerospace)

in partial fulfilment of the requirements
for the degree of Master of Engineering (Aerospace)

Chair, Department of Mechanical & Aerospace Engineering

Thesis Supervisor

Carleton University

May 1996

ABSTRACT

This thesis describes the development of a computer simulation for the purpose of predicting the performance of squirrel-cage type centrifugal fans at design point (ie. peak efficiency) and larger flow rates. A series of assumptions are made in order to simplify the models of the complex flows that exist inside this type of fan. Correlations, together with velocity triangle relations, were used in calculating rises in total and static pressure across the fan's rotor. The results generated using the computer simulation are compared to existing experimental data. The models examined in this thesis produce good predictions of fan performance at design point and higher flow rates for some fan geometries, and very poorly for others. Several explanations for this behaviour were found. The sensitivity of the predictions to all the parameters involved in the flow models was examined.

Much work remains in order for this simulation to accurately predict squirrel-cage fan performance. The simulation program was developed with the intention of modifying it for use in predicting the performance of fans with reverse flow present. The changes required to adapt the present program to simulate reverse flow are outlined at the end of this thesis. Once reverse flow is modelled, the simulation should be capable of predicting fan performance at below design point flow rates where reverse flow exists.

ACKNOWLEDGMENTS

I would very much like to thank my thesis advisor Dr. Richard J. Kind for his support and insight during the past two years. This thesis would not be possible without his guidance. Thanks also go to Dr. S.A. Sjolander for his aid in the post-defense revision of this final thesis draft. I finally wish to thank the Department of Mechanical and Aerospace Engineering at Carleton University for funding my research and studies.

TABLE OF CONTENTS

	Page
THESIS ACCEPTANCE FORM	<i>ii</i>
ABSTRACT	<i>iii</i>
ACKNOWLEDGEMENTS	<i>iv</i>
TABLE OF CONTENTS	<i>v</i>
LIST OF FIGURES	<i>viii</i>
NOMENCLATURE	<i>vi</i>
1.0 INTRODUCTION	
1.1 Objective	1
1.2 The "Squirrel-Cage" Type Fan	1
1.3 Scope of Work	3
2.0 PREVIOUS RELATED WORK	
2.1 Experimental Observations of Fan Flow	5
2.2 Computer Simulation of Fan Performance	9
2.3 Water Cascade Experiments	11
3.0 FAN FLOW MODEL	
3.1 Inlet Region	15
3.2 Blading Region	17
3.3 Volute Region	23

3.4 Blading/Volute Interface	26
3.4.1 Suddenly Enlarged Flow Model	27
3.4.2 Angular Momentum Conserving Model	29
4.0 THE COMPUTER PROGRAM - FANSIM	
4.1 Overview	31
4.2 Data Input	33
4.3 Geometry Initialization	33
4.4 Blading Performance Calculations	35
4.5 Main Solution Loop	38
4.6 Solution Output	41
5.0 COMPUTER PROGRAM PERFORMANCE	
5.1 Blading Routine	42
5.2 Main Solution Loop	43
6.0 COMPUTER PROGRAM BEHAVIOUR	
6.1 Blading Models	50
6.2 Flow Model Sensitivity	51
6.2.1 Flow Loss Parameter Sensitivity	52
6.2.2 Simulation Running Parameter Sensitivity	56
6.2.3 Fan Geometry Sensitivity	58
7.0 FUTURE WORK	
7.1 Current Simulation	62
7.2 Modelling Reverse Flow	66

8.0 CONCLUSIONS	70
REFERENCES	72
FIGURES	74
APPENDIX A - INPUT DATA FILES	117
APPENDIX B - PROGRAM CODE	119
APPENDIX C - SAMPLE OUTPUT	141

LIST OF FIGURES

Figure	Description	Page
1.1	Geometry of the "Squirrel-Cage" Fan	75
1.2	Rotor Geometry Detail	76
2.1	Experimental Fan Test Geometries	77
2.2	Spiralling Secondary Flow in Volute	78
2.3	Visualization of Flow Patterns in Fan	79
2.4	Typical Variation in Volute Pressure with Location and Flow Rate	80
2.5	Measured Pressure Distribution Around Fan Operating at Below Design-Point Flow Rate	81
2.6	Reverse Flow Behaviour in Squirrel-Cage Fans	82
2.7	Effect of Blade Sweep on Typical Squirrel-Cage Fan Performance	83
2.8	Kind & Tobin (1990) Measured Fan Performance	84
3.1	Geometric Subdivision of Fan	85
3.2	Imposed Axial Flow Rate Distributions	86
3.3	Schematic of Typical Blading Velocity Triangles	87
3.4	Volute Sector Control Volume	88
3.5	The Sudden Enlargement Flow	89
4.1	Simulation Flow Diagram	90
4.2	FANSIM Main Program Arrays	91
4.3	FANSIM Code Layout	92

Figure	Description	Page
4.4	Main Solution Code Flow Chart	93
5.1	Least Squared Error Curvefit Results	94
5.2	Fan Performance Results for Rotor1 Casing1 Using the Two Compatibility Models	95
5.3	Fan Performance Results for Rotor1 Casing2 Using the Two Compatibility Models	96
5.4	Fan Performance Results for Rotor2 Casing1 Using the Two Compatibility Models	97
5.5	Simulation Data Used In Accuracy Assessment	98
5.6	Compatibility Model Prediction of Volute Pressure Distribution	99
6.1	Effect of Deviation on Blading Flow Turning	100
6.2	Effect of Blading Loss on Rotor Total Pressure Rise	101
6.3	Effect of Blading Loss Models on Performance Simulation of Rotor1 Casing1	102
6.4	Effect of Blading Loss Models on Performance Simulation of Rotor2 Casing1	103
6.5	Effect of Rotor Leakage on Simulation of Rotor1 Casing1 Using Uniform Flow Model	104
6.6	Effect of Compatibility Coefficient on Simulation of Rotor1 Casing1 Using Sudden Enlargement Model	105
6.7	Effect of Volute Wall Friction on Simulation of Rotor1 Casing1 Using Uniform Flow Model	106
6.8	Effect of Gap Dynamic Pressure Loss on Simulation of Rotor1 Casing1 Using Uniform Flow Model	107
6.9	Effect of Inlet Loss on Simulation of Rotor1 Casing1 Using Uniform Flow Model	108

Figure	Description	Page
6.10	Effect of Number of Sectors on Simulation of Rotor1 Casing1	109
6.11	Effect of Inlet Axial Flow Distribution on Simulation of Rotor1 Casing1 Using Uniform Flow Model	110
6.12	Effect of Blade Exit Angle on Simulation of Rotor1 Casing1 Using Uniform Flow Model	111
6.13	Effect of Gap Size on Simulated Fan Performance Using Uniform Flow Model	112
6.14	Effect of Casing Exit Area on Simulated Fan Performance Using Uniform Flow Model	113
6.15	Effect of Blade Chord on Simulated Fan Performance Using Uniform Flow Model	114
6.16	Effect of Rotor Exit Area/Fan Inlet Area on Simulated Fan Performance Using Uniform Flow Model	115
6.17	Summary of Fan Geometry's Effect on Simulated Performance	116

NOMENCLATURE

Symbol	Description
A	area
B_B	fan rotor width
$B_B(i)$	blading region slice width 'i'
B_V	fan casing width
c	blade chord, (eqn 3.5d, pg. 19)
C	absolute flow velocity
C_w	absolute whirl velocity component
C_r	absolute radial velocity component
C_{diff}	coefficient of relative blading flow diffusion, (eqn 2.3a, pg. 13)
i	blading inlet flow incidence angle, $i = \beta_1 - \beta_2$
K_d	blading diffusion loss coefficient, (eqn 2.3a-c, pg. 13)
K_{exp}	sudden enlargement model arbitrary modifier, for fine tuning model
K_{ext}	fan overall efficiency factor for mechanical and windage losses
K_f	blading surface friction loss coefficient, (eqn 2.2, pg. 13)
K_{frc}	casing wall surface friction coefficient
K_{gap}	dynamic pressure loss coefficient across gap
K_i	blading incidence loss coefficient, (eqn 2.4, pg. 13)
K_{inlt}	fan inlet flow dynamic pressure loss coefficient

K_{leak}	fraction of rotor through flow that recirculates around end shrouds
K_{tot}	overall blading total pressure loss coefficient (eqn 2.1, pg. 12)
numi	number of slices that the fan is to be divided into
numj	number of sectors that the fan is to be divided into
P	static pressure
Pt	total pressure
Q	volume flow rate
R	radius (distance from fan centre)
Re	Reynolds number, $Re = (2U_2 R_{B2})/\nu$
s	blade pitch, (eqn 3.5c, pg. 19)
U	rotor blade velocity
W	relative blading flow velocity
z	number of blades in rotor
α	absolute blading flow angle, see Figure 3.3 for sign convention
β	relative blading flow angle, see Figure 3.3 for sign convention
β'	blade metal angle, see Figure 3.3 for sign convention
Δ	a change or difference
δ	blading exit flow deviation angle, $\delta = \beta_2 - \beta_2'$
η	fan efficiency, $\eta = (\Psi_{\text{net}} \times \Phi)_{\text{fan exit}} / (\Psi_{\text{Euler}} \times \Phi)_{\text{rotor exit}}$
θ	angular fan station, 0° at cut-off, and 360° at fan exit
$\theta_{\text{end}}(j)$	angular fan station of the end plane of sector 'j'
$\theta_{\text{start}}(j)$	angular fan station of the start plane of sector 'j'

ξ	blade stagger angle. $\xi = 0.5(\beta_1' + \beta_2')$
ρ	air density
τ_{wall}	casing wall shear stress due to flow friction
ν	kinematic viscosity of air
Φ	flow coefficient, $\Phi = (\text{volume flow rate})/(4U_2R_{B2}^2)$
ϕ	casing outer wall spiral angle, (eqn 3.15b, pg. 23)
Ψ	total pressure rise coefficient, $\Psi = (\text{total pressure rise})/(\rho U_2^2)$
Ψ_{Euler}	ideal (no losses) rotor total pressure rise coefficient
Ψ_{net}	net (all losses included) total pressure rise coefficient
$ld_{\perp}^w$	the perpendicular distance from a vector in the whirl direction that passes through point 'd', to point 'a'
$ld_{\perp}^r$	same as above, except from a vector in the radial direction

SUBSCRIPTS

1	entry
2	exit
B1	blading element entry
B2	blading element exit
E	no loss or Euler
I1	inlet element entry
I2	inlet element exit
gap	flow properties near cut-off

R	parameter applies to reverse flow case only
SE	sudden enlargement
V1	volute sector entry
V2	volute sector exit
mix	flow mixing between blading and volute flows
a,b,c...i	stations in volute sector control volume, see Figure 3.4

1.0 INTRODUCTION

1.1 OBJECTIVE

The objective of this work was to develop a relatively simple computer program to simulate the performance of "squirrel-cage" type centrifugal fans at design point (i.e. best efficiency) and higher flow rates. This computer program was to be designed such that in the future, it can be easily extended to deal with reverse flow through parts of the fan rotor, so that fan performance at below design point flow rates can also be simulated. The goal was to predict the distribution of pressure and flow rate throughout the fan, as well as the fan's overall total pressure rise and efficiency, over the entire fan operating flow rate range. If accurate computer predictions could be made, the simulation may help to determine how different fan geometries affect fan performance, and thus allow the production of more efficient designs.

1.2 THE "SQUIRREL-CAGE" TYPE FAN

This type of fan (Figure 1.1) is characterised by two main features that distinguish it from other common types of centrifugal fan. First, the rotor exit-to-inlet area ratio is unusually large, typically around 2.5:1. Second, the rotor has many blades of small chord, from around 30 up to as many as 80. This tends to give the rotor a drum or

"squirrel-cage" shape, as illustrated in Figure 1.2. The blades are normally constructed from a single piece of sheet metal, and are formed into a circular arc. Also, forward swept blading is normally used, resulting in a large flow rate and total pressure rise at low rotational speeds. The casing is also typically formed from flat pieces of sheet metal, bent and spot welded to form a spiral shape. This construction method permits these fans to be constructed very quickly and cheaply. The rotor is often driven by an electric motor attached via a belt and pulley on one side.

Squirrel-cage fans are most commonly used in ventilation applications requiring compact size, high flow rate, low cost, and low noise. The use of many forward swept blades in the rotor results in relatively large rises in dynamic and total pressure across the fan at low speeds, and thus low noise. Examples of current uses of these fans include: hair blow dryers, car ventilation fans, commercial and industrial ventilation systems, and residential forced air heating systems. A typical household natural gas/oil furnace blower consumes about 600 Watts of electrical power, or roughly 5% of an average Canadian household's annual electricity bill in a year (Statistics Canada, 1991 and 1995). This adds up to about 1% of Canada's total annual domestic electricity demand being consumed by these types of fans, in household furnaces alone. A comparable, and very likely larger, amount of electricity would be used by blowers of this type in commercial buildings such as shopping malls and office buildings.

The extensive use of these fans in Canada, and the rest of the world, warrants an investigation into their behaviour. It would be desirable to find out what factors affect their performance, and to improve their efficiency while continuing to maintain their simple and inexpensive construction method. Two approaches to this task can be taken: experimental study, and computer simulation. The object of this thesis is the latter.

1.3 SCOPE OF WORK

A computer program has been developed to simulate the performance of squirrel-cage fans at design point and higher fan flow rates. This is done by dividing the fan up into three smaller, distinct flow regions: inlet, blading, and volute. Each region has been modelled using steady incompressible-flow control volume analysis, with only the flow parameters at the interfaces between regions being calculated.

Squirrel-cage fans are characterised by complex flow patterns, as will be discussed later in Chapter 2.0. One of the most significant flow patterns is the presence of reverse flow (see section 2.1) at low flow rates. Since flow parameters on either side of the rotor blading are required to predict reverse flow, an entirely forward-flow situation was modelled first. Once a forward-flow solution is found, occurrence of reverse flow can be predicted and the reverse flow parameters can then be calculated. The prediction of reverse flow and its calculation, has been left for future research. However, with the intent of future modification of this simulation to account for reverse flow, some of the

required work has already been performed.

The remainder of this thesis presents the aerodynamic models used for each region of the fan, as well as the FORTRAN computer code developed. The accuracy of the various models is determined by comparing their predictions to experimental data for the performance of several test fan geometries. Chapter 2 presents previous research on topics related to squirrel-cage fans; Chapter 3 describes the aerodynamic models used for each of the three regions of the fan, as well as the fan discretization scheme used; Chapter 4 outlines the computer program and the algorithms used to implement the aerodynamic models developed; Chapter 5 compares the simulation's predicted fan performance results to existing experimental data, and discusses any observed inaccuracies; Chapter 6 explores the program's behaviour, such as sensitivity to a range of loss and fan geometry parameters; Chapter 7 outlines future work that remains to be done on this simulation, and experimental work that could be pursued regarding these fans; and finally Chapter 8 summarizes the work done in this thesis, and draws conclusions. The appendices contain print-outs related to the computer code, specifically: explanation of the three data input files, the computer code print-out, and sample simulation output files.

2.0 PREVIOUS RELATED WORK

Little research has been done to determine how squirrel-cage fans operate, or how the complex flow patterns inside these fans affect performance. This chapter discusses the research that has been done, and its bearing on the work performed in this thesis.

2.1 EXPERIMENTAL OBSERVATIONS OF FAN FLOW

Two experimental investigations of the flow inside squirrel-cage type centrifugal fans were found in literature: that of Raj and Swim (1981) and of Kind and Tobin (1990). Raj and Swim concentrated their measurements on the fluctuation of flow velocity at the rotor exit, but did perform some volute static pressure measurements and flow visualization. They tested a single fan configuration at 100% and 150% of its design flow rate of 354 litre/second, flow coefficients of about 0.4 and 0.6 respectively. The geometry of the fan tested is typical of squirrel-cage fans, and is listed in Figure 2.1.

The flow visualization tests performed by Raj and Swim (1981) identified some important characteristics of the flow inside squirrel-cage fans. They cite five main observations in their paper: flow entered the fan axially, and gradually turned radially by the time it got to the back of the rotor; the rotor inlet flow did not fill the entire rotor width, leaving a separated region at the front; the inlet flow entered the blade passage at

a very large angle of attack, resulting in suction surface flow separation; there was very little flow through the rotor in the separated region at the front; and, the rotor exit flow was found to have very little axial velocity component.

Kind and Tobin's (1990) paper describes an experimental investigation of the flow behaviour within squirrel-cage fans. A review of existing literature on the design and performance of these fans, indicates that the majority of research was focused on the effect of using various sizes and styles of rotor and volute. In order to make their pressure and velocity measurements around the rotor, Kind and Tobin used a five hole probe. In addition, flow visualization was performed to obtain a qualitative picture of the flow patterns in squirrel-cage fans at, above, and below the fan design-point (maximum efficiency) flow rate. Three different fan configurations were used in their experiment, the geometric data for which can be found in Figure 2.1. A rotor with forward swept blading was tested inside two different casings, and a second rotor with radial exit blading was tested in the first of the two casings.

Kind and Tobin (1990) also observed the flow characteristics cited by Raj and Swim (1981), noting that the flow inside a squirrel-cage fan is very complex, and is influenced by a variety of secondary flows. One of the major flow characteristics identified, the separated region at the front of the rotor (see Figure 2.2), was attributed to the large exit-to-inlet area ratio of these fans. The large deceleration required for the inlet flow to fill the entire width of the rotor would result in an adverse pressure gradient.

Since the total pressure must remain approximately uniform in the inlet region, this increase in static pressure is incompatible with the low pressure required to turn the inlet flow from an axial to a radial direction before it enters the rotor. The result is that the fan's flow passes through only the part of the blading closest to the back wall of the rotor. Raj and Swim (1981) observed that the axial size of this separated region was a strong function of circumferential position around the inlet, as well as fan flow rate.

The axial non-uniformity of the flow through the rotor results in a cross-flow within the volute. As shown in Figures 2.2 and 2.3, the high energy rotor outflow curves around towards the separated region at the front of the fan, generating a very pronounced spiralling flow pattern in the volute. Kind and Tobin (1990) suggested that this spiralling secondary flow may have a "mixing out" effect on the mean volute flow, resulting in an approximately uniform total pressure distribution in the volute in radial planes. Preliminary measurements (Kind, 1996) of total pressure in the volute provide some support for this assumption.

Another major flow characteristic identified by Kind and Tobin (1990) was the development of reverse flow at fan flow rates below design point, a result of circumferential pressure gradients in the volute. Many researchers have identified the existence of a static pressure gradient around the circumference of the volute at off-design flow rates. Figure 2.4, from Eck (1973), illustrates how the pressure distribution around the volute behaves in a typical squirrel-cage fan at various flow rates. The sudden change

in pressure near the 360° station is attributed to the effects of the casing cut-off. Similar pressure distributions are observed in other centrifugal machinery such as pumps and centrifugal blowers. Figure 2.5 shows the static pressure distribution in the volute of an eight bladed centrifugal blower operating at a below design point flow rate, as measured by Alden and Kane (1970).

The observed pressure gradient is due to the blading exit flow having to accelerate or decelerate to fill the volute. At lower flow rates, the flow from the rotor decelerates to fill the volute which has been designed for a larger flow. Kind and Tobin (1990) stated that the corresponding rise in static pressure around the circumference of the fan tends to push flow near the exit back through the rotor towards the low pressure region at the beginning of the volute (see Figure 2.6). The percentage of the fan rotor filled by this "reversed" flow tends to become larger for progressively lower fan flow rates. The reverse flow phenomenon has been observed by other researchers in physically similar turbomachinery like centrifugal pumps (Lorett & Gopalakrishnan, 1986). Cross-flow fans, which use rotors identical to those used by squirrel-cage fans, rely on the existence of a flow passing through the rotor twice, similar to the aforementioned reverse flow.

This flow behaviour is not necessarily undesirable, since the rotor blockage due to the reverse flow forces the flow entering the fan to pass through a smaller area of rotor, at consequently higher velocities. The result is a larger total pressure rise across the fan than would be expected if there was not reverse flow. An example of this effect

can be seen in Kind & Tobin's (1990) measured fan performance data (Figure 2.8), by the increase in Ψ at flow coefficient values below 0.12. The reason why the increase in total pressure rise at low flow rates was not observed for the fan using radial exit blading is unclear. It is possible that rotors with forward swept blades, because the majority of their total pressure rise is in the form of dynamic pressure, tend to "benefit" more from reverse flow than radial or backward swept rotors. However, since the radial exit bladed rotor was not tested with its optimum casing, a conclusive explanation can not be given based on existing experimental data. The overall fan performance measurements made by Kind and Tobin (1990) are consistent with those of other researchers, such as Osbourne (1977). Osbourne's values for typical squirrel-cage fan performance are shown in Figure 2.7.

An obvious disadvantage of the reverse flow, is that fan rotor blading is not normally optimized for it. Incidence losses in the blade row would be high, and the almost certain existence of flow separation would make diffusion losses very large as well. Although the regions of the rotor with forward flow may be operating at good efficiency, the drastic reduction in efficiency in the reverse flow regions normally results in an overall reduction in fan efficiency.

2.2 COMPUTER SIMULATION OF FAN PERFORMANCE

There has been a lot of study dedicated to the computer simulation of turbomachines. With the increased availability of powerful computers, complex Navier-

Stokes equation solvers have been becoming more popular. However, it was desired in this thesis, that any computer simulation developed be simple to code, and use. For this reason, finite difference and similar numerically intensive solution techniques were not considered. Elder and Gill (1985) developed a simpler analysis method in their study of centrifugal compressor surge. They divided a compressor stage into a sequence of one dimensional flow regions. By calculating the flow in each region (inducer, impeller exit, vaneless space, etc.) separately, they were able to successfully simulate the compressor stage, and predict the occurrence of surge quite well. A similar analysis method was used by Lorett and Gopalakrishnan (1986) in their study of centrifugal pump impeller-volute interaction.

Kind (1985) proposed an analysis method for squirrel-cage fans based on the successful work by Elder and Gill (1985) and Lorett and Gopalakrishnan (1986). Sagi (1990) developed a computer simulation program as the basis of his Master's thesis, that attempted to implement the flow model proposed by Kind. The simulation was designed to calculate pressure and velocity distributions around the fan, and overall fan performance, at flow rates at and above design point (best efficiency). The program did not address the existence of reverse flow at below design point flow rates. Slip correlations found in the literature were used to predict rotor flow turning. Tentative correlations were proposed to account for blading losses. These correlations, in conjunction with steady flow control volume analysis, made it possible to determine the pressure and velocity through each region of the fan: inlet, blading, and volute. By

dividing the fan up into sectors and slices (Figure 3.1). Sagi attempted to solve for the entire fan flow using a stepwise procedure. An initial selection of the blading flow rate and gap flow rate at the beginning of the fan (near cut-off) started the calculations. Use of the equations of motion (discussed in Chapter 3.0) allowed flow parameters at successive sectors to be found. Sagi's program involved several iteration loops, some of which were not actually necessary. Unfortunately his program failed to converge properly. The results presented in his thesis were subsequently found to be the product of calculations which had "hung-up" at non-converged conditions. Also, some blade angles had been input with incorrect signs.

The original objective of the present thesis was to modify Sagi's program to model reverse flow. However, since his program would require major work to correct before an attempt could be made to extend it to handle reverse flow, it was decided that a new program would be developed instead. The basic discretization scheme outlined by Kind (1985) and used by Sagi was used for the new program.

2.3 WATER CASCADE EXPERIMENTS

Kind (1996) reports the results of water cascade experiments performed on a cascade of typical sheet metal squirrel-cage fan blades. The blades tested were almost identical to those in the two rotors used by Kind & Tobin (1990), sheet metal circular arc blades with 60° camber (blades from rotor 1 had 65° camber) and a chord of 28mm. By

varying the inlet flow angle β_1 , and the blade row stagger angles ξ , the blade cascade was tested over the full possible range of flow incidence angles i . The result was a number of empirical correlations for the losses across thin sheet metal blades at a range of incidences and pitch-to-chord ratios. The overall loss coefficient across the cascade was defined as:

$$K_{tot} = \frac{\Delta P_{t_{loss}}}{0.5 \rho W_1^2} \quad (2.1)$$

where $\Delta P_{t_{loss}}$ is the loss in total pressure across the blade row, and W_1 is the relative blading inlet flow velocity. Kind (1996) broke the overall loss down into three separate types: friction K_f , diffusion K_d , and incidence K_i losses. The friction term accounts for losses due to blade passage surface boundary layers, ie. skin friction. Diffusion refers to losses resulting from both relative flow deceleration in the blade passage, and flow separations along the blade surfaces. Note that it is possible to have non-zero diffusion losses even when there is an overall acceleration in the blade passage because separated flow regions can still exist. In fact, the simple, uncountoured shape of the blades result in separated flow over much of the fan's operating range. Finally, incidence losses account for inlet flow that does not flow smoothly into the cascade at the inlet blade metal angle, resulting in leading edge separations. The correlations developed from the cascade tests are:

$$K_f = 0.005 \left(\frac{c}{s} \right) \quad (2.2)$$

$$C_{diff} = 1 - \left(\frac{W_2}{W_1} \right)^2 \quad (2.3a)$$

$$K_d = \left[\frac{C_{diff} + 2}{4} \right]^{1.7} \left(\frac{c}{s} \right)^{0.7} \quad \text{for } C_{diff} > -2 \quad (2.3b)$$

$$K_d = 0.0 \quad \text{for } C_{diff} \leq -2 \quad (2.3c)$$

$$K_i = 0.35(0.01i)^2 \left[1 + 12 \left(\frac{s}{c} - 0.85 \right)^2 \right] \quad (2.4)$$

C_{diff} is a non-dimensional measure of the amount of diffusion in the blade passage. The incidence angle, i , is defined in Figure 3.3. In addition to blading total pressure loss correlations, a correlation for blading exit flow deviation was found:

$$\beta_2 = \beta_2' + \delta \quad (2.5)$$

$$\delta = 15 \left(2 \frac{s}{c} \right)^{0.6} - 0.13i + 0.0007i^2 \quad (2.6)$$

where δ is the deviation angle, β_2 is the exit flow angle, and β_2' is the exit blade angle, as defined in Figure 3.3.

3.0 FAN FLOW MODEL

The success of any computer simulation developed depends on properly modelling the physics of the squirrel-cage fan's flow. The models used by the computer simulation in this thesis, and the many assumptions made regarding these models, are described in this chapter.

The analysis of the flow within the entire squirrel-cage fan has been simplified by dividing it up into three primary flow regions, as proposed by Kind (1985): inlet, blading, and volute (see Figure 3.1). The inlet region includes all parts of the fan inlet, from the eye up to the inlet surface of the rotor. The blading region includes only the blading itself, and not the shrouds at either end of the rotor. Finally, the volute region refers to the rectangular sectioned, spiral duct formed by the casing outer walls, and the rotor exit surface. The casing outer wall is assumed to follow an arithmetic spiral from the cut-off, the 0° location (or the start of the volute), 360° around to the fan exit. Note that a positive location angle around the fan circumference is assumed to be in the direction of the rotor's rotation.

To further simplify the analysis, each major flow region was divided into physical sub-regions. The first subdivision consists of a series of sectors around the fan circumference. The inlet and blading region sectors are further subdivided into slices in

the axial direction, as shown in Figure 3.1. Due to the strong cross flow (Figures 2.2 and 2.3) observed in the volute, only sectors were used in this region. The number and size of each of these physical sector/slice subdivisions, hereafter referred to as elements, can be varied to concentrate the simulation in key fan areas. Also, to aid in analysis of program results, and to limit flow variables to a small range of values, non-dimensional quantities are used throughout the simulation program. The non-dimensionalizing parameters used in the computer program were as follows:

$$\text{lengths} = 2R_{B2} \quad (3.1a)$$

$$\text{velocities} = U_2 \quad (3.1b)$$

$$\text{pressures} = \rho U_2^2 \quad (3.1c)$$

$$\text{volume flow rates} = 4U_2 R_{B2}^2 \quad (3.1d)$$

where R_{B2} is the outer radius of the rotor, ρ is the density of air, and U_2 is the rotor blading tip speed. Note that the equations in the remainder of this thesis are presented in dimensional form.

3.1 INLET REGION

At inlet, the flow in from the environment is assumed to be completely axial, and to have a total pressure of zero relative to the ambient (atmospheric) pressure. Losses in the inlet region were scaled on the dynamic pressure in the inlet, making the total

pressure (gage) leaving the inlet region:

$$P_{tI} = -0.5\rho K_{inI} \left(\frac{Q_{I1}}{A_{I1}} \right)^2 \quad (3.2)$$

where Q_{I1} is the volume flow rate into the inlet region, A_{I1} is the area of the fan inlet, and K_{inI} is a loss coefficient. Osbourne (1966) suggests rather large values for K_{inI} of between 0.5 and 0.8, to account for losses in the inlet eye of the fan, and the turning of the flow into the radial direction before exiting the inlet region. Kind (1996) suggested a more moderate loss of 0.1, based on the analysis of flow through rounded inlets such as found in White (1986, page 335). For the case of no reverse flow, the total pressure could be assumed to be constant throughout the inlet region. This assumption is consistent with Kind and Tobin's (1990) findings.

As discussed in section 2.1, the flow entering the fan does not fill the entire width of the rotor. Raj and Swim (1981) found the amount of rotor width that was filled varied with circumferential location around the fan, and fan flow rate. In order to simplify the analysis of flow in this region, these variations were neglected in favour of a more symmetric inlet flow distribution, such as suggested by Gessner (1967). Thus, axial variations in rotor through flow were attained by using a selection of imposed flow rate distributions (Figure 3.2), always with the maximum flow rate occurring at the back of the rotor.

The inlet flow was also assumed to turn into a completely radial direction before it leaves the inlet region. Any losses incurred during this turning were accounted for by the value of K_{inlet} . Once the inlet element exit flow rate Q_{12} had been found from the imposed distributions, the static pressure and flow velocity leaving each inlet element could be found using continuity and Bernoulli's relationship.

3.2 BLADING REGION

Blading entry flow parameters such as static pressure and velocity, were obtained from the exit flow calculations in the adjoining inlet elements. A standard velocity triangle analysis of the rotor blade row was used to calculate the values of pressure and velocity exiting the blading. A typical forward swept blade row, and its corresponding velocity triangles are shown in Figure 3.3. The introduction of reverse flow changes the signs of some terms in the analysis, but the Euler pump and turbine equation still applies. The ideal (Euler) rise in total pressure across the blade row is:

$$\Delta Pt_E = \rho(U_2 Cw_2 - U_1 Cw_1) \quad (3.3)$$

To calculate Cw_1 and Cw_2 , the two relative flow angles β_1 and β_2 were required. β_1 can be found simply from the velocity triangles using trigonometry, in terms of given geometric and flow information:

$$\beta_1 = \tan^{-1}\left(\frac{U_1 - C_{w1}}{C_{r1}}\right) \quad \text{forward flow} \quad (3.4a)$$

$$\beta_{1R} = \tan^{-1}\left(\frac{U_2 - C_{w1R}}{C_{r1R}}\right) \quad \text{reverse flow} \quad (3.4b)$$

The subscript 'R' above indicates velocity triangle values specific to the reverse flow case. The whirl velocity component in the above two equations was substituted for using $C_{r1}\tan(\alpha_1)$, $C_{r1R}\tan(\alpha_{1R})$ in the reverse flow case. For reasons of consistency, when the above equations were implemented in the simulation code, U_1 and C_{r1} were substituted for using U_2 and C_{r2} . This was accomplished by noting that the inlet/outlet blade velocities, and the radial inlet/outlet velocities are related through the ratio of the rotor entry-to-exit radius ratio.

Finding the value of β_2 was a little less straight forward since the magnitude of the exit flow deviation δ must be found first. Several different methods of estimating the deviation were examined during the development of this simulation, all of them developed for other types of turbomachinery like centrifugal pumps, and axial compressors. One such example is the relatively simple empirical relation proposed by Howell for axial compressor cascades (Dixon, 1989, pg.76). Sagi (1990) included many of these different empirical models in his simulation program. Since his program failed to iterate properly, the applicability of the various correlations he tried to the prediction of squirrel-cage fan rotor exit flow deviation could not be determined. It was decided that Kind's (1996)

correlation for deviation (equation 2.6) would be used for the time being since it is based on blades that are almost identical to those in the three test fan geometries used in this thesis.

The values of Cw_1 and Cw_2 could have been calculated using the absolute flow inlet and outlet angles, α_1 and α_2 , but β_1 and β_2 were needed later to calculate W_1 and W_2 , so it made sense to minimize calculations. With all the terms in equation (3.3) known, the Euler total pressure rise could be calculated for both the forward and reverse flow cases. Although reverse flow was not accounted for in this thesis, allowances for future development of the simulation to include reverse flow have been made, as will be discussed later in Chapter 7.0.

With the ideal total pressure rise calculated, the next step was to determine the magnitude of losses in the blade passages. Kind's (1996) correlations, as presented in section 2.3, provided a means of determining these losses. Other models of blading loss were not examined in detail since: none exist specifically for squirrel-cage fan rotor blading, and the cascade tests performed by Kind were done on almost identical blades to those used to test this simulation's performance.

Kind's (1996) correlations (equations 2.2 to 2.4) are based on the value of pitch to chord ratio s/c , incidence i , and the relative inlet and outlet velocities W_1 and W_2 . The first step was to calculate these parameters. The pitch to chord ratio is based on the

geometry of the blading, which can be found for the rotor's typical circular arc blading using the following series of equations:

$$R_{mean} = \sqrt{(R_{B1}R_{B2})} \quad (3.5a)$$

$$\xi = \frac{\beta'_1 + \beta'_2}{2} \quad (3.5b)$$

$$s = \frac{2\pi R_{mean}}{z} \quad (3.5c)$$

$$c = \frac{R_{B2} - R_{B1}}{\cos(\xi)} \quad (3.5d)$$

$$\text{pitch-to-chord ratio} = s/c \quad (3.5e)$$

In the above equations, R_{mean} is the RMS radius of the blades in the rotor, ξ is the blading stagger angle, s is the blade pitch, z is the number of blades, and c is the blade chord. Definitions of the angles used above are presented in Figure 3.3. The flow incidence angle can be calculated using the relative inlet flow angle β_1 and the inlet blade metal angle β'_1 :

$$i = \beta_1 - \beta'_1 \quad (3.6)$$

The incidence angle is defined here as positive when inlet flow impinges on the pressure surface of the blades. The relative inlet and outlet velocities were determined using the

velocity triangles in Figure 3.3. and are given by:

$$W_1 = \left(\frac{R_{B2}}{R_{B1}} \right) \left(\frac{Cr_2}{\cos(\beta_1)} \right) \quad \text{forward flow} \quad (3.7a)$$

$$W_2 = \frac{Cr_2}{\cos(\beta_2)} \quad \text{forward flow} \quad (3.7b)$$

$$W_{1R} = \left| \frac{Cr_{1R}}{\cos(\beta_{1R})} \right| \quad \text{reverse flow} \quad (3.8a)$$

$$W_{2R} = \left| \left(\frac{R_{B2}}{R_{B1}} \right) \left(\frac{Cr_{1R}}{\cos(\beta_{2R})} \right) \right| \quad \text{reverse flow} \quad (3.8b)$$

Equations (3.7a) to (3.8b) have been written in terms of the radial velocity at the rotor exit radius R_{B2} , as was already mentioned for equations (3.4a) and (3.4b).

With the required parameters now calculated, the correlations described in equations (2.2) through (2.4) could be used to calculate a value for the overall total pressure loss coefficient:

$$K_{tot} = K_f + K_d + K_i \quad (3.9)$$

In addition to the blading losses described by the cascade correlations, an additional overall rotor leakage loss (Stepanoff, 1957) was added, yielding equation (3.11), the final

total pressure loss across the blade row:

$$\Delta P_{t_{loss}} = K_{leak} \Delta P_{t_E} + 0.5 \rho K_{tot} W_1^2 \quad (3.10)$$

The loss due to flow leakage around the ends of the rotor was treated as suggested by Yaras (1993). A percentage (K_{leak}) of the total blading through flow does not enter the volute region, leaking back around the rotor ends, reducing the observed net rotor total pressure rise. The value of K_{leak} was assumed to be less than the 20% observed in centrifugal pumps (Stepanoff, 1957 pg.186) due to the very large ratio between rotor width and the space between the rotor shroud and casing. Therefore, values of between zero and five percent were considered fair estimates.

With all changes in total pressure across the rotor known, the total pressure of the flow exiting each blading element could be calculated as:

$$P_{t_{B2}} = P_{t_{B1}} + \Delta P_{t_E} - \Delta P_{t_{loss}} \quad (3.11)$$

The other flow values required, such as radial and whirl velocity, were found using the blade element flow rate, and the element's velocity triangle. The static pressure at each blade element's exit could now be calculated from Bernoulli's relationship.

3.3 VOLUTE REGION

Flow analysis for each volute sector occurs inside the control volume "abcdefgha", shown in Figure 3.4. Flow conditions entering the volute sector from the adjacent volute sector, as well as the adjoining blading elements are assumed known from previous calculations. Continuity, and conservation of angular momentum was applied to the volute sector in order to find the flow conditions at the exit of the control volume:

$$Q_{v2} = Q_{v1} + Q_{B2} \quad (3.12)$$

$$P_d A_d R_d + P_f (A_h - A_d) R_f - P_h A_h R_h - \text{friction} = \quad (3.13)$$

$$Q(Q_h C_{w_h} R_h - Q_d C_{w_d} R_d - Q_b C_{w_b} R_b)$$

The cross-sectional areas in equation (3.13) are defined in Figure 3.4. The mean volute whirl velocities C_{w_h} and C_{w_d} , were determined using the assumption that:

$$C_{w_v} = \frac{Q_v}{A_v} \quad (3.14)$$

is a good approximation. Assuming the volute radial velocity is related to the whirl velocity such that the volute flow's angle is parallel to the casing wall, then:

$$Cr_v = Cw_v \tan(\phi) \quad (3.15a)$$

$$\phi = \frac{R_{v2} - R_{v1}}{\pi(R_{v1} + R_{v2})} \quad (3.15b)$$

The angle ϕ is the spiral angle of the volute casing's outer wall, and is found using the casing radius at the cut-off R_{v1} , and at the fan exit R_{v2} .

The average of P_e and P_g were used to determine P_f in equation (3.13). The assumption was made that: the volute total pressure at any location around the fan circumference is constant in the radial direction. This assumption was based on the suggestions made by Kind and Tobin (1990), as was discussed in section 2.1. To find the velocity at any radius in the volute, the assumption made by other researchers such as Osbourne (1977, pg.130), that the flow obeys a free vortex relationship was used:

$$Cw_v R_v = \text{constant} \quad (3.16)$$

Therefore, knowing the total pressure along any radial plane (for example, along the volute sector exit plane), and the flow velocity at some radius, the velocity and static pressure could be found at any other radius. In this way, the static pressures at 'e' and 'g' could be found using the total pressure and mean flow velocities at the sector's entry and exit plane.

The friction losses in the volute sector control volume were calculated by

determining an average wall shear stress for each side of the volute:

$$\tau_{wall} = 0.5 \rho K_{fric} C w_{wall}^2 \quad (3.17)$$

The coefficient of friction, K_{fric} , was estimated from the Moody chart to be 0.003, using a typical Reynolds number for this type of fan as suggested by Kind and Tobin (1990):

$$Re = \frac{2U_2 R_{B2}}{\nu} \approx 3 \times 10^5 \quad (3.18)$$

The whirl velocity at point 'i' in Figure 3.4 was used for the sides of the casing, and the whirl velocity at point 'f' was used for the curved end wall. A loss of angular momentum in the control volume due to friction was attained by multiplying each wall's shear stress by its area, and distance of its centroid from the fan centre of rotation.

Since the flow from many rotor slices dumps into any one volute sector, flow rate averaged values of pressure and velocity are used for the blading flow's contribution to equation (3.13). A compatibility condition was imposed between the blading exit flow's static pressure, and the static pressure in the volute at the blading exit radius. Setting the blading exit pressure equal to that in the volute, a "corrected" blading exit whirl velocity was calculated, making sure that the blading flow's total pressure and exit flow angle was conserved. It was assumed that since fan sectors would be kept small in the simulation,

and the fact that volute static pressure does not vary dramatically over short distances, the static pressure at point 'c' (see Figure 3.4) would be a good estimate of the average pressure in the volute at the blading exit. The corrected blading exit whirl velocity, once calculated, was used in equation (3.13).

3.4 BLADING/VOLUTE INTERFACE

One of the most difficult questions that needed to be answered was the determination of the flow exiting each blading element. Kind (1996) suggested that the blading exit flow must go through some sort of deceleration/mixing process as it enters the volute in order to be in equilibrium with the volute flow. The nature of this process for the most part is unknown. At this point, experimental data exists only for: the flow in the inlet region (Gessner, 1967), the velocity distribution across the blading exit passages (Raj and Swim, 1981), and the pressure and velocity distributions around the fan (Kind and Tobin, 1990). Experimental data on the interaction between blading exit flow, and the volute flow does not exist at present, therefore simple analytical models were developed. Three different compatibility models were examined: forced uniform flow, suddenly enlarged flow, and an angular momentum conserving flow.

The first, forced uniform blading sector flow rate, consisted of simply dividing the desired overall fan flow rate evenly between each blade sector, and calculating the blade sector exit conditions using the equations in section 3.2. Then, the resulting volute static

and total pressure distributions are calculated using equation (3.13).

3.4.1 Suddenly Enlarged Flow Model

The sudden enlargement model is based on the concept of flow passing through a sudden increase in area, as illustrated in Figure 3.5. A loss of total pressure and a rise in static pressure results from flow through a sudden enlargement. The loss in total pressure due to a sudden enlargement (Massey, 1980) is:

$$\Delta P_{tSE} = 0.5\rho(C_{B2} - C_b)^2 \quad (3.19)$$

where C_{B2} is the velocity of the flow exiting the blading, and C_b is the velocity of the flow in the volute right at the blading exit, that the blading flow is assume to suddenly decelerate to. When this loss (equation 3.19) is included in the steady flow energy relationship across the enlargement, the result is that the rise in the blading exit flow's static pressure due to the sudden deceleration is:

$$\Delta P_{SE} = \rho C_b(C_{B2} - C_b) \quad (3.20)$$

If the velocity and static pressure of the volute flow adjacent to the rotor exit were calculated from the preceding volute region, then the pressure and velocity of the

compatible blading exit flow could be calculated using equations (3.19) and (3.20).

In addition to the loss due to sudden deceleration of the blading exit flow, another loss was included to account for the mixing of the volute flow and the "injected" blading exit flow. This loss is scaled on the difference in dynamic pressure between the blading exit flow and the volute flow at the blading exit radius R_{B2} , with a coefficient K_{exp} to allow adjustment, and is given by:

$$\Delta P_{t_{mix}} = 0.5 \rho K_{exp} (C_{B2}^2 - C_b^2) \quad (3.21)$$

The computer model was reworked so that given a guess at a blading sector's flow rate, the volute sector's exit static pressure was calculated. New guesses at the blading sector's flow rate were made until the pressure value calculated using the compatibility equations matched that determined using equation (3.13). A match indicated that the blading flow could enter the volute compatibly. Volute flow velocities at the blading exit are calculated using equation (3.16). It should be noted that this, and any other compatibility models discussed in this thesis were used only to select a blading sector flow rate. All calculations of actual simulated flow pressures and velocities were performed using the equations discussed in sections 3.1 to 3.3.

3.4.2 Angular Momentum Conserving Model

The third and final model applies the conservation of angular momentum within the volute sector control volume "abcdefgha" (see Figure 3.4). Two variations of this model were examined: conservation of momentum in the blading exit flow streamtube, and conservation in the entire volute sector control volume about point 'a'. The first variation seemed easy to implement initially, but it was later determined that the transport of momentum in and out of the blading exit flow's streamtube was very difficult to model. The second variation is similar to equation (3.13), except for the fact that moments are calculated about point 'a' (see Figure 3.4) instead of the rotor's centre of rotation. The conservation of angular momentum equation is given as:

$$P_d A_d |da|_{\perp}^w + P_r (A_h - A_d) |fa|_{\perp} + P_b A_b |ba|_{\perp}^r - P_r A_r |fa|_{\perp} - P_h A_h |ha| - \text{friction} = \quad (3.22)$$

$$Q(-Q_d Cw_d |da|_{\perp}^w - Q_d Cr_d |da|_{\perp}^r - Q_b Cw_b |ba|_{\perp}^w - Q_b Cr_b |ba|_{\perp}^r + Q_h Cw_h |ha|)$$

The $\perp$ symbol represents the least perpendicular distance between the two points, see Figure 3.4 for clarification. The superscript 'r' and 'w' are meant to distinguish between the perpendicular distance from a radial velocity vector and a whirl velocity vector respectively. Thus, $|da|_{\perp}^w$ would represent the perpendicular distance from a vector in the whirl direction that passes through point 'd', to the centre of rotation 'a'.

This model was thought to have potential for development since, unlike the sudden

enlargement model, it judges the ability of the blading flow to enter the volute directly on its static pressure and radial velocity, two parameters which one would expect compatibility to be influenced by. This model was conceived late in the development of this computer simulation, and so only a preliminary investigation into its performance was completed. The curvature of the control volume encouraged some simplifying assumptions to be made in this preliminary study: first, that the distance of Cw_b from the axis 'a' was small enough to be neglected, and second, for perpendicular distances that were difficult to calculate, estimates could be used. An example of one such estimate is the substitution of the distance $|dc|$ for the perpendicular distance $|da|_{\perp}$. These two assumptions were thought to provide a rough estimate of the model's behaviour, as long as the sector size remained small. To further simplify equation (3.22), the volute wall friction was for the time being neglected. Testing of this much simplified version of equation (3.22) gave useless results for the most part. The simplifying assumptions made above proved to be very poor, resulting in non-sense predictions, even when 100 sectors were used. Work is required to determine the merit of this model.

4.0 THE COMPUTER PROGRAM - FANSIM

In this chapter, the computer algorithms developed to implement the fan flow models presented in Chapter 3.0 are discussed.

4.1 OVERVIEW

The simulation algorithm was broken up into separate blocks: data input, geometry initialisation, blading performance calculations, main solution loop, and solution output. The flow chart in Figure 4.1 shows the breakdown of the algorithm, as well as the flow of information around the program. To function, the program required information from three external text files, divided by the type of data they store: fan geometry data in INGEO.DAT, simulation running information in INRUN.DAT, and fan aerodynamic model loss coefficients in INLOSS.DAT. A description of the layout and contents of each of these three data files can be found in Appendix A.

Throughout the program, six primary variable arrays were used to keep track of the flow parameters for each element of the fan: inlet region entry, inlet region exit, blading region entry, blading region exit, volute region entry, and volute region exit. These six arrays were dimensioned so that each fan element has a storage location for its flow parameters. Currently, the program is set up to handle a maximum of 25 slices in

the axial direction, and 100 sectors around the fan circumference. Depending on the region, different flow parameters were tracked for each fan element. The static pressure and volume flow rate were found for all six regions. All other flow parameters, such as flow angles, that were tracked are shown in Figure 4.2.

In addition to the primary flow variables, two other arrays of data were carried throughout the program: the fan discretization geometry (FANGEO), and the fan element entry/exit areas (AREAS). FANGEO contains the start and end angle of each sector in the fan, as well as the casing wall radius at each of these angular stations. The cross-sectional area of the entry and exit surface for every fan element was stored in AREAS. Details of the layout of the information stored in these arrays are also presented in Figure 4.2. Any variables used by more than one block of the simulation were declared as COMMON. The program operation information read in from the three data files is an example of such shared data.

The simulation code, written in FORTRAN 77, was designed from conception to be modular. By placing certain blocks of programming, such as blading loss models, into separate subroutines, the result was a flexible program structure that was easy to modify. The purpose for this flexibility was to easily permit the modification of this simulation to model reverse flow in the future. A chart showing the layout of the actual code can be found in Figure 4.3; note the block labels' correspondence to subroutines in the actual code (see Appendix B). The sections to follow present each algorithm block in detail,

including the particular subroutines used.

4.2 DATA INPUT

The data required by the program was divided into the three text files mentioned in the preceding section: INGEO.DAT, INRUN.DAT, and INLOSS.DAT. INGEO.DAT contained all the fan geometry information, such as rotor inner and outer radii, and blading metal angles. All data regarding the number of sectors and slices, the number of data points to find, and other simulation related information, was stored in INRUN.DAT. Finally, all of the loss coefficients and arbitrary model "fine-tuning" coefficients used by the models discussed in Chapter 3.0, were stored in INLOSS.DAT.

While the three data files were read in by the subroutine GETDAT, the program offered an opportunity for the user to double check, and change if desired, the value of all the input information. Also, checks were made to ensure that certain input data such as the number of sectors or slices, or the values of the program option switches, do not exceed the accepted limits prescribed by the program. After the input data was confirmed by the user, the program asked the user to enter an eight character output file name. Once the name for the output file was entered, a summary of the checked input data was written to the top of the output file.

4.3 GEOMETRY INITIALIZATION

Once the simulation data had been read in, the program divided the fan geometry up into the number and distribution of elements specified. Options for element distribution included: more slices at front of fan, more slices at back of fan, equal slice distribution, more sectors at start, more sectors at end, and equal sector distribution. The actual division of the fan into elements was done by dividing the rotor width B_B into slice widths, and the 360° of fan into the start/end angles of each sector. In the case of more slices at one end, a simple sine distribution was used:

$$B_B(i) = B_B \left(1 - \cos\left(\frac{i\pi}{2num_i}\right)\right) \quad \text{more slices at front} \quad (4.1)$$

$$B_B(i) = \frac{B_B}{num_i} \quad \text{uniform slices} \quad (4.2)$$

$$B_B(i) = B_B \sin\left(\frac{i\pi}{2num_i}\right) \quad \text{more slices at back} \quad (4.3)$$

where $B_B(i)$ is the width of the i^{th} slice, with slice one at the front of the fan (near inlet), and slice "numi" at the back of the fan. A similar method was used to distribute sectors around the fan circumference. In this case, the angular location of each sector's start $\theta_{\text{start}}(j)$ and end $\theta_{\text{end}}(j)$ plane was determined:

$$\theta_{start}(j) = 2\pi(1 - \cos(\frac{(j-1)\pi}{2numj})) \quad \text{more sectors at start} \quad (4.4a)$$

$$\theta_{end}(j) = 2\pi(1 - \cos(\frac{j\pi}{2numj})) \quad " \quad (4.4b)$$

$$\theta_{start}(j) = \frac{2\pi(j-1)}{numj} \quad \text{uniform sectors} \quad (4.5a)$$

$$\theta_{end}(j) = \frac{2\pi j}{numj} \quad " \quad (4.5b)$$

$$\theta_{start}(j) = 2\pi \sin(\frac{(j-1)\pi}{2numj}) \quad \text{more sectors at end} \quad (4.6a)$$

$$\theta_{end}(j) = 2\pi \sin(\frac{j\pi}{2numj}) \quad " \quad (4.6b)$$

The start/end angles were then used to find the volute wall radius at the entry and exit of each volute sector. These four pieces of data were stored in FANGE0 for each sector. This information was then combined with the slice widths to find entry and exit areas for each fan element in all three fan regions, these being stored in AREAS.

4.4 BLADING PERFORMANCE CALCULATIONS

Since the pressures and whirl velocities exiting each blading element would be

needed repeatedly during the main solution loop, over a range of flow rates, it was decided that only one calculation of blading performance over the entire flow rate range would be done, and a quadratic equation fitted to the results. This would allow the pressure and whirl velocity for any desired blading exit radial velocity to be found by making a simple mathematical function call. In this thesis, only forward blading flow was simulated, and the absolute blading inlet flow was assumed to be completely radial. However, since this computer program was intended to be modified in the future to model reverse flow in parts of the rotor, performance curves for absolute inlet flow angles from 0° to 360° were generated. This will give future developers a head start on adapting the code to simulate operating conditions with reverse flow.

The actual curve generation process started by calculating a few basic geometric and velocity triangle parameters such as pitch to chord ratio, and incidence angle. These parameters were found in the subroutine BLDVAL for forward flows, and REVBLD for reverse flows. Then, the subroutine SLIPFA (REVSLP for reverse flow) calculated the blading exit flow angle β_2 using the deviation angle correlation as discussed in section 2.3 and 3.2. Once the exit flow angle was known, the blading exit whirl velocity was calculated from trigonometry. The Euler total pressure rise was calculated using the already determined flow angles. The pressure losses in the blading as discussed in section 3.2 were calculated, and subtracted from the Euler total pressure rise to determine the net total pressure rise across the blading. Once the exit whirl velocity, Euler total pressure rise, and net total pressure rise had been found for twenty different rotor radial flow

velocities across the flow rate range of the rotor ($\Phi=0.0$ to 2.0), the three arrays were sent to the subroutine FITCRV to be converted into a 3×3 matrix of coefficients for each of the three quadratic curve fits. The entries of these matrices were determined using an algorithm for finding a j^{th} order "least squared error" polynomial through 'n' data points as presented in Pearson (1986, pg.95):

$$\begin{aligned} & \sum [y_i - (a_0 + a_1 x_i + a_2 x_i^2 \dots + a_j x_i^j)] \quad , i = 1, n \\ & \sum x_i [y_i - (a_0 + a_1 x_i + a_2 x_i^2 \dots + a_j x_i^j)] \\ & \vdots \\ & \sum x_i^j [y_i - (a_0 + a_1 x_i + a_2 x_i^2 \dots + a_j x_i^j)] \end{aligned} \quad (4.7)$$

These matrices were solved for the three coefficients of the desired "least square error" quadratic curve fit:

$$y = a_2 x^2 + a_1 x + a_0 \quad (4.8)$$

The matrices were solved in the subroutine LINEAR, a Gaussian elimination matrix solver proposed by Pearson (1986, pg.30). Three separate families of quadratic equations for: exit whirl velocity, Euler total pressure rise, and net total pressure rise, as a function of the blading exit flow radial velocity, were generated. Within each of the three families, there was one curve corresponding to each integer degree of possible blading inlet flow angle, 179 curves for all the possible forward flows ($\alpha_1 = -89^\circ$ to 89°), and

another 179 curves for all the reverse flows ($\alpha_1 = 91^\circ$ to 269°). Flow directions -90° and 90° relative to the radial direction were omitted since the term $\tan(\alpha_1)$ used in the blading calculations becomes infinite. In this thesis, since the absolute inlet flow angle was assumed to be 0° , only curve number 90 from each of the three families (exit whirl velocity, Euler total pressure rise, net total pressure rise) was actually used by the main solution loop.

4.5 MAIN SOLUTION LOOP

The main solution portion of the program consisted of a series of nested iteration loops, as shown in Figure 4.4. To initiate the solution, the gap flow rate and first blading sector flow rate had to be selected. The gap flow rate was selected by dividing the possible range of gap flow rates, assumed to be from $\Phi=0.0$ to 0.5 , into ten divisions. The value at the first division was taken as the gap flow rate. Later in the solution loop, this selection was checked to see if it was compatible with the rest of the flow in the fan. If not, then the gap flow rate was set to the value of the next division, until a compatible one was found. This iteration to find the gap flow rate is discussed in detail later.

The first blading sector flow rate was selected in a similar fashion to that of the gap flow rate. The possible range of fan flow rates ($\Phi=0.0$ to 1.0), was divided up evenly by the number of performance curve data points requested by the user. This resulted in an estimate for the overall fan flow rate expected at each data point. When the first

performance curve data point was to be found, the estimated overall fan flow rate for that point was divided amongst all the blading sectors according to how much of the rotor's exit area they took up. The flow rate assigned to the first blading sector was the flow rate selected in order to get the rest of the fan solution started. This process occurred for each of the performance curve data points.

The selected first blading sector flow rate was then distributed axially amongst the slices of the blading sector by the subroutine DIVFLO, and the flow rate weighted average exit static pressure and velocity were calculated. The static pressure at the gap was selected by setting it equal to the static pressure of the selected flow leaving the first blading sector. With the inlet flow conditions into the first volute sector selected, the exit flow could be determined by the subroutine CALCP2.

From this point on, the exit flow for each remaining volute sector (from $j=2$ to $j=numj$) was calculated in succession. This was done by guessing a blading sector flow rate, and calculating the volute sector exit static pressure using one of the compatibility models in the subroutine COMPAT. The blading sector flow rate guess was actually a marching process, where flow rates from Φ equal to 1.0 to 0.0 were tried one at a time. The returned value of P_{v2} was compared to that found by CALCP2. Blading sector flow rate guesses were stepped through until the two pressures crossed each other in value. If they crossed, the size of the flow rate step used was divided by ten, and the process started again. The end result of this step-size refinement was that at some point, the two

calculated pressures differed by less than the user-defined tolerance TOLER. If the difference in the two calculated P_{v2} values did not differ by less than the tolerance for any of the blading sector flow rates tried, the flow rate resulting in the closest match was chosen. This process repeated for the remaining fan sectors.

After the compatible blading flow rate for each sector had been found, the selected gap flow rate was checked to see if it was also compatible. Some of the flow exiting the last fan sector passes through the gap into the first sector. This flow was assumed to have the same total pressure as the last volute sector's exit flow, less the loss scaled on the change in dynamic pressure across the gap:

$$P_{t_{gap}} = P_{t_{v2,numj}} - 0.5\rho K_{gap}(C_{v2,numj}^2 - C_{v1,1}^2) \quad (4.9)$$

Note that the two flow velocities upstream and downstream off the cut-off ($C_{v2,numj}$ and $C_{v1,1}$ respectively), when used in equation (4.9), were evaluated at the gap mean radius using equation (3.16). The value of K_{gap} was assumed to be 0.0, since accelerations/decelerations across the gap should be small, and gradual, resulting in little loss. The acceleration/deceleration of the gap flow is dependent on the pressure difference across the gap. The operating nature of these fans does not allow large pressure differences across the gap to accumulate, since this would affect the flow through the blading as well as the gap (eg. reverse flow).

With the gap total pressure calculated, Bernoulli's relationship could be used to find a value of static pressure in the gap. This value was compared to the value selected at the start of the solution process. If the two pressures differed by less than the user-defined tolerance TOLER, the gap flow was considered compatible, and the overall fan performance parameters were calculated. If the two pressures did not match, a new gap flow rate selection was made, and the entire solution process repeated itself.

The above process was repeated for all the requested number of fan performance curve data points NUMM. The simulation was designed to generate NUMM data points on the fan's performance curve, for the entire operating flow rate range of Φ equal to 0.0 to 1.0.

4.6 SOLUTION OUTPUT

The checked input data, and the fan overall performance data for each point was written to the user-entered file name, with the suffix .OUT added. Two optional output files could also be written: detailed blading element flow data, and detailed volute sector flow data. These two files used the entered file name with the suffixes *.BLD and *.VOL respectively, and contained the flow pressure and velocity distributions at all of the blading and volute element boundaries. The results were written to the prospective output files by the subroutine OUTDAT.

5.0 COMPUTER PROGRAM PERFORMANCE

To determine the merit of the computer simulation developed in this thesis, its ability to accurately predict the flow conditions inside squirrel-cage fans must be determined. In this chapter, the predictions of the blading performance calculations, and the main solution loop, are compared to the available experimental data.

5.1 BLADING ROUTINE

In order to verify the validity of the blading performance calculations, pressure and velocity measurements right at the blading entry and exit are needed for a range of radial velocities (ie. rotor through flow rate). The effects of the fan casing, and the many secondary flows inside the fan, tend to complicate the gathering of rotor performance data over the desired flow rate range. At present, useable experimental data for the amount of total pressure rise developed by the rotor is not available. The accuracy of the blading model by itself can not be determined until this information is available.

The only aspect of the blading performance calculations that can be determined is the validity of using a quadratic curve fit to replace the blading calculations. Figure 5.1 shows a plot of the Euler (no losses) and net total pressure rise across the blading, as determined using the calculations discussed in section 3.2, and the corresponding least-

squared error quadratic equation that was fit to the data. The curve fit matches the calculated point very well, and so it appears that a quadratic equation will be a more than adequate substitution. If, in the future, blading models resulting in more complicated performance curve shapes are used, the program has been written to permit up to a fifth order least squared error polynomial to be used.

5.2 MAIN SOLUTION LOOP

The most important requirement of the program is that it must "simulate" the performance of squirrel-cage fans. The calculated results have been compared to the experimental fan performance data measured by Kind and Tobin (1990) shown in Figure 2.8. For each of the three fan geometries tested by Kind and Tobin, the simulation results for the two compatibility models discussed in section 3.4 (ie. uniform flow and sudden enlargement) are compared with the measured performance (Figures 5.2 to 5.4). In each case, the simulation computer program parameters giving the best match to the experimental data were used, these values being listed in Figure 5.5.

It appears that the two compatibility models produced rather mixed results, matching the experimental data well for design point and higher flow rates only for the Rotor1 Casing2 geometry. From Figure 5.2, one can see that the computer simulation predicts a peak total pressure rise of 1.1 using both compatibility models, 5% lower than the experimental peak. The peak is also predicted as being at a 28% and 45% higher

flow coefficient using the uniform flow and sudden enlargement models respectively. The simulation's prediction of fan efficiency is worse, underpredicting the peak efficiency by 7% at roughly half the flow coefficient. Also, the predicted efficiency peak is at a flow coefficient where reverse flow (which is not modelled) would be expected to exist, making the result completely inaccurate.

The simulation results for Rotor1 Casing2 (Figure 5.3) are significantly better than for the first test geometry. The uniform flow model overpredicted the peak total pressure rise by 4%, but at the correct flow coefficient. The sudden enlargement model is very close to predicting the peak total pressure rise and the flow coefficient at which it occurs exactly. The predicted peak efficiency of both compatibility models is incorrect, but the predicted efficiency curves are very close to the experimental data at flow coefficients above the design point (ie. best efficiency point). This is acceptable considering that the existence of reverse flow at below design point flow rates is not modelled by the simulation, making results incorrect here anyway.

The fact that the accuracy of the predicted results vary so much between the first two fan test geometries, implies a definite weakness in the simulation model. In both Figure 5.2 and 5.3, the two compatibility models give very similar results, predicting the value and location of total pressure rise and efficiency maximums in essentially the same locations. This suggests that the compatibility models are not at fault, but the blading performance calculations and/or the model of the volute region may be. Physically, the

two test fans differ only in one respect, the volute wall radius at the fan exit is 83mm larger in Casing2 than in Casing1. This results in a larger casing spiral angle, and a larger volute cross-sectional area at corresponding volute sector faces. Since the rotor does not change between the two test cases, it is unlikely that the error is in the blading performance calculations. The value of casing spiral angle affects the relationship between whirl and radial velocity components in the volute flow. Perhaps the assumption that radial velocity components are a linear function of whirl velocity made by (eqn. 3.14), needs to be re-examined to ensure it is valid. Also, the existence of the spiralling flow in the volute complicates the relationship between whirl and radial velocity. It may not be acceptable to ignore the effect of the spiralling flow on velocity distributions in the volute if good computer predictions are expected for a range of fan geometries.

During the development stage of the program, an angular momentum conservation model was tried in which the flow exiting the blading was assumed to decelerate inside a gradually enlarging streamtube. The major problem with this model was that no feasible method of accounting for the momentum transferred in and out the streamtube by the volute's spiralling flow could be devised, short of modelling and tracking it throughout the fan. It was undesirable for this additional complexity to be added to the simulation, but it seems that it may in the future be necessary. The fraction of the volute's total pressure accounted for by this spiralling flow may be significant, affecting not only the distribution of static pressure in the volute, but the distribution of blading flow around the fan as well.

Another possible source for the error in the volute model, is the treatment of flow in the region of the casing cut-off. Currently, the static pressure just downstream of the cut-off is set to equal the static pressure exiting the first blading sector. This was necessary in order to initiate the solution process. The final, compatible gap flow rate chosen by the simulation is found assuming that the total pressure on either side of the cut-off is equal (minus a small loss, see eqn. 4.2). This current model of the flow in the area of the gap therefore has two definite failings: the gap flow rate is dependent on the first blading sector flow rate, and the effects of the cut-off do not transmit upstream, only downstream. Another problem with this handling of the gap flow, is that an accurate gap flow loss model does not yet exist.

It should be noted that the sudden enlargement model did predict the presence of reverse flow at the lower fan flow rates. When the blading exit flow static pressure after the sudden enlargement is found to be lower than the static pressure in the volute for every possible forward blade sector flow rate, the conclusion is made that no compatible forward flow rate exists. Currently, when this occurs in the simulation, the blade sector flow rate closest to being compatible is used. The effect of approximating the blading sector flow rate with the best match can be seen by the waviness of the fan performance curves at flow coefficients below design point. When this simulation is adapted to model reverse flow, the program behaviour mentioned above could be used to tell the simulation to start looking for compatible reverse blading sector flow rates instead.

Returning to the plotted results, in particular Figure 5.4, the computer model appears to match the experimental data for the rotor with radial exit blades much worse than for the two forward facing blade cases. Total pressure rise is underpredicted by both compatibility models by 20% over the entire flow coefficient range. Efficiencies at above design point flow coefficients are underpredicted by 25%. The only aspect of the fan performance curves that was predicted, was the shape of the two curves. The fact that radial exit bladed rotors generate much of their total pressure in the form of static pressure rise inside the blade-to-blade passages, may amplify the already existing inaccuracies in the volute model discussed above. The modelling of the interaction between volute static pressure and blading exit static pressure very likely is also affected by this characteristic of radial exit blades.

The simulation results are consistent with Kind's (1996), as expected, since his models were developed in conjunction with the work in this thesis. Differences lie primarily in the sudden enlargement compatibility model. In Kind's model, he uses the whirl velocity components of the blading exit flow and the volute flow at the blading exit radius to calculate the static pressure rise (equation 3.19). As presented in section 3.3, the overall velocity is used in this thesis, thus taking into account both radial and whirl velocity components. In addition to the difference in compatibility models, Kind does not use a quadratic curve fit for the rotor performance data. He instead chose to do individual calculations each time the whirl velocity or total pressure exiting a blading sector was needed. Beyond these two differences, all assumptions made regarding the

flow in each of the fan regions are the same as those made by Kind. The similarity of Kind's results with those in this thesis suggest that his simulation may have similar sources of inaccuracy.

The final aspect of fan performance that can be examined, is the prediction of the static pressure distribution around the circumference of the volute. In Figure 5.6, the predicted average volute static pressure is plotted versus the static pressure measurements made by Raj and Swim (1981). Pressure distributions are shown for the fan tested by Raj and Swim at its design point flow rate ($\Phi = 0.4$) and at 150% the design point flow rate ($\Phi = 0.6$). Both compatibility models predict the measured static pressure distribution very poorly. The general trends of the pressure distribution seem to be predicted properly, that is: a rise in static pressure for low flow rates, and a drop in static pressure for high flow rates. The problem with the predicted results appears to be related to assumptions around the casing cut-off, angular location 0° and 360° . First, the static pressure chosen at the start of the volute by the simulation is approximately 0.7 non-dimensional pressure units below the measured value. Currently, in simulation, the static pressure at the start of the volute is set equal to the static pressure exiting the first blade sector. Since the flow rate, and thus the static pressure exiting this blade sector, is chosen by the program according to which point in the performance curve is being found, the resulting static pressure at the start of the volute is based on non-flow related parameters. Judging by the results in Figure 5.6, this is a poor model. In fact, if the predicted pressure distributions were to start at the proper static pressures, as indicated by the

measured static pressure values, the simulation would have performed quite well.

The final problem with static pressure distribution predictions, is the absence of cut-off effects upstream (ie. angular locations just below 360°). Physically, the static pressure around the fan circumference can not be discontinuous. This results in an increase in static pressure (for design point and higher flow rates) right before the cut-off, as the flow from the later part of the volute passes towards, and through the gap. The measured performance data shown in Figure 5.6 confirms this, as well as the work of many other researchers. The current gap model does not account for this continuity in the flow, instead choosing to apply a sudden pressure change across the gap. This model is not only discontinuous, but fails to transmit the effects of the cut-off's presence to the volute flow upstream of the cut-off. These problems with the gap flow model appear to have affected not only the prediction of volute pressure distribution, but the overall fan performance predictions as well, as was already discussed above. Future work on this computer program must address this very major failing of the simulation.

6.0 COMPUTER PROGRAM BEHAVIOUR

It was illustrated in Chapter 5, that the simulation does not predict fan flow conditions very well. It is possible however, that the simulation is capable of modelling the variation of squirrel-cage fan performance with changes in a range of geometric and flow related parameters. Also, parameters relating to the computer algorithms used, such as the number of sectors, may effect the simulation's predictions. The sensitivity of this computer program to changes in simulation parameters is presented in this chapter.

6.1 BLADING MODELS

The models used in this thesis to simulate the flow passing through the rotor are presented in section 3.2. Figure 6.1 compares the blading exit flow deviation correlation with the no-deviation case. The predicted relationship between radial and whirl velocity is consistent with what one would expect from the velocity triangles for forward swept (rotor 1) and radial (rotor 2) blades. As the blading exit radial velocity increases, the blading's ability to turn the flow is diminished, increasing the flow deviation angle δ , and reducing the exit whirl velocity below the ideal. Beyond the correlation for deviation developed by Kind (1996), there were no other relationships for flow deviation examined. Perhaps in the future, some of the empirical methods developed for axial flow machines could be studied.

The blading loss correlation is compared with the ideal total pressure rise expected across the rotor in Figure 6.2. The effect of blading losses on the entire fan performance using the uniform flow distribution model, is presented in Figure 6.3 and 6.4 for rotor 1 and rotor 2 respectively. By comparing the no-loss and the losses-included performance curves, it appears that blading losses have a significant impact on the peak performance of the fan. The maximum total pressure rise and efficiency seems to be strongly linked to the blading performance, but the overall shape of the fan performance curves appears to depend on the flow in the volute. This is especially evident in Figure 6.4, where the no-loss and the losses-included curves are almost identical in shape.

The blading loss correlation matches relatively well for the forward swept blading of rotor 1, as was already discussed in section 5.2. It appears that the predicted losses may be a little high for rotor 2, the experimental data lying approximately one third of the way between the losses-included and the no-loss curve. Perhaps the validity of the correlation in predicting losses across typical radial and backward swept blading should be confirmed, possibly by testing it on other rotor/casing geometries.

6.2 FLOW MODEL SENSITIVITY

A sensitivity study was performed, in order to determine the behaviour of the simulation, and the effect of the many adjustable parameters involved in the program on predicted fan performance. There are three major types of parameters that can be

adjusted by the simulation's operator: fan flow model loss parameters, simulation running parameters, and fan geometry parameters. In each of the sensitivity studies discussed below, the input data for Rotor 1 Casing 1 was used with the forced uniform flow compatibility model, unless stated otherwise. The uniform flow compatibility model was used in order to eliminate any dependence of the results on the distribution of flow around the fan's circumference. All simulation data was the same as was used for the simulation performance assessment in Chapter 5.0 (see Figure 2.1 and 5.5), except for the particular parameter being studied. Below follows the results of each of the sensitivity studies.

6.2.1 Flow Loss Parameter Sensitivity

Within the simulation, there are seven coefficients used to adjust the various loss and compatibility models. This allows the simulation results to be fine tuned to match experimental data as well as possible. The seven coefficients are: rotor leakage fraction K_{leak} , sudden enlargement compatibility model adjustment K_{exp} , volute wall friction coefficient K_{fmc} , gap dynamic pressure loss coefficient K_{gap} , fan inlet dynamic pressure loss coefficient K_{inlt} , and a non-flow related efficiency K_{ext} . The non-flow related efficiency is meant to account for rotor windage and mechanical friction losses. This efficiency is normally quite high, around 98%, and has little effect on the behaviour of squirrel-cage fans. It was therefore left out of the loss parameter sensitivity analysis. The remaining six parameters were varied over the range of values expected in squirrel-cage

fans to see how they affect the simulated fan performance. The way in which these coefficients are implemented in the simulation, has already been discussed in Chapters 3.0 and 4.0.

The first parameter, the rotor leakage fraction K_{leak} , is plotted in Figure 6.5 for values between 0% and 20%. There is roughly a 30% drop in peak total pressure and a 50% drop in peak efficiency with only 20% rotor leakage. The loss is expected since potentially useful work is done on fluid that never contributes to the volute flow, and the overall fan exit flow. Leakage percentages of 20% would likely not be encountered in commercial squirrel-cage fans. They are normally constructed with a small rotor shroud-to-casing gap, relative to the rotor's width. This would tend to reduce the amount of leakage considerably. No experimental data exists regarding what the actual percentage leakage is, or how it behaves across the fan's operating flow rate range. Judging by the severity of the performance drop due to increases in leakage, concern should be given to leakage during fan design.

In Figure 6.6, the effect of the compatibility modifier K_{exp} on the results generated using the sudden enlargement compatibility model are shown. Two extremes in the model's behaviour are illustrated in the figure: performance curve data points found only for very high fan flow rates when K_{exp} is small, and incompatible blading sector flow rates found for low fan flow rates when K_{exp} is large. The first extreme appears in the partial curve for K_{exp} equal to zero. Since the program is allowed to iterate to the first

compatible fan flow rate it can find, there is no mechanism forcing the resulting performance data points to be entirely within the normal flow rate operating range of a squirrel-cage fan (ie. flow coefficients between zero and one). As the compatibility modifier is increased from zero, the iterated solution shifts so that it lies entirely in the operating flow rate range of the fan.

As the calculated curve shifts to the left, no solution is found for progressively more data points at low flow rates. This is visible on Figure 6.6 through the irregular shape of the curves at the extreme left side of the plot. This behaviour was expected, since the model is not capable of handling the reverse flow present at lower flow rates. The inability of the model to find a solution can be used to predict the presence of reverse flow in the fan, triggering a second set of models which are formulated to specifically handle reverse flow. This idea of first predicting and then calculating reverse flow in the fan is further discussed in Chapter 7.0. Another factor that supports the use of this compatibility model to predict the existence of reverse flow is that the location and value of the maximum total pressure rise and maximum efficiency do not vary dramatically with the value of K_{comp} .

The volute wall friction coefficient K_{fmc} has a very noticeable effect on the performance of the fan (see Figure 6.7). As the friction increases, the total pressure rise and efficiency drop for all flow rates, as would be expected. The losses are more significant at higher flow rates since the friction losses scale on the dynamic pressure in

the volute. The effect of friction on the overall performance of the fan appears to be significant in comparison to other factors, such as rotor performance. This may be due to the very high dynamic pressures encountered in squirrel-cage fans, especially those with forward swept rotor blading.

The effect of the gap dynamic pressure loss coefficient K_{gap} on the simulation results can be seen in Figure 6.8. It is quite obvious that something very strange and probably non-physical is happening in the model when increasing values of K_{gap} are used. The loss term in the simulation is scaled on the difference in dynamic pressure between the gap flows just upstream and downstream of the cut-off. The larger values of K_{gap} used in the sensitivity analysis are probably unrealistic, resulting in the unusual behaviour. It was assumed in section 4.5 that the value of K_{gap} is zero, so this behaviour of the simulation is currently not a pressing concern. After the gap flow has been studied in more detail however, this loss term should be re-examined. This current behaviour casts definite doubt on the use of the current gap loss model. The irregular behaviour is very likely compounded by the problems in the way the gap flow is handled altogether, as was previously discussed in section 5.2.

The fifth parameter examined was the fan inlet dynamic pressure loss coefficient K_{inlet} . The effect of a range of values for K_{inlet} is plotted in Figure 6.9. Even without the modelling of reverse flow, the inlet loss has little effect on the simulated fan performance at low flow rates where the dynamic pressure in the fan inlet region is low. As the flow

rate, and thus dynamic pressure increases, so does the inlet loss. This behaviour is consistent with what would be expected in the actual fan. Inlet losses are one of the main design criteria currently addressed by squirrel-cage fan designers (Eck, 1973; Gessner, 1967). Many design strategies have been developed to address losses in this region.

6.2.2 Simulation Running Parameter Sensitivity

There are several parameters that can be varied in the simulation program that do not relate to the geometry of the fan, or the flow models used. These parameters include: the number of sectors, the number of slices, the distribution of sectors and slices, and the axial distribution of flow in the fan inlet region. The sensitivity of the forced uniform flow and the sudden enlargement compatibility models to the number of fan sectors is shown in Figure 6.10. The forced uniform flow model is essentially insensitive to the number of sectors, and the sudden enlargement model is only very slightly sensitive, as one would desire in any computer simulation. When a similar comparison was done for different numbers of slices, it was found that both models were almost completely insensitive. This can be explained by the fact that the average of the flow leaving all the slices is used in the volute and compatibility models.

This fact also explains why using different distributions of slices (ie. more near front or back of rotor) had almost no effect on the solution as well. The distribution of sectors around the fan's circumference does however have a slight effect on the simulation

results. Three different distributions are available in the program: cosine, putting more sectors at the start of the fan (at cut-off), uniform, and sine that makes more sectors at the end of the fan. It was observed that the simulation predicts slightly higher total pressure rises and efficiencies when more sectors are used near the end of the fan. This behaviour was found using both the forced uniform flow, and the sudden enlargement compatibility models. At this point, it is still unclear why this occurs. The magnitude of the increase in performance was small, therefore the source of this behaviour was not a pressing concern.

The final model parameter that was examined was the axial flow distribution in the inlet region. The five different distributions used are illustrated in Figure 3.2, and the effect of using them with the forced uniform flow model is presented in Figure 6.11. The figure indicates a very definite trend in the simulation results. As more of the flow is distributed near the back of the rotor, the maximum total pressure rise and efficiency increase, but the performance decreases at high flow rates. This behaviour may be due to the fact that when the fan's flow is gathered all at the back of the rotor, the radial velocity and thus the exit whirl velocity from the blading is significantly higher here. This means that the total pressure rise across the impeller will be higher for distributions such as parabolic or equal area than for a uniform axial distribution. Also, even though the flow passes through a larger change in total pressure, the losses in the inlet region are the same as if the axial distribution of flow was uniform, since the axial distribution of flow does not affect the dynamic pressure in the fan inlet. Finally, since the blading exit

flow has to quickly enlarge to fill the volute, more rapidly than if the flow was uniformly distributed between the blading slices, the losses in the flow should be rather high. Therefore, at higher flow rates when the sudden enlargement loss term (equation 3.21) is large, the predicted performance is lower.

6.2.3 Fan Geometry Sensitivity

The final set of parameters that can be varied in the program is the geometry defining the fan being simulated. To study the effect of fan geometry on the simulated performance of the fan, five non-dimensional geometry based parameters are studied. The effects of the geometric parameters on predicted fan performance should compare well with that observed in experiment, at least this is if the simulation developed in this thesis is a good model of actual squirrel-cage fan behaviour. The five parameters studied were:

$$\text{blade exit angle} = \beta_2' \quad (6.1a)$$

$$\text{gap size ratio} = (R_{V1} - R_{B2})/R_{B2} \quad (6.1b)$$

$$\text{fan exit/fan inlet area ratio} = [(R_{V2} - R_{V1})B_V]/(\pi R_{B1}^2) \quad (6.1c)$$

$$\text{blading inner/outer radius ratio} = R_{B1}/R_{B2} \quad (6.1d)$$

$$\text{rotor exit/fan inlet area ratio} = 2B_B/R_{B1} \quad (6.1e)$$

In each of the last four parameters, the geometric value adjusted was: the radius of the cut-off R_{V1} , the radius of the casing at the fan exit R_{V2} , the inner blading radius R_{B1} , and

the rotor width B_b respectively. All other geometry values were set to the same as for the Rotor 1 Casing 1 test fan (see Figure 2.1) with the exception of the casing width when parameter (6.1e) was studied. In this case, the non-dimensional width of the casing was kept at 0.3 units larger than the rotor width, the same as the three fan test geometries used in Chapter 5.0. In all of the simulation runs discussed below, the forced uniform flow compatibility model was used, as well as the equal-area axial flow distribution (see Figure 5.5).

The first parameter, the blade exit angle β_2' , was varied from 45° to -75° (see Figure 3.3), and the overall fan performance calculated (Figure 6.12). The behaviour of the simulation seems to match that suggested by other researchers (Figure 2.7), except of course for the effects of reverse flow at low flow rates. The peak efficiencies actually attainable by radial and backward facing blading were not predicted of course, since the optimum casing for such a fan configuration was not used in the simulation.

When the gap size was varied, the program predicted a very dramatic change in fan performance, as shown in Figure 6.13. As the gap size was reduced, total pressure and efficiency rose significantly. Worster (1963) observed the same behaviour in his experimental analysis of centrifugal pump volutes. He attributed the rise in performance to the increase in the volute exit or "throat" area that accompanies the gap size decrease.

The only practical drawback of a smaller gap would be that the characteristic low

noise of these fans would be lost. The pressure fluctuations caused by the rotor blades moving closely past the cut-off would generate large amounts of noise and vibration. Therefore a trade-off between improved performance and silent operation must be made during the design of squirrel-cage fans.

In Figure 6.14, the effect of varying the fan exit area is presented. It appears that having a larger casing results in higher fan performance, especially at high flow rates. This behaviour, again, was identified by Worster (1963) in centrifugal pumps. The improvement in fan performance due to the volute's size can be explained by noting that all of the losses in the volute scale on the volute flow's dynamic pressure. A larger casing allows the volute flow to decelerate more, preserving the total pressure (minus any diffusion losses), but reducing the dynamic pressure and the related losses. Depending on the application of the fan, it may be beneficial to have more of the fan's total pressure in the form of static pressure rise. Of course, casing size is restricted by cost and the space available where the fan is to be mounted, as well as the preferred duct work size.

The fourth parameter examined was the ratio of rotor blading inner radius to outer radius, the effects of varying which are shown in Figure 6.15. The simulation program suggests that there is an optimum value of this ratio, for this fan geometry, somewhere between 0.8 and 0.9. This, by coincidence, is the typical value found in squirrel-cage fans, the two rotors used in this thesis for example have ratio values of 0.81 and 0.82. This behaviour is similar to that found by Eck (1973, pg.151), and may be explained by

noting that as the blade chord increases, so do frictional losses in the blade passages, decreasing the efficiency of the rotor. As the blade chord decreases, the frictional losses drop, but the ability of the blade to effectively guide the flow is reduced. At some point, the deviation of the exiting flow is large enough to compensate for any increases in total pressure rise across the rotor due to reduced friction. This explanation is supported by Figure 6.15. As the radius ratio is increased, the fan's performance at high flow rates, where friction losses are high, reduces dramatically. At a point, the fan's performance no longer increases at high flow rates, and the entire performance curve begins to drop. This shift in the performance curve is indicative of an increase in exit flow deviation.

The final parameter examined was the rotor exit area to fan inlet area ratio. Figure 6.16 shows that the fan's predicted performance increases as the area ratio is increased. The reason for this increase was unclear, since the change in rotor and casing width also changed the fan exit area. It was already shown that the fan exit area has an effect on the fan's performance. In fact, upon comparison of Figure 6.14 and 6.16, one can see that they are very similar. Therefore, the impact of the rotor exit area to fan inlet area ratio on fan performance is still uncertain, except that it defines what centrifugal fans fit into the category of squirrel-cage fans. The accepted value for the rotor exit area to fan inlet area ratio for squirrel-cage type centrifugal fans is 2.5 or larger, the fan geometries used in this thesis being around 2.7. A summary of the effect of the last four parameters discussed (6.1b -6.1e) is presented in Figure 6.17. Again, one can see that the casing exit area ratio and rotor exit to fan inlet area ratio curves are very similar.

7.0 FUTURE WORK

7.1 CURRENT SIMULATION

The performance of the computer simulation developed within this thesis has been discussed in the previous two chapters. Many improvements are required if satisfactory prediction of squirrel-cage fan performance is expected. Probably the largest improvement that can be made to the simulation is the inclusion of reverse flow modelling capabilities. This potential future modification is discussed in the next section. After reverse flow modelling, the next important improvement required is a better model of flow in the region of the casing cut-off. Comparison of the simulation's predictions of fan performance to experimental data revealed (see section 5.2) that the current handling of flow in the gap is very poor. Fan overall performance predictions (ie. total pressure rise and efficiency), when compared to experimental measurements, showed mixed results, hinting that the use of a discontinuous model across the gap does not work well. The performance predictions, along with the predictions of volute static pressure distributions, also suggest that the setting of the gap static pressure equal to the first blading sector exit static pressure, is not an accurate model. An alternative to these gap models must be found, possibly with the aid of existing gap flow work on other centrifugal machines (eg. Brownell and Flack, 1984) or new experimental data.

Another area of the simulation that should be improved is the compatibility model. Use of the sudden enlargement model seems to result in satisfactory predictions of flow rate and pressure distribution around the fan circumference. Also, the model predicts the existence of reverse flow through the correct parts of the fan rotor, at the correct fan flow rates. The problem with continuing to use this model occurs when the simulation is modified to account for reverse flow at below design point flow rates. When a blading sector is found to have reverse flow, the sudden enlargement model is no longer valid. Either a separate model will have to be developed for the sectors with reverse flow, or a new universal model should be found.

The conservation of angular momentum model proposed in section 3.4.2, has the potential to be a universal compatibility model, capable of selecting blade sector flow rates regardless of whether they are forward or backward flows. This model should be investigated further to determine if it is indeed the model of choice. Development of this model, or any other compatibility model, would be greatly helped by experimental measurements of how the blading exit flow interacts with the volute flow. Knowledge of how the blading exit flow mixes with the rest of the volute flow would be very beneficial. Making the necessary quantitative measurements within the fan is a technically daunting task. An alternative may be to set up a test rig that simulates the flow through only one sector of the fan. Flow pressures and velocities entering from the blading and the upstream volute sector could be controlled, and varied systematically.

In addition to the main solution loop, there are several areas of future work regarding the blading total pressure rise calculations. To begin, only Kind's (1996) loss and deviation correlations were used in this simulation. There are no other correlations specifically for blading of the type used in squirrel-cage fans, but well established correlations for axial compressor stages may be applicable. Sagi (1990) investigated many of these models, but failed to determine the merit in using any of them.

To better justify the use of Kind's (1996) correlations, more experimental data is required. Overall fan performance measurements are needed over a larger range of fan geometries. The lack of such data limited the accuracy assessment of the simulation to fan blades essentially identical to those used in Kind's cascade experiment. The accuracy of the correlation for different blade geometries must be determined. Another method of assessing the blading model's accuracy would be to make pressure and velocity measurements right at the blading entry and exit. The hot-wire anemometer measurements of Raj and Swim (1981), and the five-hole probe measurements of Kind and Tobin (1990), provide only some of the required data. The proximity of the measuring instrument to the spinning rotor presents many technical difficulties in such experiments.

Raj and Swim (1981) found that the size of the separated region at the front of the rotor varied with circumferential location, and fan flow rate. Currently, this fact is ignored, a simple axial flow distribution being applied to each inlet region element

regardless of its location, or the fan's flow rate. The implementation of a more accurate inlet flow model, one that accounted for the variations of the separated region's size, would be fairly simple if used in conjunction with reverse flow modelling. The extra overall fan flow rate iteration loop added by the modelling of reverse flow (see next section) would allow inlet sector axial flow distributions to be determined from the previous iteration's circumferential flow rate distribution. Development of this inlet flow model is dependent on the gathering of quantitative data regarding the variation of axial flow distribution with circumferential location and fan flow rate.

Another area that should be addressed is the affect of the spiralling volute flow on fan performance. The behaviour of this secondary flow should be examined, as well as its variation in intensity with location and fan flow rate. The accuracy assessment of the fan performance predictions hinted that this secondary flow may have a significant influence on fan operation. Further investigation is required in order to determine if the spiralling flow in the volute must be simulated as well.

A final area for future work regarding the existing simulation relates to the leakage around the ends of the rotor. The leakage model, and the value of K_{leak} used in the simulation have not been confirmed experimentally. A study of rotor leakage as a function of rotor/casing geometry, and fan flow rate should be performed.

7.2 MODELLING REVERSE FLOW

As was already mentioned, the largest improvement in the simulation's prediction accuracy can be attained by adding the modelling of reverse flow. The program developed in this thesis has been designed with the intention of using it as the core for a complete computer simulator, capable of modelling squirrel-cage fans operating with reverse flow present. Several programming steps have already been taken towards reverse flow capability, but major sections of code remain to be written. The program already calculates blading performance curves for all possible absolute inlet flow angles and flow rates. Thus, for any blading sector inlet flow angle, the rise in total pressure and exit whirl velocity can be found. This information, that is the coefficients for each of the quadratic curvefits, is stored in the simulation in the following arrays: exit whirl velocity curves in WRLCRV, Euler total pressure rise in EULCRV, and net total pressure rise in ACTCRV. Each array has 359 locations containing the three coefficients for the curvefit. The 359 array locations correspond to the 358 possible inlet flow angles, from -89° counterclockwise to 269° (see flow angle convention in Figure 3.3). A gap is left at 90° (and -90°) since some terms in the blading performance calculations are undefined at this inlet flow angle. The consequence of this is that an exactly zero blading through flow rate can not be calculated.

Another step towards reverse flow capability is the performance of the sudden enlargement compatibility model. This compatibility model predicts the existence of

reverse flow in parts of the rotor at below design point fan flow rates, through not finding solutions at these operating conditions. A scheme for calculating reverse flow thus presents itself, consisting of first assuming entirely forward flow around the fan, and calculating the resulting blading sector flow rates. If during the solution process, the sudden enlargement model does not find a compatible blading sector flow rate using the entire range of possible forward flow rates, the iteration process is repeated for the possible range of reverse flow rates. Once the first solution pass through all the fan sectors is complete, the calculated flow distribution around the fan is used to recalculate the entire fan again. For example, if a sector was found to have a reversed flow, then the flow exiting that sector into the inlet region is assumed to flow directly across the fan into whatever fan sector is opposite it, as found using the flow's exit angle. Any blading sectors that do not have flow entering them from other blading sectors with reverse flow are solved for in the usual way (ie. compatibility model). This process is repeated until the flow rates of all the sectors converge to a single value, completing the first performance curve data point.

Although the current sudden enlargement compatibility model is capable of predicting the existence of reverse flow, it is unable to select the proper blading sector flow rate. Once reverse flow is detected, a new compatibility model will have to be used, possibly the conservation of angular momentum model discussed in section 3.4.2. There is an obvious attraction to developing a universal compatibility model, capable of selecting both forward and reverse blade sector flow rates.

The largest section of code remaining to be written is the iteration loop that finds the final overall fan flow rate. As each pass through the fan is used to help calculate the next pass through, the blading sector flow rates, and the overall fan flow rate will hopefully converge to a unique value. A suitable criterion for this convergence is required, the simplest being the fan exit flow rate. After a solution pass through all the sectors is completed, this convergence criterion should be checked: a pass would indicate that the fan flow distribution is complete, and fan performance parameters are to be found; a fail would indicate that another pass is required. The loop "DO 90 L=1,MAXIT" that is currently found in the simulation code, is the loop that should be used. The only thing that remains to be added is the convergence check near the end of the loop. The value of MAXIT is currently set to one, so that the iteration loop is essentially non-active.

The other major section of code missing from the program required for modelling reverse flow is the "house-keeping" involved with determining which sectors receive their flow from sectors with reverse flow. The two arrays containing the blading element entry and exit flow conditions, BLADE1 and BLADE2, already have a variable entry for storing the inlet and exit flow angles. Before a main solution pass through all the sectors is to begin, each blading sector must be examined in turn to see if flow is entering it from blading sectors across the rotor from it, as calculated in the previous iteration. This should be a simple matter of geometry, as the sector's location is known, as well as the exit flow angle from opposite blading sectors. Some sort of weighted averaging scheme will have to be developed to account for inlet flow from more than one blading sector.

This entire house-keeping routine could be plugged in as a separate subroutine that is called from the main solution loop, right after the check for overall fan flow rate convergence is complete.

One final area of investigation may be the search for a non-iterative solution method. During the development of this program, an initial attempt was made at solving all of the fan sectors as a system of linear equations. This approach was initially unsuccessful, but since then many of the flow models used have changed, and it may be possible to reduce the amount of iteration used in the simulation by using this algebraic approach.

8.0 CONCLUSIONS

Based on the work presented in the preceeding chapters, several conclusions can be drawn:

Squirrel-cage type centrifugal fans are probably the most widely used air moving machine in use today. Surprisingly, very little research has been done on these turbomachines. Only research at Carleton University appears to address to some extent, the behaviour of the flow inside squirrel-cage fans, and its effects on fan performance. The existing research does strongly suggest that any computer model of squirrel-cage fan performance must address the many secondary flows that exist inside this type of fan.

The computer simulation developed in this thesis gave good performance predictions for the Rotor 1 Casing 2 fan geometry, at above design point flow rates. Only fair predictions were given for the other two test fan configurations. The static pressure distribution around the volute circumference was poorly predicted. Several major shortcomings in the simulation were identified, in an attempt to explain the shortcomings in the predictions. The most significant of these is the current treatment of flow in the area of the casing cut-off.

The sensitivity of the simulation program to the many parameters involved in its

operation are all reasonable, and as expected. The only exception is sensitivity to the gap dynamic pressure loss coefficient, which causes the simulation to return non-physical predictions when it has a value greater than 0.5. This problem is again linked to the current gap flow model. The changes in fan performance predicted by the simulation due to changes in fan geometry correspond well to that observed in experiments by other researchers.

In order to improve the understanding of how these fans operate, and to improve the simulation's accuracy, further experimental research is required. There are several flow phenomena that should be studied, some of which are: flow in the casing cut-off region, the mixing of blading exit flow with the volute flow, the spiralling volute flow, variation in the rotor separation region with location and fan flow rate, and rotor leakage. In addition to these specific areas of study, more overall fan performance measurements are needed for a larger range of fan geometries.

In order to simulate the squirrel-cage fan's performance at flow rates below design-point, reverse flow must be modelled. The sudden enlargement compatibility model already predicts the presence of reverse flow at low fan flow rates. The simulation program FANSIM has been written with the intention of making future modifications, permitting reverse flow to be modelled.

REFERENCES

- Alden, J.L. & Kane, J.M., 1970, **Design of Industrial Exhaust Systems**, 4th edition, Industrial Press Inc., New York.
- Brownell, R.B. Jr., & Flack, R.D. Jr., 1984, **Flow Characteristics in the Volute and Tongue Region of a Centrifugal Pump**, ASME Paper No. 84-GT-82
- Dixon, S.L., 1989, **Fluid Mechanics, Thermodynamics of Turbomachinery**, 3rd edition, Pergamon Press Ltd., Toronto.
- Eck, B., 1973, **Fans**, 1st edition, Pergamon Press Ltd., Toronto.
- Elder, R.L. & Gill, M.E., 1985, **A Discussion of the Factors Affecting Surge in Centrifugal Compressors**, ASME Journal of Engineering for Gas Turbines and Power, Vol. 107, pp. 497-506.
- Gessner, F.B., 1967, **An Experimental Study of Centrifugal Fan Inlet Flow and Its Influence on Fan Performance**, ASME Paper No. 67-FE-21.
- Kind, R.J., 1985, private communication.
- Kind, R.J. & Tobin, M.G., 1990, **Flow in a Centrifugal Fan of the Squirrel-Cage Type**, ASME Journal of Turbomachinery, Vol. 112.
- Kind, R.J., 1996, **Prediction of Flow Behaviour & Performance of Squirrel-Cage Centrifugal Fans Operating at Medium and High Flow Rates**, accepted for presentation at ASME Turbomachinery Expo, Birmingham, England.
- Lorett, J.A. & Gopalakrishnan, S., 1986, **Interaction Between Impeller and Volute of Pumps at Off-Design Conditions**, ASME Journal of Fluids Engineering, Vol. 108, pp. 12-18.

- Massey, B.S., 1980, **Mechanics of Fluids**, 4th edition, Van Nostrand Reinhold Co., New York.
- Osbourne, W.C., 1977, **Fans**, 2nd edition, Pergamon Press Ltd., Toronto.
- Pearson, C.E., 1986, **Numerical Methods in Engineering and Science**, Van Nostrand Reinhold Co., New York.
- Raj, D. & Swim, W.B., 1981, **Measurements of the Mean Flow Velocity and Velocity Fluctuations at the Exit of an FC Centrifugal Fan Rotor**, ASME Journal of Engineering for Power, Vol. 103, pp. 393-399.
- Sagi, R.S.C.M., 1990, **Computer Simulation for Aerodynamic Design & Analysis of Centrifugal Fans**, Master's Thesis, Department of Mechanical & Aerospace Engineering, Carleton University.
- Statistics Canada, 1991, **1991 Canadian Census**, Cat# 93-311.
- Statistics Canada, 1995, **Energy Statistics Handbook**, Cat# 57-601.
- Stepanoff, A.J., 1957, **Centrifugal and Axial Flow Pumps**, 2nd edition, John Wiley & Sons Inc.
- White, F.M., 1986, **Fluid Mechanics**, 2nd edition, McGraw-Hill Inc.
- Worster, R.C., 1963, **The Flow in Volute and Its Effect on Centrifugal Pump Performance**, Proceedings of the Institute of Mechanical Engineers, Vol. 177, pp. 843-875.
- Yaras, M.I., 1993, 88.435/541 Turbomachinery course notes, Department of Mechanical & Aerospace Engineering, Carleton University.

FIGURES

Figure 1.1 Geometry of the "Squirrel-Cage" Fan

Figure 1.2 Rotor Geometry Detail

parameter	Kind & Tobin (1990)			Raj & Swim (1981)	program variable name
	Rotor 1 Casing 1	Rotor 1 Casing 2	Rotor 2 Casing 1		
β'_1	0 deg.	0 deg.	60 deg.	6 deg.	BETAB1
β'_2	-65 deg.	-65 deg.	0 deg.	-64 deg.	BETAB2
B_B	0.152	0.152	0.152	0.142	BB
B_V	0.192	0.192	0.192	0.187	BV
R_{B1}	0.1125	0.1125	0.1145	0.113	RADB1
R_{B2}	0.1395	0.1395	0.1395	0.1345	RADB2
z	50	50	70	48	NUMBLD
R_{V1}	0.168	0.168	0.168	0.1695	RADV2(1)
R_{V2}	0.3336	0.4161	0.3336	0.3475	RADV2(2)

Length units in metres

Figure 2.1 Experimental Fan Test Geometries

Figure 2.2 Spiralling Secondary Flow in Volute

Figure 2.3 Visualization of Flow Patterns in Fan

Figure 2.4 Typical Variation in Volute Pressure with Location and Flow Rate

Tap No.	Static Pressure in Water
18	2.10
19	2.13
20	2.15

Wheel 12" dia x 4" wide, 8 blades

Outlet $10\frac{1}{4}" \times 4\frac{1}{8}"$

Nos. 23, 24, 25, & 26 are radial taps in scroll

All others are axial in side of casing -

Alden & Kane (1970, pp.197)

Figure 2.5 Measured Pressure Distribution Around Fan Operating at Below Design-Point Flow Rate

2 OF 2

PM-1 3 1/2 x 4 PHOTOGRAPHIC MICROCOPY TARGET
NBS 1010a ANSI/ISO #2 EQUIVALENT

PRECISIONSM RESOLUTION TARGETS

Figure 2.6 Reverse Flow Behaviour in Squirrel-Cage Fans, Kind & Tobin (1990)

Volute sector control volume boundary = abcde fgha

Figure 3.4 Volute Sector Control Volume

Figure 2.8 Kind & Tobin (1990) Measured Fan Performance

Figure 3.2 Imposed Axial Flow Rate Distributions

**Forward
Flow**

**Reverse
Flow**

Figure 3.3 Schematic of Typical Blading Velocity Triangles

Figure 3.4 Volute Sector Control Volume

$$\text{Total Pressure Loss} = 0.5\rho(C_1 - C_2)^2 \quad \text{Static Pressure Rise} = \rho C_2(C_1 - C_2)$$

$$C_1 A_1 = C_2 A_2$$

Figure 3.5 The Sudden Enlargement Flow

Figure 4.1 Simulation Flow Diagram

Main Code Variables		
Array Name	Array Dimensions	Data Stored
INLET1	(numi,numj,2)	static pressure, & flow rate in
INLET2	(numi,numj,3)	static pressure, flow rate, & flow angle out
BLADE1	(numi,numj,3)	static pressure, flow rate, & flow angle in
BLADE2	(numi,numj,5)	static pressure, flow rate, whirl velocity, radial velocity, & flow angle out
VOLUT1	(numj,3)	average static pressure, flow rate, & whirl velocity in
VOLUT2	(numj,3)	average static pressure, flow rate, & whirl velocity out
Geometry Variables		
Array Name	Array Dimensions	Data Stored
FANGE0	(numj,4)	entrance angle, exit angle, volute entrance radius, & volute exit radius
AREAS	(numi,numj,3,2)	entrance and exit area for inlet and exit of each 3 fan regions

Figure 4.2 FANSIM Main Program Arrays

Main Code	<div>variable declaration</div> <div>call GETDAT call HEADER</div> <div>call INIGEO</div> <div>call INIBLD</div> <div>main solution loop</div> <div>call OUTDAT</div>	<div>92</div> <div>Read in and check input data, then print as header to output file.</div> <div>Using θ_i & θ_j, discretize fan into elements.</div> <div>Use fan geometry data to calculate blading performance curves.</div> <div>Output solution to file with user defined name.</div>
Volute Models	<div>subroutine COMPAT</div> <div>subroutine CALCP2</div>	<div>Using blading sector guess and a compatibility relation, find the volute sector exit pressure.</div> <div>Calculate volute sector exit pressure using conservation of angular momentum.</div>
Blading Curves	<div>subroutine INIBLD</div> <div>subroutine BLDVAL</div> <div>subroutine REVBLO</div> <div>subroutine SLIPFA</div> <div>subroutine REVSLP</div> <div>subroutine LOSCAL</div>	<div>Create quadratic eq'ns for blading pressure rise vs. flow rate for a range of incidence angles.</div> <div>Calculate some basic blading geometric parameters.</div> <div>Same as BLDVAL except for reverse flow.</div> <div>Calculate blading exit flow angle.</div> <div>Same as SLIPFA except for reverse flow</div> <div>Use user selected loss model to find pressure losses in blading.</div>
Geometry/Flow Initialisation	<div>subroutine INIGEO</div> <div>subroutine DIVFLO</div>	<div>Using inputted θ_i & θ_j, discretize fan, and calculate fan element geometries.</div> <div>Using blading sector flow rate guess, find pressure, radial & whirl velocities, and flow angles for each blading element.</div>
Input/Output	<div>subroutine GETDAT</div> <div>subroutine HEADER</div> <div>subroutine OUTDAT</div>	<div>Read in and check input data.</div> <div>Print user checked input data to top of output file.</div> <div>Print final solution to output file.</div>
Misc. Subroutines	<div>subroutine FITCRV</div> <div>subroutine LINEAR</div> <div>function CURVE</div>	<div>Fit a least squares quadratic through 20 data points.</div> <div>Solve a $N \times N$ matrix using Gaussian elimination.</div> <div>Given the three coefficients for a quadratic, and a value for x, return y.</div>

Figure 4.3 FANSIM Code Layout

Figure 4.4 Main Solution Code Flow Chart

**Figure 5.1 Blading Total Pressure Rise Calculations:
Quadratic Curvefit vs. Correlation Results**

**Figure 5.2 Fan Performance Results for Rotor1 Casing1
Using the Two Compatibility Models**

**Figure 5.3 Fan Performance Results for Rotor1 Casing2
Using the Two Compatibility Models**

**Figure 5.4 Fan Performance Results for Rotor2 Casing1
Using the Two Compatibility Models**

"INRUN.DAT" data file entries for results in Chapter 5

Parameter	Value	Parameter	Value
NUM	10	IAXIAL	5
NUMJ	25	IEULER	3
TOLER	0.01	ISLIP	2
NUMM	10	IMODEL	see below
ISEGM	2	IOUT	1
ISECT	2		

"INLOSS.DAT" data file entries for results in Chapter 5

Simulation	IMODEL	CONLT	COLEAK	COEXP	COCOMP	COFRIC	COGAP	COEXT
Rotor1 Case1 (Figure 5.2) forced uniform flow sudden enlargement	1	0.1	0.0	0.0	0.0	0.003	0.0	0.98
	2	0.1	0.0	0.5	0.0	0.003	0.0	0.98
Rotor1 Case2 (Figure 5.3) forced uniform flow sudden enlargement	1	0.1	0.0	0.0	0.0	0.003	0.0	0.98
	2	0.1	0.0	0.7	0.0	0.003	0.0	0.98
Rotor2 Case1 (Figure 5.4) forced uniform flow sudden enlargement	1	0.1	0.0	0.0	0.0	0.003	0.0	0.98
	2	0.1	0.0	1.2	0.0	0.003	0.0	0.98
Pressure Distr. (Figure 5.6) forced uniform flow sudden enlargement	1	0.1	0.0	0.0	0.0	0.003	0.0	0.98
	2	0.1	0.0	0.7	0.0	0.003	0.0	0.98

N.B. definition of simulation parameters given in Appendix A

Figure 5.5 Simulation Data Used In Accuracy Assessment

Figure 5.6 Simulation Prediction of Volute Pressure Distribution at 100% and 150% Design Fan Flow Rate

Figure 6.1 Effect of Deviation on Blading Flow Turning

Figure 6.2 Effect of Blading Loss on Rotor Total Pressure Rise

**Figure 6.3 Effect of Blading Loss Models on Performance
Simulation of Rotor1 Casing1**

**Figure 6.4 Effect of Blading Loss Models on Performance
Simulation of Rotor2 Casing1**

Figure 6.5 Effect of Rotor Leakage on Simulation of Rotor 1 Casing 1 Using Uniform Flow Model

Figure 6.6 Effect of Compatibility Coefficient on Simulation of Rotor1 Casing1 Using Sudden Enlargement Model

Figure 6.7 Effect of Volute Wall Friction on Simulation of Rotor1 Casing1 Using Uniform Flow Model

Figure 6.8 Effect of Gap Dynamic Pressure Loss on Simulation of Rotor1 Casing1 Using Uniform Flow Model

Figure 6.9 Effect of Inlet Loss on Simulation of Rotor1 Casing1 Using Uniform Flow Model

Figure 6.10 Effect of Number of Sectors On Simulation of Rotor1 Casing1

Figure 6.11 Effect of Inlet Axial Flow Distribution on Simulation of Rotor1 Casing1 Using Uniform Flow Model

Figure 6.12 Effect of Blade Exit Angle on Simulation of Rotor1 Casing1 Using Uniform Flow Model

Figure 6.13 Effect of Gap Size on Simulated Fan Performance Using Uniform Flow Model

Figure 6.14 Effect of Casing Exit Area on Simulated Fan Performance Using Uniform Flow Model

Figure 6.15 Effect of Blade Chord on Simulated Fan Performance Using Uniform Flow Model

Figure 6.16 Effect of Rotor Exit Area/Fan Inlet Area on Simulated Fan Performance Using Uniform Flow Model and Equal Area Axial Flow Distribution

Figure 6.17 Summary of Fan Geometry's Effect on Simulated Performance

APPENDIX A - INPUT DATA FILES

File1: INGEO.DAT

Parameter	Sample Value	Description
BETAB1	0.0	Blade inlet angle
BETAB2	-65.0	Blade outlet angle
BB	0.152	Impeller width
BV	0.1920	Volute width
RADB1	0.1125	Impeller inner radius
RADB2	0.1395	Impeller outer radius
NUMBLD	50	No. of impeller blades
RADV2	0.168 0.3331	Volute Radial co-ordinates

File2: INRUN.DAT

Parameter	Sample Value	Description
NUMI	4	No. of slices, less than 25
NUMJ	30	No. of sectors, less than 100
TOLER	0.01	Iteration will continue until the % difference between successive values is less than this
NUMM	10	No. of fan performance curve points to find
ISEGM	1	Cosine slice size distribution-more at inlet
	2	Equal slice size distribution
	3	Sine slice size distribution-more at back
ISECT	1	Cosine sector size distribution-more at start
	2	Equal sector size distribution
	3	Sine sector size distribution-more at exit

```

      ALPH1R=REAL(A-90)*PI/180.
      DO 40 I=1,20
        CR1R=(-0.1*I)/A2
        CALL REVBOLD(ISLIP,CR1R,ALPH1R,SOVERC,INCID,BETA1R,BETA2R,
1          SLIP)
        CW2R=SLIP*(RADB1/RADB2-(RADB2/RADB1)*CR1R*TAN(BETA2R))
        XYW(I,1)=CR1R
        XYW(I,2)=CW2R
        CW1R=1.0+CR1R*TAN(BETA1R)
        DP=(RADB1/RADB2)*CW2R-CW1R
        XYE(I,1)=CR1R
        XYE(I,2)=DP
        CALL LOSCAL(REVERS,IEULER,INCID,BETA1R,BETA2R,SOVERC,CR1R,
1          DP,PLOSS)
        DP=DP-PLOSS
        XY(I,1)=CR1R
        XY(I,2)=DP
40      CONTINUE
        CALL FITCRV(A+180,XYW,WRLCRV)
        CALL FITCRV(A+180,XYE,EULCRV)
        CALL FITCRV(A+180,XY,ACTCRV)
30      CONTINUE
        RETURN
      END

```

*
 * the primary blading performance variables are calculated, including:
 * S/C, INCIDENCE, BETA1, BETA2, SLIP FACTOR
 *

```

      SUBROUTINE BLDVAL(ISLIP,CR2,ALPHA1,SOVERC,INCID,BETA1,BETA2,SLIP)
      PARAMETER(PI=3.141592654)
      REAL BETAB1,BETAB2,BB,BV,RADB1,RADB2,RADV2(2),RADDMM,STGR,S,C,
1      SOVERC,ALPHA1,BETA1,BETA2,INCID,CR2
      INTEGER NUMBLD,ISLIP
      *
      COMMON /GEOMET/ BETAB1,BETAB2,BB,BV,RADB1,RADB2,RADV2,NUMBLD
      * calculate blade row geometry data for correlations
      RADDMM=SQRT(RADB1*RADB2)
      STGR=(BETAB1+BETAB2)/2.
      S=2.*PI*RADDMM/NUMBLD
      C=(RADB2-RADB1)/COS(STGR)
      SOVERC=S/C
      BETA1=ATAN(RADB1**2/(RADB2**2*CR2)-TAN(ALPHA1))
      INCID=BETA1-BETAB1
      CALL SLIPFA(ISLIP,INCID,SOVERC,SLIP,BETA2)
      RETURN
      END

```

*
 * the blading variables for REVERSE FLOW CASE are calculated, including:
 * INCIDENCE, BETA1, BETA2, SLIP FACTOR
 *

```

      SUBROUTINE REVBOLD(ISLIP,CR1R,ALPH1R,SOVERC,INCID,BETA1R,BETA2R,
1      SLIP)
      PARAMETER(PI=3.141592654)
      REAL BETAB1,BETAB2,BB,BV,RADB1,RADB2,RADV2(2),SOVERC,ALPH1R,
1      BETA1R,BETA2R,INCID,CR1R
      INTEGER NUMBLD,ISLIP
      *
      COMMON /GEOMET/ BETAB1,BETAB2,BB,BV,RADB1,RADB2,RADV2,NUMBLD
      BETA1R=ATAN(TAN(ALPH1R)-1./CR1R)
      INCID=BETA1R-(-1*BETAB2)
      CALL REVSLP(ISLIP,INCID,SOVERC,SLIP,BETA2R)
      RETURN

```

APPENDIX B - PROGRAM CODE

*** FANSIM ***

Carleton University, Ottawa, CANADA

March 1996

R.J. Kind, J.C. Thompson

A centrifugal, "squirrel-cage" fan performance simulation program, under development since 1989 under the name CENTFAN.

Input data is given from three separate input files:

File1: INGEO - For fan geometry
File2: INRUN - For fan run condition parameters
File3: INLOSS - For fan loss model coefficients

See FANSIM.DOC for details on the input file formats. Note that input data is in standard SI units, unless otherwise noted.

```
PROGRAM FANSIM
PARAMETER (PI=3.141592654,IMAX=25,JMAX=100,MAXORD=5,QMAX=1.0,
1          GAPMAX=0.5,MAXIT=1)
```

IMAX= maximum number of slices allowed
JMAX= maximum number of sectors allowed
MAXORD= maximum order of blading calc. curve fit allowed
QMAX= maximum overall fan flow rate allowed (non-dimensional)
GAPMAX= maximum gap flow rate allowed (non-dimensional)
MAXIT= maximum number of iterations allowed

IMPLICIT LOGICAL (A-Z)

* the fan geometry data...(see FANSIM.DOC)
REAL BETAB1,BETAB2,BB,BV,RADB1,RADB2,RADV2(2)
INTEGER NUMBLD

* the fan simulation running parameters...(see FANSIM.DOC)
REAL TOLER
INTEGER NUMI,NUMJ,NUMM

* fan component loss parameters...(see FANSIM.DOC)
REAL COINLT,COLEAK,COEXP,COCOMP,COFRIC,COGAP,COEXT

* program option switches...(see FANSIM.DOC)
INTEGER ISEGM,ISECT,IAXIAL,IEULER,ISLIP,IMODEL,IOUT

* fan flow variables to be carried through the simulation...(see FANSIM.DOC)
REAL INLET1(IMAX,JMAX,2),INLET2(IMAX,JMAX,3),BLADE1(IMAX,JMAX,3),
1 BLADE2(IMAX,JMAX,5),VOLUT1(JMAX,3),VOLUT2(JMAX,3)

* final fan overall performance parameters...
REAL FLOW,STATIC,TOTAL,DYNAM,EFFIC,BLADE(IMAX,JMAX,7),
1 VOLUTE(JMAX,8)
LOGICAL GAP,SECT(JMAX)

* FLOW= overall fan flow rate

```

*      STATIC= overall fan exit static pressure
*      TOTAL= overall fan exit total pressure
*      DYNAM= overall fan exit dynamic pressure
*      EFFIC= overall fan efficiency
*      BLADE(imax,jmax,7)= flow rate, radial & whirl velocity,
*                          static & total pressure, volute static
*                          pressure at blade exit, adjusted whirl
*                          velocity -- all at blading element exit
*      VOLUTE(jmax,8)= flow rate, whirl velocity, static & total
*                      pressure -- at volute sector inlet & exit
*      GAP= 'true' if gap flow was compatible, 'false' if not
*           compatible
*      SECT(JMAX)= 'true' if blading sector j flow rate was
*                 compatible, 'false' if flow rate not compatible
*
* other variables used by the program...
  REAL FANGEO(JMAX,4),AREAS(IMAX,JMAX,3,2),TANPHI,PB2AVG,WH'RL,
1    RADIAL,ACTCRV(359,MAXORD+1),WRLCRV(359,MAXORD+1),CURV',
2    EULCRV(359,MAXORD+1),QBLADE,DP,POUT,LOWERQ,EULER,PGAP,R,
3    T1,STEP,LOWRDP,DGP,QSTART,LOWERG,GSTEP,LOWDGP,GSTART,
4    NEWRL(IMAX,JMAX,2),DQ,LASTQ,LASTG,DG,RC,ALPHA
  INTEGER I,J,K,L,M,N,DPSIGN,GSIGN
  CHARACTER *8 OUTFIL
*
  COMMON /GEOMET/ BETAB1,BETAB2,BB,BV,RADB1,RADB2,RADV2,NUMBLD
  COMMON /GENRL/  NUMI,NUMJ,TOLER,NUMM
  COMMON /LOSSES/ COINLT,COLEAK,COEXP,COCOMP,COFRIC,COGAP,COEXT
  COMMON /SWITCH/ ISEGM,ISECT,IAAXIAL,IEULER,ISLIP,IMODEL,IOUT
  COMMON /FLOW/  INLET1,INLET2,BLADE1,BLADE2,VOLUT1,VOLUT2
  COMMON /CALGEO/ FANGEO,AREAS,TANPHI
  COMMON /SOLUTN/ FLOW,STATIC,TOTAL,DYNAM,EFFIC,BLADE,VOLUTE,
1    GAP,SECT
*
  PRINT *, '                      *** FANSIM ***'
  PRINT *
* read in and check all input data
  CALL GETDAT
  PRINT *
  PRINT *, 'Enter a name for the output file (8 chars)'
  READ (*,5)OUTFIL
5  FORMAT (A8)
  OPEN (UNIT=4,FILE=OUTFIL//''.out')
* overall fan performance calc. and simulation data stored in UNIT=4,
* filename.OUT
  IF(IOUT.EQ.2) THEN
    OPEN (UNIT=10,FILE=OUTFIL//''.bld')
* blading sector flow data stored in UNIT=10, filename.BLD
    OPEN (UNIT=11,FILE=OUTFIL//''.vol')
* volute sector flow data stored in UNIT=11, filename.VOL
  ENDIF
* print checked input data to top of filename.OUT
  CALL HEADER(OUTFIL)
* convert the inputed blade angles to radians
  BETAB1=BETAB1*PI/180
  BETAB2=BETAB2*PI/180
* use inputed data to setup fan element geometry
  CALL INIGEO(ISEGM,ISECT,IAAXIAL,NUMI,NUMJ)
* calculate blading performance curves ACTCRV, EULCRV, & WRLCRV
*      ACTCRV= net total pressure rise (incl. losses)
*      EULCRV= Euler (no losses) total pressure rise
*      WRLCRV= blade exit whirl velocity
  CALL INIBLD(IEULER,ISLIP,ACTCRV,EULCRV,WRLCRV)

```

```

      PRINT *
      PRINT *, 'STARTING SOLUTION...'
*
***** MAIN ITERATION LOOP *****
*
* repeat for required number of performance curve data points
      DO 10 M=1, NUMM
* iterate 'til fan flowrate for current data point is stationary
* N.B. this loop is to be used in the reverse flow version of program
      DO 90 L=1, MAXIT
* init. start point, end point, & step size for gap flow rate iteration
      LOWDGP=10.0
      GSTART=0.01
      GSTEP=(GAPMAX-GSTART)/10.
110      CONTINUE
      LASTG=10.0
      GAP=.FALSE.
* pick gap flowrates until gap flow criteria are met
      DO 20 N=1, 11
          IF(N.NE.1) LASTG=VOLUT1(1,2)
          VOLUT1(1,2)=GSTART+GSTEP*(N-1)
          VOLUT1(1,3)=VOLUT1(1,2)/AREAS(1,1,3,1)
* pick a 1st blading sector flow rate to get solution started
          SECT(1)=.TRUE.
* flow rate picked by dividing max. fan flow rate range evenly between
* no. of data points to find, then the first blading sector is given its
* share, equal to the percentage of the rotor's circumference that it
* takes up
          QBLADE=QMAX*M/NUMM*(FANGE0(1,2)-FANGE0(1,1))/
              1          (2.*PI)
* divide blade sector flow rate up amongst slices, and calc. flow rate
* averaged exit whirl & radial velocity, and exit static pressure
          CALL DIVFLO(IAXIAL, COINLT, ACTCRV, WRLCRV, QBLADE, 1,
              1          WHIRL, RADIAL, PB2AVG)
* apply continuity to get volute sector exit flow rate
          VOLUT2(1,2)=VOLUT1(1,2)+(1.-COLEAK)*QBLADE
          VOLUT2(1,3)=VOLUT2(1,2)/AREAS(1,1,3,2)
* assume Pv1 @RADB2 equals Pb2, to get Pv2
          RC=(FANGE0(1,3)+RADB2)/(2.*RADB2)
          VOLUT1(1,1)=PB2AVG+0.5*VOLUT1(1,3)**2*(1.+TANPHI**2)
              1          *(RC**2-1.)
* knowing all flow parameters into volute sector, find exit static
* pressure
          CALL CALCP2(NUMI,1,NEWRL)
* now, step through each fan sector, solving the flow as we go
          DO 30 J=2, NUMJ
              VOLUT1(J,1)=VOLUT2(J-1,1)
              VOLUT1(J,2)=VOLUT2(J-1,2)
              VOLUT1(J,3)=VOLUT2(J-1,3)
* initialize start point, end point, and step size for blade sector flow
* rate iteration
              QSTART=QMAX/5.
              STEP=(QSTART-0.000001)/100.
* end point of 0.000001 used to avoid strange model behaviour at zero
80          CONTINUE
          LASTQ=10.0
          SECT(J)=.FALSE.
* iterate through blade sector flowrates until one passes
          DO 40 K=1, 101
              IF(J.NE.1) LASTQ=QBLADE
              IF(IMODEL.NE.1) THEN
                  QBLADE=QSTART-STEP*(K-1)

```

```

ELSE
* if using forced uniform flow model, set all sector flow rates the same
  QBLADE=QBLADE*(FANGE0(J,2)-FANGE0(J,1))/(
    1      (FANGE0(J-1,2)-FANGE0(J-1,1))
  )
  ENDIF
  VOLUT2(J,2)=VOLUT1(J,2)+(1.-COLEAK)*QBLADE
  VOLUT2(J,3)=VOLUT2(J,2)/AREAS(1,J,3,2)
  CALL DIVFLO(IAxIAL,COINLT,ACTCRV,WRLCRV,QBLADE,
    1      J,WHIRL,RADIAL,PB2AVG)
* calculate Pv2 using compatability & ang. momentum and compare...
  CALL CALCP2(NUMI,J,NEWRL)
  CALL COMPAT(IMODEL,J,QBLADE,WHIRL,RADIAL,PB2AVG,
    1      POUT)
* if forcing uniform blading flow around fan, skip past iteration
  IF(IMODEL.EQ.1) GOTO 150
  DP=VOLUT2(J,1)-POUT
  DQ=ABS(QBLADE-LASTQ)
  IF(K.EQ.1) THEN
* if this is the first run through iteration, set all starting values
    DPSIGN=INT(SIGN(1.,DP))
    LOWRDP=DP
    LOWERQ=QBLADE
    ELSEIF(ABS(DP*10./VOLUT2(J,1)).LE.TOLER.OR.
      1      (DQ*1000./QBLADE).LE.TOLER) THEN
* check to see if difference in volute exit pressure and blade sector
* flow rate from the last iteration is smaller than % tolerance
* N.B. the multipliers on DP and DQ above are to ensure that iteration
* with the difference smaller than TOLER is picked, and that the
* matching of the pressure is more important
    SECT(J)=.TRUE.
    GOTO 30
    ELSEIF(DP.LT.0.0.AND.DPSIGN.GT.0.0.OR.
      1      DP.GT.0.0.AND.DPSIGN.LT.0.0) THEN
* check if volute exit pressure curves have crossed each other, if so,
* go back a step, and refine search by factor of 10
    QSTART=QBLADE+1.0*STEP
    STEP=STEP/10.
    GOTO 80
    ELSEIF(ABS(DP).LT.ABS(LOWRDP)) THEN
* if the current iteration is closer to being compatible than the last,
* save it
    LOWRDP=DP
    LOWERQ=QBLADE
    ELSEIF(K.EQ.101) THEN
* if no solution found, use closest (ie. Q giving smallest DP)
    QBLADE=LOWERQ
    VOLUT2(J,2)=VOLUT1(J,2)+(1.-COLEAK)*QBLADE
    VOLUT2(J,3)=VOLUT2(J,2)/AREAS(1,J,3,2)
    CALL DIVFLO(IAxIAL,COINLT,ACTCRV,WRLCRV,QBLADE,
      1      J,WHIRL,RADIAL,PB2AVG)
    CALL CALCP2(NUMI,J,NEWRL)
  )
  ENDIF
40      CONTINUE
150      CONTINUE
30      CONTINUE
* now that run through sectors is done, check if gap flow guess ok
  R=(FANGE0(NUMJ,4)+RADB2)/(FANGE0(1,3)+RADB2)
  T1=0.5*COGAP*((VOLUT2(NUMJ,3)*R)**2-VOLUT1(1,3)**2)
  PGAP=VOLUT2(NUMJ,1)+0.5*(VOLUT2(NUMJ,3)**2
    1      -VOLUT1(1,3)**2)*(1.+TANPHI**2)-T1*(1.+TANPHI**2)
* calc. gap static pressure based on constant total pressure across
* cut-off

```

```

DGP=PGAP-VOLUT1(1,1)
DG=ABS(LASTG-VOLUT1(1,2))
IF(ABS(DGP/VOLUT1(1,1)).LE.TOLER.OR.
1  (DG*1000./VOLUT1(1,2)).LE.TOLER) THEN
* check to see if difference in gap pressure or gap flow rate from the
* last iteration is smaller than % tolerance
* N.B. the multipliers on DGP and DG above are to ensure that iteration
* with the difference smaller than TOLER is picked, and that the
* matching of the pressure is more important
GAP=.TRUE.
DO 50 J=1,NUMJ
DO 60 I=1,NUMI
* gap flow matches, so store all the calculated fan element flow
* parameters
BLADE(I,J,1)=BLADE2(I,J,2)
BLADE(I,J,2)=BLADE2(I,J,4)
BLADE(I,J,3)=BLADE2(I,J,3)
BLADE(I,J,4)=BLADE2(I,J,1)
BLADE(I,J,5)=BLADE2(I,J,1)+0.5*(BLADE2(I,J,3)**2
1 +BLADE2(I,J,4)**2)
BLADE(I,J,6)=NEWRL(I,J,1)
BLADE(I,J,7)=NEWRL(I,J,2)
60 CONTINUE
VOLUTE(J,1)=VOLUT1(J,2)
VOLUTE(J,2)=VOLUT1(J,3)
VOLUTE(J,3)=VOLUT1(J,1)
VOLUTE(J,4)=VOLUT1(J,1)+0.5*VOLUT1(J,3)**2*
1 (1.+TANPHI**2)
VOLUTE(J,5)=VOLUT2(J,2)
VOLUTE(J,6)=VOLUT2(J,3)
VOLUTE(J,7)=VOLUT2(J,1)
VOLUTE(J,8)=VOLUT2(J,1)+0.5*VOLUT2(J,3)**2*
1 (1.+TANPHI**2)
50 CONTINUE
* calculate overall fan flow rate and total pressure rise
FLOW=VOLUT2(NUMJ,2)-VOLUT1(1,2)
TOTAL=VOLUTE(NUMJ,8)
GOTO 100
ELSEIF(N.EQ.1) THEN
* if this is the first run through iteration, set all starting values
GSIGN=INT(SIGN(1.,DGP))
LOWDGP=DGP
LOWERG=VOLUT1(1,2)
ELSEIF(DGP.LT.0.0.AND.GSIGN.GT.0.0.OP.
1 DGP.GT.0.0.AND.GSIGN.LT.0.0) THEN
* check if DGP has changed sign (ie. solution point crossed), if so,
* go back a step, and refine search by factor of 10
GSTART=VOLUT1(1,2)-1.0*GSTEP
GSTEP=GSTEP/10.
GOTO 110
ENDIF
IF(ABS(DGP).LE.ABS(LOWDGP)) THEN
* if the current iteration is closer to being compatible than the last,
* save it
LOWDGP=DGP
LOWERG=VOLUT1(1,2)
DO 55 J=1,NUMJ
DO 65 I=1,NUMI
BLADE(I,J,1)=BLADE2(I,J,2)
BLADE(I,J,2)=BLADE2(I,J,4)
BLADE(I,J,3)=BLADE2(I,J,3)
BLADE(I,J,4)=BLADE2(I,J,1)

```

```

1          BLADE(I,J,5)=BLADE2(I,J,1)+0.5*(BLADE2(I,J,3)**2
          +BLADE2(I,J,4)**2)
          BLADE(I,J,6)=NEWRL(I,J,1)
          BLADE(I,J,7)=NEWRL(I,J,2)
65      CONTINUE
          VOLUTE(J,1)=VOLUT1(J,2)
          VOLUTE(J,2)=VOLUT1(J,3)
          VOLUTE(J,3)=VOLUT1(J,1)
          VOLUTE(J,4)=VOLUT1(J,1)+0.5*VOLUT1(J,3)**2*
1          (1.+TANPHI**2)
          VOLUTE(J,5)=VOLUT2(J,2)
          VOLUTE(J,6)=VOLUT2(J,3)
          VOLUTE(J,7)=VOLUT2(J,1)
          VOLUTE(J,8)=VOLUT2(J,1)+0.5*VOLUT2(J,3)**2*
1          (1.+TANPHI**2)
55      CONTINUE
          FLOW=VOLUT2(NUMJ,2)-VOLUT1(1,2)
          TOTAL=VOLUTE(NUMJ,8)
          ENDIF
20      CONTINUE
100     CONTINUE
90      CONTINUE
* after iterations are complete, calc. overall performance
* characteristics
      T1=FLOW/(AREAS(1,NUMJ,3,2)-AREAS(1,1,3,1))
      DYNAM=0.5*T1**2*(1.+TANPHI**2)
      STATIC=TOTAL-DYNAM
      EULER=0.0
      DO 70 I=1,NUMI
        DO 70 J=1,NUMJ
          IF(BLADE2(I,J,4).LT.0.0) ALPHA=BLADE1(I,J,3)+270.
          IF(BLADE2(I,J,4).GE.0.0) ALPHA=BLADE1(I,J,3)+90
* convert inlet flow angle to 360deg. system so proper blade performance
* curve can be accessed, depending on forward or reverse flow
70      EULER=EULER+BLADE(I,J,1)*CURVE(EULCRV,BLADE(I,J,2),
1      ALPHA)
          EFFIC=COEXT*TOTAL*FLOW/EULER
* calc. efficiency = (fan total pressure rise*fan flow rate)/sum of
* (blade element total pressure rise*element flow rate)
          CALL OUTDAT(IOUT)
* print results to data files
          WRITE(*,75)'Done perf. curve data point #',M,' of ',NUMM
75      FORMAT(A,I2,A,I2)
10      CONTINUE
          CLOSE(4)
          IF(IOUT.EQ.2) THEN
            CLOSE(10)
            CLOSE(11)
          ENDIF
          PRINT*,'ALL DONE!'
          STOP
          END
***** END MAIN CODE *****
*
*****
*
*          MAIN CODE FLOW MODEL SUBROUTINES
*
*****
* using the sector flowrate guess, calculate the volute outlet pressure
* using the compatability relationship
*
      SUBROUTINE COMPAT(IMODEL,J,Q,WHIRL,RADIAL,PB2AVG,POUT)

```

```

PARAMETER (PI=3.141592654, IMAX=25, JMAX=100)
REAL TOLER, BETAB1, BETAB2, BB, BV, RADB1, RADB2, RADV2(2), TANPHI,
1 COINLT, COLEAK, COCOMP, COFRIC, COGAP, COEXT, POUT, WHIRL,
2 FANGEQ(JMAX, 4), AREAS(IMAX, JMAX, 3, 2), INLET1(IMAX, JMAX, 2), Q,
3 INLET2(IMAX, JMAX, 3), BLADE1(IMAX, JMAX, 3), VOLUT1(JMAX, 3),
4 VOLUT2(JMAX, 3), BLADE2(IMAX, JMAX, 5), RA, RC, RE, RG, DC, FB, BA, HA,
5 RADIAL, T1, T2, T3, T4, T5, PB2AVG, COEXP, CA, AREAF, AREAB, PTB2,
6 T6, T7, T8, NEWRAD
INTEGER IMODEL, NUMBLD, NUMI, NUMJ, NUMM, J
COMMON /GENRL/ NUMI, NUMJ, TOLER, NUMM
COMMON /LOSSES/ COINLT, COLEAK, COEXP, COCOMP, COFRIC, COGAP, COEXT
COMMON /FLOW/ INLET1, INLET2, BLADE1, BLADE2, VOLUT1, VOLUT2
COMMON /GEOMET/ BETAB1, BETAB2, BB, BV, RADB1, RADB2, RADV2, NUMBLD
COMMON /CALGEO/ FANGEQ, AREAS, TANPHI
IF(IMODEL.EQ.1) THEN
* force equal blading sector flow around entire fan
ELSEIF(IMODEL.EQ.2) THEN
* use sudden enlargement model for the compatibility equation
RA=(FANGEQ(J,4)+RADB2)/(2.*RADB2)
RC=(FANGEQ(J,3)+RADB2)/(2.*RADB2)
T1=0.5*VOLUT1(J,3)**2*(1.+TANPHI**2)*(1.-RC**2)
T2=0.5*VOLUT2(J,3)**2*(1.+TANPHI**2)*(1.-RA**2)
T3=SQRT((1.+TANPHI**2)*(VOLUT1(J,3)*RC+VOLUT2(J,3)*RA)**2/4.)
T4=SQRT(WHIRL**2+RADIAL**2)
T5=2.*T3*(T4-T3)-COEXP*(T4**2-T3**2)
POUT=-VOLUT1(J,1)-T1-T2+2.*PB2AVG+T5
ELSEIF(IMODEL.EQ.3) THEN
* use conserv. of ang. momentum in volute about 'a' for compat. eq'n
* N.B. this model DOES NOT WORK in its current form, and should be
* replaced completely with the correct one
RA=(FANGEQ(J,4)+RADB2)/(2.*RADB2)
RC=(FANGEQ(J,3)+RADB2)/(2.*RADB2)
RG=(FANGEQ(J,4)+RADB2)/(2.*FANGEQ(J,3))
RE=(FANGEQ(J,3)+RADB2)/(2.*FANGEQ(J,3))
DC=(FANGEQ(J,3)-RADB2)/2.
T1=(FANGEQ(J,3)+FANGEQ(J,4))/2.
FB=T1-RADB2
BA=(FANGEQ(J,2)-FANGEQ(J,1))*RADB2/2.
HA=(FANGEQ(J,4)-RADB2)/2.
CA=2.*BA
AREAF=(FANGEQ(J,2)-FANGEQ(J,1))*BV*T1/(4.*RADB2**2)
AREAB=(FANGEQ(J,2)-FANGEQ(J,1))*BB*RADB2/(4.*RADB2**2)
* calc. sudden enlargement adjustments to blading flow
T1=SQRT((1.+TANPHI**2)*(VOLUT1(J,3)*RC+VOLUT2(J,3)*RA)**2/4.)
T2=SQRT(WHIRL**2+RADIAL**2)
PTB2=PB2AVG+0.5*T2**2-0.5*((T2-T1)*COEXP)**2
PA=VOLUT2(J,1)+0.5*VOLUT2(J,3)**2*(1.+TANPHI**2)*(1.-RA**2)
PC=VOLUT1(J,1)+0.5*VOLUT1(J,3)**2*(1.+TANPHI**2)*(1.-RC**2)
T3=(PA+PC)/2.
T4=RADIAL/WHIRL
T5=(PTB2-T3)/(0.5*(1.+T4**2))
IF(T5.LT.0.0) T5=0.0
NEWRAD=SQRT(T5)*T4
* calc. terms of ang. momentum eq'n, and find Pv2
T1=AREAS(1,J,3,1)*DC+0.5*(AREAS(1,J,3,2)-AREAS(1,J,3,1))*FB
1 -AREAF*BA
T2=-AREAS(1,J,3,2)*HA+0.5*(AREAS(1,J,3,2)-AREAS(1,J,3,1))*FB
1 -AREAF*BA
T3=0.25*VOLUT1(J,3)**2*(1.+TANPHI**2)*(1.-RE**2)
1 *((AREAS(1,J,3,2)-AREAS(1,J,3,1))*FB-AREAF*BA)
T4=0.25*VOLUT2(J,3)**2*(1.+TANPHI**2)*(1.-RG**2)
1 *((AREAS(1,J,3,2)-AREAS(1,J,3,1))*FB-AREAF*BA)

```



```

      T5=0.5*(PA+PC)*AREAB*BA
      T6=VOLUT1(J,2)*VOLUT1(J,3)*(DC+TANPHI*CA)
      T7=(1.-COLEAK)*Q*NEWRAD*BA
      T8=VOLUT2(J,2)*VOLUT2(J,3)*HA
      POUT=(-VOLUT1(J,1)*T1-T3-T4-T5-T6-T7+T8)/T2
    ENDIF
    RETURN
  END

*
* using the blade sector flowrate and Pfl, calculate the volute outlet
* pressure using the conservation of angular momentum equation
*
  SUBROUTINE CALCP2(NUMI,J,NEWRL)
    PARAMETER (IMAX=25,JMAX=100,MAXIT=50)
    DIMENSION NEWRL(IMAX,JMAX,2)
    REAL BETAB1,BETAB2,BB,BV,RADB1,RADB2,RADV2(2),NEWRL,TANPHI,T1,
1      COINLT,COLEAK,COEXP,COCOMP,COFRIC,COGAP,COEXT,FRICT,
2      FANGE0(JMAX,4),AREAS(IMAX,JMAX,3,2),INLET1(IMAX,JMAX,2),
3      INLET2(IMAX,JMAX,3),BLADE1(IMAX,JMAX,3),VOLUT1(JMAX,3),T2,
4      VOLUT2(JMAX,3),BLADE2(IMAX,JMAX,5),PTB2,T3,T4,T5,
5      RADD,RADH,RADI,RADF,RC,RE,RG
    INTEGER NUMBLD,J
    COMMON /FLOW/ INLET1,INLET2,BLADE1,BLADE2,VOLUT1,VOLUT2
    COMMON /LOSSES/ COINLT,COLEAK,COEXP,COCOMP,COFRIC,COGAP,COEXT
    COMMON /GEOMET/ BETAB1,BETAB2,BB,BV,RADB1,RADB2,RADV2,NUMBLD
    COMMON /CALGEO/ FANGE0,AREAS,TANPHI

*
* calc. various radii and radius ratios around the volute sector
  RADD=(FANGE0(J,3)+RADB2)/2.
  RADH=(FANGE0(J,4)+RADB2)/2.
  RADF=(FANGE0(J,3)+FANGE0(J,4))/2.
  RADI=(RADD+RADH)/2.
  RC=RADD/RADB2
  RE=RADD/FANGE0(J,3)
  RG=RADH/FANGE0(J,4)

* determine blading whirl velocity after flow dumps into volute
  SUMB2=0.0
  DO 10 I=1,NUMI
    PTB2=BLADE2(I,J,1)+0.5*(BLADE2(I,J,3)**2+BLADE2(I,J,4)**2)
    NEWRL(I,J,1)=VOLUT1(J,1)+0.5*VOLUT1(J,3)**2*(1.+TANPHI**2)
1    *(1.-RC**2)
* NEWRL(I,J,1) is the volute static pressure at the blade exit radius
    T1=BLADE2(I,J,4)/BLADE2(I,J,3)
    T2=(PTB2-NEWRL(I,J,1))/(0.5*(1.+T1**2))
    IF(T2.LT.0.0) T2=0.0
    NEWRL(I,J,2)=SQRT(T2)

* NEWRL(I,J,2) is the blade exit whirl velocity req'd for the static
* pressure at blade exit to be compatible with the volute flow, and
* still conserve blade exit flow total pressure
    SUMB2=SUMB2+(1.-COLEAK)*BLADE2(I,J,2)*NEWRL(I,J,2)
10  CONTINUE

* calculate the volute wall friction term
    T1=(VOLUT1(J,3)+VOLUT2(J,3))/2.
    T2=T1*RADI/RADF
    T3=BV*(FANGE0(J,2)-FANGE0(J,1))*RADF/(4.*RADB2**2)
    T4=0.5*(FANGE0(J,2)-FANGE0(J,1))*(FANGE0(J,3)**2
1    +FANGE0(J,4)**2-2.*RADB2**2)/(4.*RADB2**2)
    FRICT=0.5*COFRIC*(T1**2*T4*RADI+T2**2*T3*RADF)

* calculate the volute sector exit pressure
    T1=(AREAS(1,J,3,2)-AREAS(1,J,3,1))*RADF/2.
    T2=T1+AREAS(1,J,3,1)*RADD
    T3=T1/2.*(1.+TANPHI**2)*(1.-RE**2)+AREAS(1,J,3,1)*RADD

```

```

T4=T1/2.*(1.+TANPHI**2)*(1.-RG**2)-AREAS(1,J,3,2)*RADH
T5=T1-AREAS(1,J,3,2)*RADH
VOLUT2(J,1)=(-VOLUT1(J,1)*T2-VOLUT1(J,3)**2*T3-VOLUT2(J,3)**2
1      *T4-SUMB2*RADB2+FRICT)/T5
RETURN
END
*****
*      IMPELLER PERFORMANCE CURVE GENERATING SUBROUTINES      *
*****
* this subroutine calculates the pressure rise across the given rotor
* for a series of 20 flowrates, and then fits a curve to them for use as
* a function call later in the program (see CURVE). The resulting curve
* will be non-dimensional exit radial velocity versus exit total
* pressure rise, Euler total pressure rise, or exit whirl velocity.
* Curves for absolute inlet flow angles (alpha1) from -89 to 89 deg.(fwd
* flow) and 91 to 269 deg. (rev flow) are found.
*
SUBROUTINE INIBLD(IEULER,ISLIP,ACTCRV,EULCRV,WRLCRV)
PARAMETER(PI=3.141592654,MAXORD=5)
DIMENSION ACTCRV(359,MAXORD+1),EULCRV(359,MAXORD+1),
1      WRLCRV(359,MAXORD+1)
REAL BETAB1,BETAB2,BB,BV,RADB1,RADB2,RADV2(2),WRLCRV,ACTCRV,
1      EULCRV,XY(20,2),XYE(20,2),SOVERC,ALPHA1,BETA1,BETA2,
2      INCID,DP,A2,XYW(20,2),CR2,CW1,CW2,ALPH1R,BETA1R,BETA2R,
3      CR1R,CW1R,CW2R
INTEGER NUMBLD,IEULER,ISLIP,A,I
LOGICAL REVERS
*
COMMON /GEOMET/ BETAB1,BETAB2,BB,BV,RADB1,RADB2,RADV2,NUMBLD
PRINT *
PRINT *, 'COMPUTING IMPELLER PERFORMANCE CURVES...'
IF (IEULER.EQ.1) ISLIP=0
A2=PI*BB/(2.*RADB2)
* calculate impeller perf. for a range of flowrates and inlet flow
* angles
REVERS=.FALSE.
DO 10 A=1,179
ALPHA1=REAL(A-90)*PI/180.
DO 20 I=1,20
CR2=(0.1*I)/A2
CALL BLDVAL(ISLIP,CR2,ALPHA1,SOVERC,INCID,BETA1,BETA2,SLIP)
CW2=SLIP*(1.-CR2*TAN(BETA2))
XYW(I,1)=CR2
XYW(I,2)=CW2
CW1=(RADB1/RADB2)-(RADB2/RADB1)*CR2*TAN(BETA1)
DP=CW2-(RADB1/RADB2)*CW1
XYE(I,1)=CR2
XYE(I,2)=DP
CALL LOSCAL(REVERS,IEULER,INCID,BETA1,BETA2,SOVERC,CR2,DP,
1      PLOSS)
DP=DP-PLOSS
XY(I,1)=CR2
XY(I,2)=DP
20 CONTINUE
CALL FITCRV(A,XYW,WRLCRV)
CALL FITCRV(A,XYE,EULCRV)
CALL FITCRV(A,XY,ACTCRV)
10 CONTINUE
* now do the same for the REVERSE FLOW CASE
REVERS=.TRUE.
DO 30 A=1,179

```

```

      ALPH1R=REAL(A-90)*PI/180.
      DO 40 I=1,20
        CR1R=(-0.1*I)/A2
        CALL REVBLD(ISLIP,CR1R,ALPH1R,SOVERC,INCID,BETA1R,BETA2R,
1          SLIP)
        CW2R=SLIP*(RADB1/RADB2-(RADB2/RADB1)*CR1R*TAN(BETA2R))
        XYW(I,1)=CR1R
        XYW(I,2)=CW2R
        CW1R=1.0+CR1R*TAN(BETA1R)
        DP=(RADB1/RADB2)*CW2R-CW1R
        XYE(I,1)=CR1R
        XYE(I,2)=DP
        CALL LOSCAL(REVERS,IEULER,INCID,BETA1R,BETA2R,SOVERC,CR1R,
1          DP,PLOSS)
        DP=DP-PLOSS
        XY(I,1)=CR1R
        XY(I,2)=DP
40      CONTINUE
        CALL FITCRV(A+180,XYW,WRLCRV)
        CALL FITCRV(A+180,XYE,EULCRV)
        CALL FITCRV(A+180,XY,ACTCRV)
30      CONTINUE
        RETURN
      END

*
* the primary blading performance variables are calculated, including:
* S/C, INCIDENCE, BETA1, BETA2, SLIP FACTOR
*
      SUBROUTINE BLDVAL(ISLIP,CR2,ALPHA1,SOVERC,INCID,BETA1,BETA2,SLIP)
      PARAMETER(PI=3.141592654)
      REAL BETAB1,BETAB2,BB,BV,RADB1,RADB2,RADV2(2),RADDMM,STGR,S,C,
1      SOVERC,ALPHA1,BETA1,BETA2,INCID,CR2
      INTEGER NUMBLD,ISLIP

*
      COMMON /GEOMET/ BETAB1,BETAB2,BB,BV,RADB1,RADB2,RADV2,NUMBLD
* calculate blade row geometry data for correlations
      RADDMM=SQRT(RADB1*RADB2)
      STGR=(BETAB1+BETAB2)/2.
      S=2.*PI*RADDMM/NUMBLD
      C=(RADB2-RADB1)/COS(STGR)
      SOVERC=S/C
      BETA1=ATAN(RADB1**2/(RADB2**2*CR2)-TAN(ALPHA1))
      INCID=BETA1-BETAB1
      CALL SLIPFA(ISLIP,INCID,SOVERC,SLIP,BETA2)
      RETURN
      END

*
* the blading variables for REVERSE FLOW CASE are calculated, including:
* INCIDENCE, BETA1, BETA2, SLIP FACTOR
*
      SUBROUTINE REVBLD(ISLIP,CR1R,ALPH1R,SOVERC,INCID,BETA1R,BETA2R,
1      SLIP)
      PARAMETER(PI=3.141592654)
      REAL BETAB1,BETAB2,BB,BV,RADB1,RADB2,RADV2(2),SOVERC,ALPH1R,
1      BETA1R,BETA2R,INCID,CR1R
      INTEGER NUMBLD,ISLIP

*
      COMMON /GEOMET/ BETAB1,BETAB2,BB,BV,RADB1,RADB2,RADV2,NUMBLD
      BETA1R=ATAN(TAN(ALPH1R)-1./CR1R)
      INCID=BETA1R-(-1*BETAB2)
      CALL REVSLP(ISLIP,INCID,SOVERC,SLIP,BETA2R)
      RETURN

```

```

      END
*
* this subroutine calculates the slip factor and thus exit flow angle
* from the impeller based on different models for the slip
*
      SUBROUTINE SLIPFA(ISLIP, INCID, SOVERC, SLIP, BETA2)
      PARAMETER(PI=3.141592654)
      REAL BETAB1, BETAB2, BB, BV, RADB1, RADB2, RADV2(2), SLIP, INCID,
1      SOVERC, BETA2, DELTA
      INTEGER NUMBLD, ISLIP
      COMMON /GEOMET/ BETAB1, BETAB2, BB, BV, RADB1, RADB2, RADV2, NUMBLD
*
      IF (ISLIP.EQ.0) THEN
* the no slip case
        SLIP=1.0
        BETA2=BETAB2
      ELSEIF (ISLIP.EQ.1) THEN
* use the correlation from Kind's (1996) cascade tests for slip
        SLIP=1.0
        DELTA=(PI/12.)*(2.*SOVERC)**0.6-0.13*INCID+(0.126/PI)*INCID**2
        BETA2=BETAB2+DELTA
        IF(BETA2.GT.(89*PI/180)) BETA2=89*PI/180
        IF(BETA2.LT.(-89*PI/180)) BETA2=-89*PI/180
      ENDIF
      RETURN
      END
*
* this subroutine calculates the slip factor and thus exit flow angle
* from the impeller based on different models for the slip for the
* REVERSE FLOW CASE
*
      SUBROUTINE REVSLP(ISLIP, INCID, SOVERC, SLIP, BETA2R)
      PARAMETER(PI=3.141592654)
      REAL BETAB1, BETAB2, BB, BV, RADB1, RADB2, RADV2(2), SLIP, INCID,
1      SOVERC, BETA2R, DELTA
      INTEGER NUMBLD, ISLIP
      COMMON /GEOMET/ BETAB1, BETAB2, BB, BV, RADB1, RADB2, RADV2, NUMBLD
*
      IF (ISLIP.EQ.0) THEN
* the no slip case
        SLIP=1.0
        BETA2R=-1.*BETAB1
      ELSEIF (ISLIP.EQ.1) THEN
* use the correlation from Kind's (1996) cascade tests for slip
        SLIP=1.0
        DELTA=(PI/12.)*(2.*SOVERC)**0.6-0.13*INCID+(0.126/PI)*INCID**2
        BETA2R=-1.*BETAB1+DELTA
        IF(BETA2R.GT.(89*PI/180)) BETA2R=89*PI/180
        IF(BETA2R.LT.(-89*PI/180)) BETA2R=-89*PI/180
      ENDIF
      RETURN
      END
*
* this subroutine calculates the pressure losses across the blade row
*
      SUBROUTINE LOSCAL(RFVERS, IEULER, INCID, BETA1, BETA2, SOVERC, CR, DP,
1      PLOSS)
      PARAMETER(PI=3.141592654)
      REAL COINLT, COLEAK, COEXP, COCOMP, COFRIC, COGAP, COEXT, BETAB1, BETAB2,
1      BB, BV, RADB1, RADB2, RADV2(2), PLOSS, BETA1, BETA2, CR, INCID, DP,
2      KFRIC, KDIFU, KINC, KTOTAL, W1, W2, CDIFF
      INTEGER NUMBLD, IEULER

```

```

LOGICAL REVERS
COMMON /GEOMET/ BETAB1,BETAB2,BB,BV,RADB1,RADB2,RADV2,NUMBLD
COMMON /LOSSES/ COINLT,COLEAK,COEXP,COCOMP,COFRIC,COGAP,COEXT
*
KFRIC=0.005/SOVERC
IF(REVERS) THEN
  W1=ABS(CR/COS(BETA1))
  W2=ABS((RADB2*CR)/(RADB1*COS(BETA2)))
ELSE
  W1=(RADB2*CR)/(RADB1*COS(BETA1))
  W2=CR/COS(BETA2)
ENDIF
IF (IEULER.EQ.1.OR.IEULER.EQ.2) THEN
* the no loss case
  PLOSS=0.0
ELSEIF (IEULER.EQ.3) THEN
* the full losses case using Kind's (1996) correlations
  CDIFF=1.-(W2/W1)**2
  IF((W2/W1).GT.SQRT(3.0)) THEN
    KDIFU=0.0
  ELSE
    KDIFU=((CDIFF+2.)/4.)*1.7*(1/SOVERC)**0.7
  ENDIF
  KINC=0.35*(1.8*INCID/PI)**2*(1.+12.*(SOVERC-0.85)**2)
  KTOTAL=KFRIC+KDIFU+KINC
  PLOSS=COLEAK*DP+0.5*KTOTAL*W1**2
ENDIF
RETURN
END
*****
*          GEOMETRY/FLOW INITIALISATION SUBROUTINES          *
*****
* calculate the fan element geometry based on the geometry input data,
* including sector inlet and outlet areas.
*
SUBROUTINE INIGEO(ISEGM,ISECT,IAXIAL,NUMI,NUMJ)
PARAMETER (PI=3.141592654,IMAX=25,JMAX=100)
REAL BETAB1,BETAB2,BB,BV,RADB1,RADB2,RADV2(2),
1 BBI(IMAX),FANGEQ(JMAX,4),AREAS(IMAX,JMAX,3,2),TANPHI
INTEGER ISEGM,ISECT,IAXIAL,NUMBLD,NUMI,NUMJ,J,I
COMMON /GEOMET/ BETAB1,BETAB2,BB,BV,RADB1,RADB2,RADV2,NUMBLD
COMMON /CALGEO/ FANGEQ,AREAS,TANPHI
PRINT *
PRINT *,'GENERATING FAN ELEMENT GEOMETRY INFO...'
* calculate the widths of each fan segment
DO 30 I=1,NUMI
  IF (ISEGM.EQ.1) THEN
* COSINE distr.--more segments towards the fan inlet
    BBI(I)=BB*(1-COS(I*PI*0.5/NUMI))
  ELSEIF (ISEGM.EQ.2) THEN
* EQUAL distr.--sectors are equal in size
    BBI(I)=BB/NUMI
  ELSEIF (ISEGM.EQ.3) THEN
* SINE distr.--more segments at back of fan
    BBI(I)=BB*SIN(I*PI*0.5/NUMI)
  ENDIF
30 CONTINUE
* calculate the start-end angles of each sector element
DO 5 J=1,NUMJ
  IF (ISECT.EQ.1) THEN
* COSINE distr.--more sectors at start of volute

```

```

      FANGE0(J,1)=2.*PI*(1.-COS((J-1)*PI*0.5/NUMJ))
      FANGE0(J,2)=2.*PI*(1.-COS(J*PI*0.5/NUMJ))
    ELSEIF (ISECT.EQ.2) THEN
* EQUAL distr.--sectors are equal in size
      FANGE0(J,1)=(J-1)*2.*PI/NUMJ
      FANGE0(J,2)=J*2.*PI/NUMJ
    ELSEIF (ISECT.EQ.3) THEN
* SINE distr.--more sectors at exit of volute
      FANGE0(J,1)=2.*PI*SIN((J-1)*PI*0.5/NUMJ)
      FANGE0(J,2)=2.*PI*SIN(J*PI*0.5/NUMJ)
    ENDIF
5    CONTINUE
* calculate the start and end radii of the volute sectors assuming an
* arithmetic volute spiral.
    DO 10 J=1,NUMJ
      FANGE0(J,3)=FANGE0(J,1)/(2.0*PI)*(RADV2(2)-RADV2(1))
1      +RADV2(1)
      FANGE0(J,4)=FANGE0(J,2)/(2.0*PI)*(RADV2(2)-RADV2(1))
1      +RADV2(1)
10   CONTINUE
    IF (IAXIAL.EQ.5) THEN
* if using constant area streamtube, divide impeller into two slices
      NUMI=2
      BBI(1)=(2.*BB-RADB1)/2.
      BBI(2)=RADB1/2.
    ENDIF
* calculate the NON-DIM. area of the inlet and outlet face of each fan
* element
    DO 15 J=1,NUMJ
      DO 20 I=1,NUMI
        AREAS(I,J,1,1)=(FANGE0(J,2)-FANGE0(J,1))*RADB1**2/
1          (8.0*RADB2**2)
        AREAS(I,J,1,2)=(FANGE0(J,2)-FANGE0(J,1))*RADB1*BBI(I)/
1          (4.*RADB2**2)
        AREAS(I,J,2,1)=AREAS(I,J,1,2)
        AREAS(I,J,2,2)=(FANGE0(J,2)-FANGE0(J,1))*BBI(I)/(4.*RADB2)
20   CONTINUE
        AREAS(1,J,3,1)=(FANGE0(J,3)-RADB2)*BV/(4.*RADB2**2)
        AREAS(1,J,3,2)=(FANGE0(J,4)-RADB2)*BV/(4.*RADB2**2)
15   CONTINUE
* calculate the tangent of the volute casing spiral angle PHI assuming
* an arithmetic spiral
      TANPHI=(RADV2(2)-RADV2(1))/(PI*(RADV2(1)+RADV2(2)))
      RETURN
    END
*
* using the sector flowrate guess, set the initial element flowrates
* and velocity components. Also, find the flowrate weighted average
* blading exit static pressure and whirl velocity
*
    SUBROUTINE DIVFLO(IAXIAL,COINLT,PCRV,WCRV,Q,J,WHIRL,RADIAL,PB2AVG)
    PARAMETER (PI=3.141592654,IMAX=25,JMAX=100,MAXORD=5)
    DIMENSION PCRV(359,MAXORD+1),WCRV(359,MAXORD+1)
    REAL TOLER,BETAB1,BETAB2,BB,BV,RADB1,RADB2,RADV2(2),
1    FANGE0(JMAX,4),AREAS(IMAX,JMAX,3,2),TANPHI,Q,ALPHA,
2    INLET1(IMAX,JMAX,2),INLET2(IMAX,JMAX,3),BLADE1(IMAX,JMAX,3),
3    BLADE2(IMAX,JMAX,5),VOLUT1(JMAX,3),VOLUT2(JMAX,3),PTIN,
4    ASUM,X1,X2,PCRV,WCRV,EQAREA,WHIRL,RADIAL,PB2AVG,QSUM
    INTEGER NUMBLD,NUMI,NUMJ,NUMM,IAXIAL,I,J,K,MIDLI
    COMMON /GENRL/ NUMI,NUMJ,TOLER,NUMM
    COMMON /FLOW/ INLET1,INLET2,BLADE1,BLADE2,VOLUT1,VOLUT2
    COMMON /GEOMET/ BETAB1,BETAB2,BB,BV,RADB1,RADB2,RADV2,NUMBLD

```

```

COMMON /CALGEO/ FANGEQ,AREAS,TANPHI
*
DO 25 I=1,NUMI
  DO 30 K=1,NUMJ
* initialise ALPHA1 to 0 deg. (radial inflow) since not considering
* reverse flow
    BLADE1(I,K,3)=0.0
30    CONTINUE
25    CONTINUE
    ALPHA=90
    PTIN=-0.5*COINLT*(Q/AREAS(1,J,1,1))*2
* calculate the impeller inlet total pressure
    ASUM=RADB1*(FANGEQ(J,2)-FANGEQ(J,1))*BB/(4.*RADB2**2.)
    X2=0.
    EQAREA=1.0-RADB1/(2.*BB)
    WHIRL=0.0
    RADIAL=0.0
    PB2AVG=0.0
    QSUM=0.0
    DO 10 I=1,NUMI
* divide the assumed starting flowrate through each fan slice
    X1=X2
    X2=X2+(AREAS(I,J,1,2)/ASUM)
    IF (IAXIAL.EQ.1) THEN
      INLET2(I,J,2)=Q*(X2-X1)
    ELSEIF (IAXIAL.EQ.2) THEN
      INLET2(I,J,2)=Q*(X2**2.-X1**2.)
    ELSEIF (IAXIAL.EQ.3) THEN
      INLET2(I,J,2)=Q*(X2**3.-X1**3.)
    ELSEIF (IAXIAL.EQ.4) THEN
      IF (X1.LT.0.5) THEN
        INLET2(I,J,2)=0.
        MIDLI=I
      ELSEIF (X1.GE.0.5) THEN
        INLET2(I,J,2)=Q*(2.*X2**2-2.*X1**2-X2+X1)
        QSUM=QSUM+INLET2(I,J,2)
      ENDIF
    ELSEIF (IAXIAL.EQ.5) THEN
      IF (I.EQ.1) INLET2(I,J,2)=0.0
      IF (I.EQ.2) INLET2(I,J,2)=Q
    ENDIF
    BLADE1(I,J,2)=INLET2(I,J,2)
    BLADE2(I,J,2)=BLADE1(I,J,2)
    BLADE2(I,J,4)=BLADE2(I,J,2)/AREAS(I,J,2,2)
    IF (BLADE2(I,J,4).LT.0.0) ALPHA=BLADE1(I,J,3)+270.
    IF (BLADE2(I,J,4).GE.0.0) ALPHA=BLADE1(I,J,3)+90.
    BLADE2(I,J,3)=CURVE(WCRV,BLADE2(I,J,4),ALPHA)
    BLADE2(I,J,5)=ATAN(BLADE2(I,J,4)/BLADE2(I,J,3))
    BLADE2(I,J,1)=PTIN+CURVE(PCRV,BLADE2(I,J,4),ALPHA)-
1      0.5*(BLADE2(I,J,3)**2+BLADE2(I,J,4)**2)
* keep sum of whirl & radial velocities, and static pressure so flow
* rate weighted average can be found later
    WHIRL=WHIRL+BLADE2(I,J,3)*BLADE2(I,J,2)
    RADIAL=RADIAL+BLADE2(I,J,4)*BLADE2(I,J,2)
    PB2AVG=PB2AVG+BLADE2(I,J,1)*BLADE2(I,J,2)
10    CONTINUE
* add any missing flow at interface of zero/non-zero flow
    IF (IAXIAL.EQ.4) THEN
      INLET2(MIDLI,J,2)=Q-QSUM
      BLADE1(MIDLI,J,2)=INLET2(MIDLI,J,2)
      BLADE2(MIDLI,J,2)=BLADE1(MIDLI,J,2)
      BLADE2(MIDLI,J,4)=BLADE2(MIDLI,J,2)/AREAS(MIDLI,J,2,2)

```

```

      IF (BLADE2(MIDLI,J,4).LT.0.0) ALPHA=BLADE1(MIDLI,J,3)+270.
      IF (BLADE2(MIDLI,J,4).GE.0.0) ALPHA=BLADE1(MIDLI,J,3)+90.
      BLADE2(MIDLI,J,3)=CURVE(WCRV,BLADE2(MIDLI,J,4),ALPHA)
      BLADE2(MIDLI,J,1)=PTIN+CURVE(PCRV,BLADE2(MIDLI,J,4),ALPHA)
1      -0.5*(BLADE2(MIDLI,J,3)**2+BLADE2(MIDLI,J,4)**2)
      WHIRL=WHIRL+BLADE2(MIDLI,J,3)*BLADE2(MIDLI,J,2)
      RADIAL=RADIAL+BLADE2(MIDLI,J,4)*BLADE2(MIDLI,J,2)
      PB2AVG=PB2AVG+BLADE2(MIDLI,J,1)*BLADE2(MIDLI,J,2)
      ENDIF
* finish calculation of flow rate weighted averages
      WHIRL=WHIRL/Q
      RADIAL=RADIAL/Q
      PB2AVG=PB2AVG/Q
      RETURN
      END
*****
*                               DATA INPUT/OUTPUT SUBROUTINES                               *
*****
*
* get the fan geometry, and simulation running conditions
*
      SUBROUTINE GETDAT
      PARAMETER (IMAX=25,JMAX=100,QMAX=1.0,PI=3.141592654)
      REAL BETAB1,BETAB2,BB,BV,RADB1,RADB2,RADV2(2),TOLER,
1      COINLT,COLEAK,COEXP,COCOMP,COFRIC,COGAP,COEXT
      INTEGER NUMBLD,NUMI,NUMJ,NUMM,PICK,
1      ISEGM,ISECT,IAXIAL,IEULER,ISLIP,IMODEL,IOUT
*
      COMMON /GEOMET/ BETAB1,BETAB2,BB,BV,RADB1,RADB2,RADV2,NUMBLD
      COMMON /GENRL/ NUMI,NUMJ,TOLER,NUMM
      COMMON /LOSSES/ COINLT,COLEAK,COEXP,COCOMP,COFRIC,COGAP,COEXT
      COMMON /SWITCH/ ISEGM,ISECT,IAXIAL,IEULER,ISLIP,IMODEL,IOUT
      PRINT *,'INPUT DATA CHECK...'
      PRINT *
      OPEN (UNIT=1,FILE='INGEO.DAT')
      PRINT *,'*** Fan geometry data ***'
      READ (1,*)BETAB1,BETAB2,BB,BV,RADB1,RADB2,NUMBLD,RADV2
10     PRINT *
      PRINT *,'BETAB1 BETAB2 BB BV RADB1 RADB2 NUMBLD
1     RADV2'
      WRITE (*,5)BETAB1,BETAB2,BB,BV,RADB1,RADB2,NUMBLD,RADV2
5     FORMAT(F6.2,2X,F6.2,4(2X,F6.4),3X,I3,5X,2(F6.4,2X))
      PRINT *
      PRINT *,'Enter the item no. to be changed (0 for none)'
      READ (*,*)PICK
      IF (PICK.EQ.1) THEN
        PRINT *,'Enter the new blade inlet angle...'
        READ (*,*)BETAB1
      ELSEIF (PICK.EQ.2) THEN
        PRINT *,'Enter the new blade outlet angle...'
        READ (*,*)BETAB2
      ELSEIF (PICK.EQ.3) THEN
        PRINT *,'Enter the new impeller width...'
        READ (*,*)BB
      ELSEIF (PICK.EQ.4) THEN
        PRINT *,'Enter the new volute width...'
        READ (*,*)BV
      ELSEIF (PICK.EQ.5) THEN
        PRINT *,'Enter the new impeller inlet radius...'
        READ (*,*)RADB1
      ELSEIF (PICK.EQ.6) THEN
        PRINT *,'Enter the new impeller outlet radius...'

```



```

      READ (*,*)RADB2
    ELSEIF (PICK.EQ.7) THEN
      PRINT *, 'Enter the new number of blades...'
      READ (*,*)NUMBLD
    ELSEIF (PICK.EQ.8) THEN
      PRINT *, 'Enter the first volute wall radius (at tongue)...'
      READ (*,*)RADV2(1)
      PRINT *, 'Enter the final volute wall radius (at exit)...'
      READ (*,*)RADV2(2)
    ENDIF
    IF (PICK.NE.0) THEN
      GOTO 10
    ENDIF
    CLOSE (1)
    OPEN (UNIT=2, FILE='INRUN.DAT')
    READ (2,*)NUMI, NUMJ, TOLER, NUMM,
1      ISEGM, ISECT, IAXIAL, IEULER, ISLIP, IMODEL, IOUT
    PRINT *
    PRINT *, '*** Fan operation data ***'
25    PRINT *
    PRINT *, '  NUMI    NUMJ    TOLER    NUMM'
    WRITE (*,20)NUMI, NUMJ, TOLER, NUMM
20    FORMAT (2X, 2(I3,4X), F6.4, 3X, I3)
    PRINT *
    PRINT *, 'Enter the item no. to be changed (0 for none)'
    READ (*,*)PICK
    IF (PICK.EQ.1) THEN
      PRINT *, 'Enter the number of segments...'
      READ (*,*)NUMI
    ELSEIF (PICK.EQ.2) THEN
      PRINT *, 'Enter the number of sectors...'
      READ (*,*)NUMJ
    ELSEIF (PICK.EQ.3) THEN
      PRINT *, 'Enter the new iteration tolerance...'
      READ (*,*)TOLER
    ELSEIF (PICK.EQ.4) THEN
      PRINT *, 'Enter the new number of data points to find...'
      READ (*,*)NUMM
    ENDIF
    IF (PICK.NE.0) THEN
      GOTO 25
    ENDIF
45    PRINT *
    PRINT *, 'ISEGM  ISECT  IAXIAL  IEULER  ISLIP  IMODEL  IOUT'
    WRITE (*,40)ISEGM, ISECT, IAXIAL, IEULER, ISLIP, IMODEL, IOUT
40    FORMAT (2X, 7(I1,7X))
    PRINT *
    PRINT *, 'Enter the item no. to be changed (0 for none)'
    READ (*,*)PICK
    IF (PICK.EQ.1) THEN
      WRITE (*,50)
      READ (*,*)ISEGM
    ELSEIF (PICK.EQ.2) THEN
      WRITE (*,50)
      READ (*,*)ISECT
    ELSEIF (PICK.EQ.3) THEN
      WRITE (*,50)
      READ (*,*)IAXIAL
    ELSEIF (PICK.EQ.4) THEN
      WRITE (*,50)
      READ (*,*)IEULER
    ELSEIF (PICK.EQ.5) THEN

```

```

WRITE (*,50)
READ (*,*)ISLIP
ELSEIF (PICK.EQ.6) THEN
WRITE (*,50)
READ (*,*)IMODEL
ELSEIF (PICK.EQ.7) THEN
WRITE (*,50)
READ (*,*)IOUT
ENDIF
50  FORMAT ('What is the new switch setting? ',I1)
    IF (PICK.NE.0) THEN
        GOTO 45
    ENDIF
    CLOSE(2)
    OPEN (UNIT=3,FILE='INLOSS.DAT')
    READ (3,*)COINLT,COLEAK,COEXP,COCOMP,COFRIC,COGAP,COEXT
35  PRINT *
    PRINT *, ' COINLT COLEAK COEXP COCOMP COFRIC COGAP COEXT'
    WRITE (*,30)COINLT,COLEAK,COEXP,COCOMP,COFRIC,COGAP,COEXT
30  FORMAT (7(F7.4,1X))
    PRINT *
    PRINT *, 'Enter the item no. to be changed (0 for none)'
    READ (*,*)PICK
    IF (PICK.EQ.1) THEN
        PRINT *, 'Enter the new fan inlet loss...'
        READ (*,*)COINLT
    ELSEIF (PICK.EQ.2) THEN
        PRINT *, 'Enter the new impeller flow leakage factor...'
        READ (*,*)COLEAK
    ELSEIF (PICK.EQ.3) THEN
        PRINT *, 'Enter the new % of blade flow rapidly expanded...'
        READ (*,*)COEXP
    ELSEIF (PICK.EQ.4) THEN
        PRINT *, 'Enter the new compatability model modifier...'
        READ (*,*)COCOMP
    ELSEIF (PICK.EQ.5) THEN
        PRINT *, 'Enter the new volute wall drag coefficient...'
        READ (*,*)COFRIC
    ELSEIF (PICK.EQ.6) THEN
        PRINT *, 'Enter the new gap dynamic pressure loss...'
        READ (*,*)COGAP
    ELSEIF (PICK.EQ.7) THEN
        PRINT *, 'Enter the new non-flow related efficiency...'
        READ (*,*)COEXT
    ENDIF
    IF (PICK.NE.0) THEN
        GOTO 35
    ENDIF
    CLOSE(3)
* check to see if any of the data is out of the program's range
    IF (NUMI.GT.IMAX) THEN
        PRINT *, 'NUMI exceeds this program's limit of 25.'
        GOTO 25
    ELSEIF (NUMJ.GT.JMAX) THEN
        PRINT *, 'NUMJ exceeds this program's limit of 50.'
        GOTO 25
    ELSEIF (ISEGM.LT.1.OR.ISEGM.GT.3) THEN
        PRINT*, 'ISEGM has a value I don't understand!'
        GOTO 45
    ELSEIF (ISECT.LT.1.OR.ISECT.GT.3) THEN
        PRINT*, 'ISECT has a value I don't understand!'
        GOTO 45

```

```

ELSEIF(IAXIAL.LT.1.OR.IAXIAL.GT.5) THEN
  PRINT*, 'IAXIAL has a value I don't understand!'
  GOTO 45
ELSEIF(IEULER.LT.1.OR.IEULER.GT.3) THEN
  PRINT*, 'IEULER has a value I don't understand!'
  GOTO 45
ELSEIF(ISLIP.LT.1.OR.ISLIP.GT.1) THEN
  PRINT*, 'ISLIP has a value I don't understand!'
  GOTO 45
ELSEIF(IMODEL.LT.1.OR.IMODEL.GT.3) THEN
  PRINT*, 'IMODEL has a value I don't understand!'
  GOTO 45
ELSEIF(IOUT.NE.1.AND.IOUT.NE.2) THEN
  PRINT*, 'IOUT has a value I don't understand!'
  GOTO 45
ENDIF
RETURN
END

*
* print the simulation main info as a header in the output file
*
  SUBROUTINE HEADER(OUTFIL)
    REAL BETAB1,BETAB2,BB,BV,RADB1,RADB2,RADV2(2),TOLER,
1    COINLT,COLEAK,COEXP,COCOMP,COFRIC,COGAP,COEXT
    INTEGER NUMBLD,NUMI,NUMJ,NUMM,
1    ISEGM,ISECT,IAXIAL,IEULER,ISLIP,IMODEL,IOUT
    CHARACTER *8 OUTFIL
*
    COMMON /GEOMET/ BETAB1,BETAB2,BB,BV,RADB1,RADB2,RADV2,NUMBLD
    COMMON /GENRL/ NUMI,NUMJ,TOLER,NUMM
    COMMON /LOSSES/ COINLT,COLEAK,COEXP,COCOMP,COFRIC,COGAP,COEXT
    COMMON /SWITCH/ ISEGM,ISECT,IAXIAL,IEULER,ISLIP,IMODEL,IOUT
    PRINT *
    PRINT *, 'WRITING HEADER INFO...'
    WRITE (4,*) '
                                     *** FANSIM ***'
    WRITE (4,*) 'Centrifugal Fan Simulation'
    WRITE (4,*)
    WRITE (4,5) 'run name: ',OUTFIL
5    FORMAT(59X,2A)
    WRITE (4,*)
    WRITE (4,*) 'Simulation Input Data...'
    WRITE (4,*)
    WRITE (4,*) 'BETAB1  BETAB2    BB      BV      RADB1  RADB2  NUMBLD
1    RADV2'
    WRITE (4,10) BETAB1,BETAB2,BB,BV,RADB1,RADB2,NUMBLD,RADV2
10    FORMAT(F6.2,2X,F6.2,4(2X,F6.4),3X,I3,5X,2(F6.4,2X))
    WRITE (4,*)
    WRITE (4,*) '  NUMI  NUMJ  TOLER  NUMM'
    WRITE (4,15) NUMI,NUMJ,TOLER,NUMM
15    FORMAT (2X,2(I3,4X),F6.4,3X,I3)
    WRITE (4,*)
    WRITE (4,*) 'ISEGM  ISECT  IAXIAL  IEULER  ISLIP  IMODEL  IOUT'
    WRITE (4,20) ISEGM,ISECT,IAXIAL,IEULER,ISLIP,IMODEL,IOUT
20    FORMAT (2X,7(I1,7X))
    WRITE (4,*)
    WRITE (4,*) 'COINLT  COLEAK  COEXP  COCOMP  COFRIC  COGAP  COEX
1T'
    WRITE (4,25) COINLT,COLEAK,COEXP,COCOMP,COFRIC,COGAP,COEXT
25    FORMAT (7(F7.4,1X))
    WRITE (4,*)
    WRITE (4,*) '*****'

```

```

WRITE(4,*)
WRITE(4,*)'The overall fan performance parameters are:'
WRITE(4,*)'(values are non-dimensional, pressures are gauge)'
WRITE(4,*)
WRITE(4,30)
30  FORMAT(' Fan      Exit Static  Exit Dyna.  Exit Total  Overall
1    Qgap/  Gap/Blading')
WRITE(4,35)
35  FORMAT('Flowrate  Pressure      Pressure  Pressure  Eff''y
1    Qfan    Match')
WRITE(4,*)
IF(IOUT.EQ.2) THEN
WRITE(10,5)'run name: ',OUTFIL
WRITE(10,*)'Impeller Flow Data by Sector/Slice...'
WRITE(10,*)
WRITE(11,5)'run name: ',OUTFIL
WRITE(11,*)'Volute Flow Data by Sector...'
WRITE(11,*)
ENDIF
RETURN
END

*
* output the results, including: overall fan performance & sector flow
* variables
*
SUBROUTINE OUTDAT(IOUT)
PARAMETER (IMAX=25,JMAX=100)
REAL FLOW,STATIC,TOTAL,DYNAM,EFFIC,TOLER,BLADE(IMAX,JMAX,7),
1  VOLUTE(JMAX,8),T1
INTEGER NUMI,NUMJ,NUMM,IOUT,I,J
LOGICAL GAP,SECT(JMAX)
CHARACTER *3 MATCH,MATCH2

*
COMMON /GENRL/ NUMI,NUMJ,TOLER,NUMM
COMMON /SOLUTN/ FLOW,STATIC,TOTAL,DYNAM,EFFIC,BLADE,VOLUTE,
1  GAP,SECT
IF(GAP) THEN
MATCH='yes'
ELSE
MATCH=' no'
ENDIF
DO 22 J=1,NUMJ
IF(SECT(J)) THEN
MATCH2='yes'
ELSE
MATCH2=' no'
GOTO 23
ENDIF
22 CONTINUE
23 T1=VOLUTE(1,1)/FLOW
WRITE(4,20)FLOW,STATIC,DYNAM,TOTAL,EFFIC,T1,MATCH,MATCH2
20  FORMAT(F8.5,2X,5(F9.5,3X),A3,1X,A3)
IF(IOUT.EQ.2) THEN
WRITE(10,17)'Fan Flowrate=',FLOW
17  FORMAT(A,F8.5)
WRITE(10,*)
WRITE(10,15)
15  FORMAT(' J  I  Qb2      Crb2      Cwb2      Pb2      Ptb2      P
1v-  Cwb2'' conv.'')
DO 40 J=1,NUMJ
DO 50 I=1,NUMI
IF(SECT(J)) THEN

```

```

        MATCH='yes'
      ELSE
        MATCH=' no'
      ENDIF
      WRITE(10,16)J,I,(BLADE(I,J,N),N=1,7),MATCH
      FORMAT(I2,I3,7F9.5,3X,A)
16    CONTINUE
50    CONTINUE
40    WRITE(10,*)
      WRITE(11,17)'Fan Flowrate=',FLOW
      WRITE(11,*)
      WRITE(11,55)
55    FORMAT(' J      Qv1      Cwv1      Pv1      Ptv1      Qv2      Cwv2
1      Pv2      Ptv2')
      DO 60 J=1,NUMJ
60    WRITE(11,56)J,(VOLUTE(J,N),N=1,8)
56    FORMAT(I2,8F9.5)
      WRITE(11,*)
      ENDIF
      RETURN
      END
*****
*          MISCELLANEOUS SUBROUTINES          *
*****
* fits a "least-squares" quadratic to 20 points from the blading
* calculation
*
      SUBROUTINE FITCRV(ALPHA,XY,CURVE)
      PARAMETER (MAXORD=5)
      DIMENSION XY(20,2),CURVE(359,MAXORD+1)
      REAL XY,CURVE,A(MAXORD+1,MAXORD+1),B(MAXORD+1),X(MAXORD),
1      CRVFIT(MAXORD+1)
      INTEGER N,I,J,ALPHA
      N=3
      X(1)=20.
      B(1)=0.
      DO 30 I=2,5
        X(I)=0.0
        B(I)=0.0
30    CONTINUE
      DO 5 I=1,20
        DO 10 J=2,5
          X(J)=X(J)+(XY(I,1))**(J-1)
10    CONTINUE
        DO 15 J=1,N
          B(J)=B(J)+XY(I,2)*XY(I,1)**(J-1)
15    CONTINUE
5    CONTINUE
      DO 20 I=1,N
        DO 25 J=1,N
          A(I,J)=X(I-J+3)
25    CONTINUE
20    CONTINUE
      CALL LINEAR(N,A,B,CRVFIT)
      DO 40 I=1,N
        CURVE(ALPHA,I)=CRVFIT(I)
40    CONTINUE
      RETURN
      END
*
* solves a NxN linear equation using Gaussian elimination
*

```

```

SUBROUTINE LINEAR(N,A,B,X)
PARAMETER (MAXORD=5)
DIMENSION A(MAXORD+1,MAXORD+1),B(MAXORD+1),X(MAXORD+1)
REAL A,B,X,T,R,S
INTEGER N,I,J,K,IP
DO 6 I=1,(N-1)
  IP=I+1
  R=A(I,I)
  DO 7 J=IP,N
    T=A(J,I)/R
    B(J)=B(J)-T*B(I)
    DO 8 K=I,N
      A(J,K)=A(J,K)-T*A(I,K)
8    CONTINUE
7    CONTINUE
6  CONTINUE
DO 9 I=N,1,-1
  IP=I+1
  S=0.0
  DO 10 J=IP,N
    S=S+X(J)*A(I,J)
10  CONTINUE
  X(I)=(B(I)-S)/A(I,I)
9  CONTINUE
RETURN
END

*
* this function uses 3 predetermined coefficients for a quadratic
* equation to calculate one of three things:
*
*   curvefit coeff's      X      Y
*   ACTCRV()              Crb2    actual rotor total pressure rise
*   EULCRV()              "       rotor Euler total pressure rise
*   WRLCRV()              "       rotor exit whirl velocity
*
REAL FUNCTION CURVE(CRVFIT,X,ALPHA)
PARAMETER (MAXORD=5)
DIMENSION CRVFIT(359,MAXORD+1)
* The 3 parameters req'd by the quadratic curvefit are stored in an
* array with a set of 3 parameters for each possible absolute inlet flow
* angle. The array CRVFIT has curvefit parameters for inlet flow angles
* -89 to 89 degrees stored in locations 1 to 179, and for inlet flow
* angles 91 to 269 (ie. reverse flow) in locations 181 to 359.
A=CRVFIT(INT(ALPHA),1)
B=CRVFIT(INT(ALPHA),2)
C=CRVFIT(INT(ALPHA),3)
CURVE=A*X**2+B*X+C
RETURN
END

```

APPENDIX C - SAMPLE OUTPUT

OUT.OUT

*** FANSIM *** Centrifugal Fan Simulation

run name: out

Simulation Input Data...

BETAB1	BETAB2	BB	BV	RADB1	RADB2	NUMBLD	RADV2
0.00	-65.00	0.1520	0.1920	0.1125	0.1395	50	0.1680 0.3336
NUMI	NUMJ	TOLER	NUMM				
5	5	0.0100	5				
ISEGM	ISECT	IAXIAL	IEULER	ISLIP	IMODEL	IOUT	
2	2	5	3	2	1	2	
COINLT	COLEAK	COEXP	COCOMP	COFRIC	COGAP	COEXT	
0.1000	0.0000	0.0000	1.0000	0.0030	0.0000	0.9800	

The overall fan performance parameters are:
(values are non-dimensional, pressures are gauge)

Fan Flowrate	Exit Static Pressure	Exit Dyna. Pressure	Exit Total Pressure	Overall Eff'y	Qgap/Qfan	Gap/Blading Match	
0.20000	0.75569	0.12120	0.87688	0.58891	0.45131	yes	no
0.40000	0.58838	0.48479	1.07317	0.55646	0.28862	yes	no
0.60000	-0.16703	1.09078	0.92375	0.39424	0.22655	yes	no
0.80000	-1.52233	1.93916	0.41683	0.15254	0.19013	yes	no
1.00000	-3.45497	3.02994	-0.42503	-0.13724	0.16680	yes	no

OUT.BLD

run name: out

Impeller Flow Data by Sector/Slice...

Fan Flowrate= 0.20000

J	I	Qb2	Crb2	Cwb2	Pb2	Ptb2	Pv~	Cwb2'	conv
1	1	0.00000	0.00000	1.00389	-0.06533	0.43857	-0.13595	1.07194	yes
1	2	0.04000	0.31576	1.45922	-0.13595	0.97856	-0.13595	1.45922	yes
2	1	0.00000	0.00000	1.00389	-0.06533	0.43857	0.32712	0.47212	no
2	2	0.04000	0.31576	1.45922	-0.13595	0.97856	0.32712	1.11561	no
3	1	0.00000	0.00000	1.00389	-0.06533	0.43857	0.40211	0.27003	no
3	2	0.04000	0.31576	1.45922	-0.13595	0.97856	0.40211	1.04944	no
4	1	0.00000	0.00000	1.00389	-0.06533	0.43857	0.40487	0.25960	no

4	2	0.04000	0.31576	1.45922	-0.13595	0.97856	0.40487	1.04692	no
5	1	0.00000	0.00000	1.00389	-0.06533	0.43857	0.38077	0.34000	no
5	2	0.04000	0.31576	1.45922	-0.13595	0.97856	0.38077	1.06869	no

Fan Flowrate= 0.40000

J	I	Qb2	Crb2	Cwb2	Pb2	Ptb2	Pv~	Cwb2'	conv
1	1	0.00000	0.00000	1.00389	-0.08832	0.41557	-0.58585	1.41522	yes
1	2	0.08000	0.63153	1.89000	-0.58585	1.39963	-0.58585	1.89000	yes
2	1	0.00000	0.00000	1.00389	-0.08832	0.41557	-0.13053	1.04509	no
2	2	0.08000	0.63153	1.89000	-0.58585	1.39963	-0.13053	1.65920	no
3	1	0.00000	0.00000	1.00389	-0.08832	0.41557	-0.14590	1.05969	no
3	2	0.08000	0.63153	1.89000	-0.58585	1.39963	-0.14590	1.66751	no
4	1	0.00000	0.00000	1.00389	-0.08832	0.41557	-0.26198	1.16409	no
4	2	0.08000	0.63153	1.89000	-0.58585	1.39963	-0.26198	1.72900	no
5	1	0.00000	0.00000	1.00389	-0.08832	0.41557	-0.42275	1.29485	no
5	2	0.08000	0.63153	1.89000	-0.58585	1.39963	-0.42275	1.81071	no

Fan Flowrate= 0.60000

J	I	Qb2	Crb2	Cwb2	Pb2	Ptb2	Pv~	Cwb2'	conv
1	1	0.00000	0.00000	1.00389	-0.12665	0.37725	-1.37563	1.87237	yes
1	2	0.12000	0.94729	2.29626	-1.37563	1.70944	-1.37563	2.29626	yes
2	1	0.00000	0.00000	1.00389	-0.12665	0.37725	-1.11482	1.72746	no
2	2	0.12000	0.94729	2.29626	-1.37563	1.70944	-1.11482	2.19705	no
3	1	0.00000	0.00000	1.00389	-0.12665	0.37725	-1.32852	1.84704	no
3	2	0.12000	0.94729	2.29626	-1.37563	1.70944	-1.32852	2.27866	no
4	1	0.00000	0.00000	1.00389	-0.12665	0.37725	-1.66394	2.02049	no
4	2	0.12000	0.94729	2.29626	-1.37563	1.70944	-1.66394	2.40116	no
5	1	0.00000	0.00000	1.00389	-0.12665	0.37725	-2.06393	2.20961	no
5	2	0.12000	0.94729	2.29626	-1.37563	1.70944	-2.06393	2.53952	no

Fan Flowrate= 0.80000

J	I	Qb2	Crb2	Cwb2	Pb2	Ptb2	Pv~	Cwb2'	conv
1	1	0.00000	0.00000	1.00389	-0.18031	0.32359	-2.47539	2.36600	yes
1	2	0.16000	1.26305	2.67797	-2.47539	1.90801	-2.47539	2.67797	yes
2	1	0.00000	0.00000	1.00389	-0.18031	0.32359	-2.58403	2.41148	no
2	2	0.16000	1.26305	2.67797	-2.47539	1.90801	-2.58403	2.71095	no
3	1	0.00000	0.00000	1.00389	-0.18031	0.32359	-3.11048	2.62071	no
3	2	0.16000	1.26305	2.67797	-2.47539	1.90801	-3.11048	2.86541	no
4	1	0.00000	0.00000	1.00389	-0.18031	0.32359	-3.77118	2.86174	no
4	2	0.16000	1.26305	2.67797	-2.47539	1.90801	-3.77118	3.04820	no
5	1	0.00000	0.00000	1.00389	-0.18031	0.32359	-4.51691	3.11143	no
5	2	0.16000	1.26305	2.67797	-2.47539	1.90801	-4.51691	3.24215	no

Fan Flowrate= 1.00000

J	I	Qb2	Crb2	Cwb2	Pb2	Ptb2	Pv~	Cwb2'	conv
1	1	0.00000	0.00000	1.00389	-0.24930	0.25460	-3.85704	2.86763	yes
1	2	0.20000	1.57882	3.03514	-3.85704	1.99533	-3.85704	3.03514	yes
2	1	0.00000	0.00000	1.00389	-0.24930	0.25460	-4.49903	3.08338	no
2	2	0.20000	1.57882	3.03514	-3.85704	1.99533	-4.49903	3.19728	no
3	1	0.00000	0.00000	1.00389	-0.24930	0.25460	-5.46084	3.38096	no
3	2	0.20000	1.57882	3.03514	-3.85704	1.99533	-5.46084	3.42587	no
4	1	0.00000	0.00000	1.00389	-0.24930	0.25460	-6.55912	3.69154	no
4	2	0.20000	1.57882	3.03514	-3.85704	1.99533	-6.55912	3.66952	no
5	1	0.00000	0.00000	1.00389	-0.24930	0.25460	-7.76190	4.00412	no
5	2	0.20000	1.57882	3.03514	-3.85704	1.99533	-7.76190	3.91901	no

OUT.VOL

run name: out

Volute Flow Data by Sector...

Fan Flowrate= 0.20000

J	Qv1	Cwv1	Pv1	Ptv1	Qv2	Cwv2	Pv2	Ptv2
1	0.09026	1.28401	0.04302	0.87646	0.13026	0.85704	0.50925	0.88057
2	0.13026	0.85704	0.50925	0.88057	0.17026	0.72860	0.61531	0.88368
3	0.17026	0.72860	0.61531	0.88368	0.21026	0.66670	0.65802	0.88272
4	0.21026	0.66670	0.65802	0.88272	0.25026	0.63028	0.67936	0.88018
5	0.25026	0.63028	0.67936	0.88018	0.29026	0.60628	0.69107	0.87688

Fan Flowrate= 0.40000

J	Qv1	Cwv1	Pv1	Ptv1	Qv2	Cwv2	Pv2	Ptv2
1	0.11545	1.64229	-0.29307	1.07038	0.19545	1.28593	0.27950	1.11543
2	0.19545	1.28593	0.27950	1.11543	0.27545	1.17873	0.41209	1.11447
3	0.27545	1.17873	0.41209	1.11447	0.35545	1.12706	0.46146	1.10361
4	0.35545	1.12706	0.46146	1.10361	0.43545	1.09666	0.48124	1.08921
5	0.43545	1.09666	0.48124	1.08921	0.51545	1.07663	0.48720	1.07317

Fan Flowrate= 0.60000

J	Qv1	Cwv1	Pv1	Ptv1	Qv2	Cwv2	Pv2	Ptv2
1	0.13593	1.93365	-0.96975	0.92040	0.25593	1.68386	-0.41176	1.02160
2	0.25593	1.68386	-0.41176	1.02160	0.37593	1.60872	-0.28916	1.01912
3	0.37593	1.60872	-0.28916	1.01912	0.49593	1.57251	-0.25567	0.99438
4	0.49593	1.57251	-0.25567	0.99438	0.61593	1.55120	-0.25528	0.96111
5	0.61593	1.55120	-0.25528	0.96111	0.73593	1.53716	-0.27072	0.92375

Fan Flowrate= 0.80000

J	Qv1	Cwv1	Pv1	Ptv1	Qv2	Cwv2	Pv2	Ptv2
1	0.15210	2.16367	-1.96720	0.39939	0.31210	2.05343	-1.53850	0.59307
2	0.31210	2.05343	-1.53850	0.59307	0.47210	2.02026	-1.47132	0.59194
3	0.47210	2.02026	-1.47132	0.59194	0.63210	2.00428	-1.48338	0.54737
4	0.63210	2.00428	-1.48338	0.54737	0.79210	1.99487	-1.52567	0.48606
5	0.79210	1.99487	-1.52567	0.48606	0.95210	1.98868	-1.58243	0.41683

Fan Flowrate= 1.00000

J	Qv1	Cwv1	Pv1	Ptv1	Qv2	Cwv2	Pv2	Ptv2
1	0.16680	2.37279	-3.24587	-0.39973	0.36680	2.41332	-3.05489	-1.1067
2	0.36680	2.41332	-3.05489	-0.11067	0.56680	2.42551	-3.09812	-1.12408
3	0.56680	2.42551	-3.09812	-0.12408	0.76680	2.43139	-3.19237	-1.20390
4	0.76680	2.43139	-3.19237	-0.20390	0.96680	2.43485	-3.30570	-1.30872
5	0.96680	2.43485	-3.30570	-0.30872	1.16680	2.43713	-3.42762	-1.42503

END

1 | 9 - 0 | 6 - 9 7

FIN