



uOttawa

L'Université canadienne
Canada's university

**FACULTÉ DES ÉTUDES SUPÉRIEURES
ET POSTDOCTORALES**



**FACULTY OF GRADUATE AND
POSTDOCTORAL STUDIES**

Derek Lastiwka

AUTEUR DE LA THÈSE / AUTHOR OF THESIS

M.A.Sc. (Mechanical Engineering)

GRADE / DEGREE

Department of Mechanical Engineering

FACULTÉ, ÉCOLE, DÉPARTEMENT / FACULTY, SCHOOL, DEPARTMENT

Influence of Rotor Blade Scaling on the Numerical Simulation of a High Pressure Gas Turbine

TITRE DE LA THÈSE / TITLE OF THESIS

S. Tavoularis

DIRECTEUR (DIRECTRICE) DE LA THÈSE / THESIS SUPERVISOR

CO-DIRECTEUR (CO-DIRECTRICE) DE LA THÈSE / THESIS CO-SUPERVISOR

EXAMINATEURS (EXAMINATRICES) DE LA THÈSE / THESIS EXAMINERS

B. Jodoin

S. Siolander

Gary W. Slater

Le Doyen de la Faculté des études supérieures et postdoctorales / Dean of the Faculty of Graduate and Postdoctoral Studies

Influence of Rotor Blade Scaling on the Numerical Simulation of a High Pressure Gas Turbine

Derek Lastiwka

Thesis submitted to the Faculty of Graduate and Postdoctoral Studies
in partial fulfillment of the requirements for the degree of

MASTER OF APPLIED SCIENCE

in Mechanical Engineering

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

May 14, 2009

© Derek Lastiwka, Ottawa, Canada, 2009



Library and Archives
Canada

Published Heritage
Branch

395 Wellington Street
Ottawa ON K1A 0N4
Canada

Bibliothèque et
Archives Canada

Direction du
Patrimoine de l'édition

395, rue Wellington
Ottawa ON K1A 0N4
Canada

Your file *Votre référence*
ISBN: 978-0-494-58207-7
Our file *Notre référence*
ISBN: 978-0-494-58207-7

NOTICE:

The author has granted a non-exclusive license allowing Library and Archives Canada to reproduce, publish, archive, preserve, conserve, communicate to the public by telecommunication or on the Internet, loan, distribute and sell theses worldwide, for commercial or non-commercial purposes, in microform, paper, electronic and/or any other formats.

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

In compliance with the Canadian Privacy Act some supporting forms may have been removed from this thesis.

While these forms may be included in the document page count, their removal does not represent any loss of content from the thesis.

AVIS:

L'auteur a accordé une licence non exclusive permettant à la Bibliothèque et Archives Canada de reproduire, publier, archiver, sauvegarder, conserver, transmettre au public par télécommunication ou par l'Internet, prêter, distribuer et vendre des thèses partout dans le monde, à des fins commerciales ou autres, sur support microforme, papier, électronique et/ou autres formats.

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

Conformément à la loi canadienne sur la protection de la vie privée, quelques formulaires secondaires ont été enlevés de cette thèse.

Bien que ces formulaires aient inclus dans la pagination, il n'y aura aucun contenu manquant.


Canada

Abstract

Rotor blade scaling is a method aimed at reducing the computational domain in turbomachinery simulations by changing the number of blades such that the blade-to-vane ratio in the scaled geometry is equal to the ratio of two small integers (e.g., 1/1, 1/2, 1/3, 2/3, 3/4 etc.), but without changing the airfoil shape. The objective of this study is to determine the quantitative effects that rotor blade scaling has on the numerical simulations of a representative single-stage gas turbine, with particular focus on parameters that influence vibratory stresses on the rotor blades. Simulations were conducted using the commercial software FLUENT 6.3.26. Results from an unscaled case were compared to LDV measurements and to two scaled cases. Average computing times for the three cases confirmed that the cases with the smallest computational domain were the most efficient. The present results demonstrate that blade scaling affects significantly the magnitude of unsteady pressure fluctuations, which influence the level of vibratory stresses on the blades. Moreover, scaling has been found to affect the size of blade wakes and the magnitude of losses in the blade wakes. As the rotor blade size increased pressure fluctuations on the surface of the rotor blade increased. Pressure fluctuations on the surface of the stator vane decreased as the blade to vane ratio more closely approached an integer number. These results suggest that, in numerical simulations of gas turbine operation, scaling should be kept to very low levels to avoid strong departures from the predicted performance of the unscaled geometry. The present work addresses the interest of the aerospace industry in using standardized CFD tools and reducing computational time without seriously compromising simulation accuracy.

to James McBryan. If only everyone enjoyed life as much as he did.

Acknowledgments

Without my parents, June and Dennis Lastiwka, my accomplishments would not have been possible. A simple thanks can never be enough for their encouragement, freedom and support they gave me.

I would also like to thank my supervisor, Dr. Stavros Tavoularis, for his constant guidance, patience and assistance with this project and for the opportunities that he presented to me. Recognition goes out to Dr. Dongil Chang for his support and help that was available everyday and for conducting the final simulations. This research could not have been completed without the financial support received from P&WC, NSERC, the University of Ottawa, and HPCVL. Thanks goes out to them for their choice to invest in our project over the many other available projects.

A great amount of gratitude must be given to my friends and colleagues; Andrew Cameron, George Choueiri, Amandine Hamel, Michael Paciocco, Julia Turriff and Christina Vanderwel for their help, and participation in stress relief events during this time. Last but not least, I would like to thank my Calgary friends for their long distance support that is never easy.

Contents

Nomenclature	vii
List of Figures	xviii
List of Tables	xxii
1 Introduction	2
1.1 Motivation	2
1.2 Objective and Scope	6
1.3 Outline	7
2 Literature Review	9
2.1 Computational Methods for Turbomachinery	9
2.2 Domain Scaling Method	12
2.2.1 Time-mean Rotor Blade Surface Pressure	14
2.2.2 Unsteady Rotor Blade Surface Pressure	15
2.2.3 Passage Vortices	17
2.2.4 Secondary Flows	17
2.2.5 Pressure Losses	18
2.2.6 Summary	18
3 Fluid Mechanical Background	20
3.1 Flows in Gas Turbines	20
3.1.1 Tip Leakage Flow	21
3.1.2 Shock Waves	23
3.1.3 Wakes	24
3.1.4 Secondary Flows	24
3.1.5 The Hub Passage Vortex	28
3.2 Quantification of Losses	31
3.3 Domain Scaling Method	37
3.4 Coherent Structure Identification	42

4	Numerical Background	44
4.1	Governing Equations	44
4.1.1	Reynolds Averaged Navier-Stokes Equations	45
4.1.2	Favre Averaged Navier-Stokes Equations	47
4.1.3	Modelling Rotation	48
4.2	Turbulence Modelling	53
4.2.1	Spalart-Allmaras (S-A) model	54
4.2.2	Standard $k - \varepsilon$ model	55
4.2.3	RNG $k - \varepsilon$ model	55
4.2.4	Realizable $k - \varepsilon$ model	56
4.2.5	Shear Stress Turbulence (SST) model	56
4.2.6	Reynolds Stress Model (RSM)	58
4.3	Spatial Discretization	59
4.4	Simulation Verification	63
4.4.1	Grid Convergence Index (GCI) Method	66
4.4.2	Approximate Error Spline (AES) Method	70
5	Overview of Relevant Experimental Studies	76
5.1	T106 Low Pressure Turbine Cascade	76
5.1.1	Experimental Conditions	78
5.1.2	Static Surface Pressure Measurement	80
5.2	TTM High Pressure Turbine	81
5.2.1	Experimental Conditions	81
5.2.2	Measurement Planes	83
5.2.3	LDV System	85
6	Numerical Conditions and Procedures	89
6.1	Hardware and Software	90
6.2	Computational Domain and Grid Generation	90
6.2.1	T106 Cascade	90
6.2.2	TTM Turbine	94
6.3	Computational Conditions	101
6.3.1	T106 Cascade	102
6.3.2	TTM Turbine	103
6.4	Numerical Schemes	110
6.4.1	T106 Cascade	110
6.4.2	TTM Turbine	111
6.5	Simulation Methodology	112
6.6	Simulation Uncertainty	113
7	Preliminary Simulations	120
7.1	Simulation Verification	122
7.2	Turbulence Model Validation	125

8	Reference Simulations and Comparison with Experimental Results	133
8.1	Simulation Verification	134
8.2	Simulation Validation	137
9	Comparative Assessment of Scaling Effects	156
9.1	Computational Gain by Scaling	159
9.2	Turbine Performance Parameters	161
9.3	Rotor Blade Surface Pressure	163
9.3.1	Time-Averaged Pressure	163
9.3.2	Pressure Fluctuation Amplitude	165
9.3.3	Unsteady Pressure	167
9.4	Stator Vane Surface Pressure	172
9.4.1	Time-Averaged Pressure	172
9.4.2	Pressure Fluctuation Amplitude	173
9.4.3	Unsteady Pressure	173
9.5	In-Flow Pressure	176
9.5.1	Pressure Fluctuation Amplitude	180
9.6	Turbine Losses	180
9.6.1	Time-Averaged Losses	182
9.7	Turbulent Kinetic Energy	184
9.7.1	Time-Averaged Turbulent Kinetic Energy	184
9.8	Vorticity	184
9.8.1	Time-Averaged Vorticity	185
9.9	Unsteady In-Flow Properties	187
9.9.1	Pressure	187
9.9.2	Losses	190
9.10	Concluding Remarks	193
10	Conclusions and Recommendations for Future Work	198
10.1	Conclusions	198
10.2	Recommendations for Future Work	200
	Appendix A. Variable Tip Gap	210
	Appendix B. Post-Processing	213

Nomenclature

A	area
a_1	constant associated with the SST turbulence model
C_a	coefficient of axial force
C_f	coefficient of shear stress
C_l	coefficient of lift
C_p	coefficient of pressure
C_p	specific heat at constant pressure
C_θ	coefficient of tangential/circumferential force
c	chord
c_q	strain rate modification constant used in modified Q -criterion
D	number of spatial dimensions
D_h	hydraulic diameter
D_{hub}	turbomachine local hub diameter
d	average length of one side of a cell in a CFD grid
d_p	average seeding particle diameter
E	uncertainty

E_a	approximate error
E_f	estimated fractional error
E_t	estimated true error
e	specific internal energy
F	force
f_v	fundamental vane passing frequency
G	constant associated with the SST model; 0 in shear flows and 1 in boundary layer flows
g	any flow parameter of interest
g_{exact}	true value of any flow parameter of interest
h	specific enthalpy
K	constant associated with the GCI and AES methods; 3 when the order of convergence is estimated from 2 grids and 1.25 when the order of convergence is estimated from 3 grids
k	turbulent kinetic energy or kinetic energy of unresolved velocity fluctuations
k_{diss}	dissipation of turbulent kinetic energy
k_{prod}	production of turbulent kinetic energy
k_{tot}	kinetic energy of resolved and unresolved velocity fluctuations
L	lift force
LE	leading edge
L_{rp}	rotor blade pitch

L_{sp}	stator vane pitch
L_{se}	location of flow separation
L_{re}	location of flow reattachment
M	torque
Mn	Mach number
MW	molecular weight
m	number of nodes
m_{cells}	number of cells
$\dot{m}$	mass flow rate
N_{rt}	number of rotor blades
N_{st}	number of stator vanes
P	static pressure
p	order of convergence
Q	threshold value used for identification of coherent structures based on the Q -criterion
Q_m	threshold value used for identification of coherent structures based on the modified Q -criterion
q	scaling ratio
R	local radius of curvature
$\mathbf{R}$	gas constant
Re	Reynolds number
r	vector magnitude or distance

S	span
S_{ij}	strain rate tensor
s	specific entropy
s, b, n	streamwise coordinate, spanwise coordinate, and coordinate normal to streamline in circumferential direction
T	temperature
TE	trailing edge
t	time
U	velocity magnitude
u, v, w	streamwise, crossflow/secondary flow, and spanwise velocity components
u_i, u_j, u_k	velocity vector in index notation
u_a, u_θ, u_r	axial, tangential/circumferential, radial velocity components
u_x, u_y, u_z	velocity components in cartesian coordinate system
u_A, u_B	velocity components along streamlines AAA and BBB
V	volume
W	power or work
a, r, θ	axial, radial and circumferential coordinates respectively (cylindrical coordinates)
x, y, z	cartesian coordinates
Y_p	total pressure loss coefficient
y^+	non dimensional distance from the wall ($= \rho y u_\tau / \mu_w$)

Greek symbols

α	blade inlet flow angle
β	blade outlet flow angle
γ	ratio of specific heats
Δ	difference in property
δ	relative change in a flow parameter
δ_{ij}	kronocker delta
ε	turbulent dissipation rate
ζ_h	enthalpy loss coefficient
ζ_s	entropy loss coefficient
η_{tt}	total-to-total/isentropic efficiency
η_{ts}	total-to-static efficiency
θ_{rp}	angle of rotationally periodic sector of full turbomachine
κ	coefficient of thermal conductivity of material
κ_{eff}	effective coefficient of thermal conductivity ($= \kappa + \kappa_t$)
κ_t	coefficient of thermal conductivity due to turbulence
Λ	ratio between the number of rotor blades and the number of stator vanes in a stage
Λ_{sc}	ratio between the number of rotor blades and the number of stator vanes in a scaled stage

μ	dynamic viscosity
μ_{eff}	effective turbulent dynamic viscosity ($= \mu + \mu_t$)
μ_t	turbulent dynamic viscosity
ν_t	turbulent kinematic viscosity
ξ	under relaxation factor
ρ	density
ρ_p	seeding particle density
σ	grid refinement ratio
τ_c	seeding particle time constant
τ_w	wall shear stress
τ_t	turbulent shear stress
τ_{ij}	components of the viscous stress tensor
τ_p	the simulation rotational period, which is the time it takes the rotor to rotate by an angle equal to the angle between the periodic boundaries
τ_r	the rotor blade passing period, which is the time it takes the rotor to rotate by an angle equal to the angle between two consecutive rotor blades
τ_s	the stator vane passing period, which is the time it takes the rotor to rotate by an angle equal to the angle between two consecutive stator vanes
Φ	residual for a given flow parameter

Ω	turbomachine rotational speed
Ω_{ij}	rotation tensor
ω	specific dissipation rate
ω_{mag}	vorticity magnitude
$\omega_s, \omega_b, \omega_n$	streamwise, spanwise, and normal components of vorticity
$\omega_x, \omega_y, \omega_z$	components of vorticity in cartesian coordinates
$\omega_a, \omega_r, \omega_\theta$	axial, radial, and tangential/circumferential components of vorticity
$\omega_i, \omega_j, \omega_k$	vorticity vector in index notation

Subscripts

A, B	associated with streamline AAA or BBB
a, θ, r	axial, tangential/circumferential, and radial components
amb	ambient conditions
$a\theta$	composed of tangential and axial velocity components
$Bulk$	bulk property
$B1$	on plane B1
$C1$	on plane C1
c	associated with the airfoil chord
cas	associated with the turbomachine casing
cc	associated with cycle-to-cycle variations
cv	associated with in cycle variations

D_h	hydraulic diameter
d	associated with drag
$exit$	associated with the exit chanber in TTM experiments
hub	associated with the hub of the turbomachine
I	grid or index number
i, j, k	index notation
in	conditions or property at inlet boundary for simulations
LE	associated with the leading edge of vane or blade
l	associated with lift
MC	associated with the mixing chamber in TTM experiments
max	maximum value
n	final value
out	conditions immediately downstream of rotor in TTM experiments; property at outlet boundary for simulations
prs	associated with the pressure side of the airfoil
rel	in the relative frame of reference
res	resolved component
rot	associated with the rotating frame
rt	associated with the rotor
SD	standard deviation
s, b, n	streamwise, spanwise and normal components
st	associated with the stator

<i>sc</i>	scaled geometry parameter
<i>shift</i>	shifted coordinates
<i>sim</i>	associated with the CFD simulation model
<i>suc</i>	associated with the suction side of the airfoil
<i>spin</i>	due to the motion of the rotating frame
<i>t</i>	turbulent parameter
<i>ures</i>	unresolved component
<i>x, y, z</i>	cartesian coordinate components
∞	inlet or farfield condition
0	total or stagnation property
12, 23	associated or calculated with grids or cases 1 and 2 or grids or cases 2 and 3
1, 2, 3	property at state 1, 2 or 3 or associated with the fine, intermediate or coarse grids
01, 02, 03	total property at state 1, 2 or 3
01s, 02s, 03s	total isentropic property at state 1, 2 or 3
1s, 2s, 3s	isentropic property at state 1, 2 or 3
01ss, 02ss, 03ss	total property at state 1, 2 or 3 after isentropic stage
1ss, 2ss, 3ss	state 1, 2 or 3 after isentropic stage

Other notation

$\vec{g}$	vector
$\overline{\overline{g}}$	space-time area weighted average
$\widetilde{\overline{g}}$	space-time mass weighted average
$\bar{g}$	time average
$\tilde{g}$	density weighted average
$\langle g \rangle$	ensemble of realizations
g'	fluctuating component
$\{g''^2\}$	velocity fluctuations, which are not resolved by the solution of the URANS equations
$\{u''_i u''_j\}$	Reynolds stresses
ϵ_{ijk}	Levi-Civita tensor
1, 2, 3	state 1, 2 or 3
01, 02, 03	total property at state 1, 2 or 3
01s, 02s, 03s	total property at state 1, 2 or 3 after isentropic process
1s, 2s, 3s	state 1, 2 or 3 after isentropic process
1ss, 2ss, 3ss	state 1, 2 or 3 after isentropic stage

Acronyms

CFL	Courant-Friedrichs-Lewy number
CFD	computational fluid dynamics
DNS	direct numerical simulation
DoE	Department of Energy
FANS	Favre averaged Navier-Stokes equations
GCI	grid convergence index
HPCVL	<i>High Performance Computing and Virtual Laboratory</i>
LDV	laser Doppler velocimetry
LES	large eddy simulation
RANS	Reynolds averaged Navier-Stokes equations
RNG	renormalization group
RSM	Reynolds stress model
S-A	Spalart-Allmaras
SST	shear stress turbulent
TTM	Thermal Turbomachinery and Machine Dynamics
UFANS	unsteady Favre averaged Navier-Stokes equations
URANS	unsteady Reynolds averaged Navier-Stokes equations

List of Figures

3.1	Schematic illustration of the developement of secondary flow in a rotor blade passage (Lakshminarayana (1996)).	26
3.2	Passage vortex model of Takeishi et al. (1990).	29
3.3	Horseshoe vortex topology (Xun et al. (2005)).	31
3.4	Enthalpy-entropy diagram for the expansion process in a single-stage turbine (based on Lakshminarayana (1996)).	35
3.5	Schematic diagram of a single stage turbine.	40
4.1	Schematic diagram of a rotating domain (based on Ansys (2006)).	50
4.2	Schematic of a sliding mesh interface (based on Ansys (2006)).	52
4.3	H-type structured grid.	62
4.4	Unstructured tetrahedral grid.	64
4.5	Hybrid mesh with structured cells near wall regions and unstructured cells elsewhere.	65
4.6	Schematic representation of the GCI.	71
4.7	Plots of approximate error as a function of cell size for monotonic and oscillatory convergence.	73
4.8	Plots of flow parameter as a function of cell size for monotonic and oscillatory convergence.	74
5.1	Experimental setup for T106 experiments (Stieger et al. (2003)).	77
5.2	Detail of T106 cascade (Stieger (2002)).	78
5.3	T106 blade profile.	79
5.4	Profile of the TTM test section. Stator vanes and rotor blades are shown on the left and right respectively; Dimensions are in mm (Göttlich et al. (2004)).	81
5.5	Three-dimensional view of 2 stator vanes and 3 rotor blades in the TTM turbine stage and relative locations of planes B1 and C1 (Göttlich et al. (2004)).	82
5.6	TTM experimental boundary conditions (Göttlich et al. (2004)).	84
5.7	TTM LDV traverse support (Göttlich et al. (2004)).	86
5.8	LDV velocity measurement directions (Göttlich et al. (2004)).	86

5.9	Photograph showing the window in the casing of the TTM turbine used for the LDV measurements (Göttlich et al. (2004)).	88
6.1	T106 computational domain with boundary conditions.	91
6.2	Schematic of the structured mesh in the near wall regions.	92
6.3	Various grids used to model the T106 cascade.	93
6.4	TTM model.	95
6.5	TTM surface mesh for grid 3.	96
6.6	TTM surface mesh for grid 2.	97
6.7	TTM surface mesh for grid 1.	97
6.8	A comparison of the grid around the rotor blades for the original mesh (grid 2) and the refined mesh.	98
6.9	Global position of rotor profile data and effect of scaling on the scaled profile location.	100
6.10	Profiles of rotor blade tips for the three cases, viewed in the spanwise direction from the casing to the hub.	101
6.11	Profiles of rotor blades, viewed in the tangential direction and in the direction of rotation.	102
6.12	TTM boundary conditions.	107
6.13	TTM boundary conditions.	107
6.14	TTM rotor boundary conditions.	109
7.1	Sketch of the T106 turbine blade with applicable forces.	121
7.2	x -component of the coefficient of wall shear stress $C_f = 2\tau_{w,x}/\rho_{in}U_{in}^2$ on the suction side of the T106 blade for various grid densities ($\tau_{w,x}$ is the x -component of the wall shear stress).	124
7.3	Vector plots, focused on the trailing edge of the T106 turbine blade. Vector colours correspond to the values of the non-dimensionalized velocity magnitude U/U_∞	126
7.4	Non-dimensionalized time averaged turbulent kinetic energy k/U_{in}^2 near the blade trailing edge.	127
7.5	Pressure distribution along the T106 blade for various turbulence models.	130
7.6	Pressure distribution along the T106 blade for various turbulence models.	131
7.7	x -component of the coefficient of wall shear stress $C_f = 2\tau_{w,x}/\rho_{in}U_{in}^2$ over T106 blade for various turbulence models.	132
8.1	Relative positions of rotor blades at different times during the cycle; each plot is viewed in a direction normal to a plane tangent to the hub at a location indicated by the dashed line; the entire spans of vanes and blades are shown.	138
8.2	Comparison between ensemble average of tangential velocity and cycle-to-cycle fluctuations of the tangential velocity from the simulation.	139
8.3	Comparison of predicted and measured (Göttlich et al. (2004)) non-dimensionalized velocities $U_{a\theta}/\bar{U}_{a\theta}$ on plane B1.	141
8.4	Detail of shock after stator vanes.	142

8.5	Comparison of predicted and measured (Göttlich et al. (2004)) non-dimensionalized velocities $U_{a\theta}/\bar{U}_{a\theta}$ on plane C1.	145
8.6	Predicted isocontours of entropy, showing the influence of rotor blade position on stator wakes at $t/\tau_r = 0$ and 50% span. Stator vanes are indicated as sv1 and sv2, whereas rotor blades are indicated as rb1, rb2 and rb3; entropy increases from blue towards red.	146
8.7	Predicted isocontours of time-averaged entropy loss coefficient $\bar{\zeta}_s$ on plane C1; high entropy loss coefficient near the root of the rotor blade gives possible indication to existence of passage vortices.	147
8.8	Kinetic energies of various velocity fluctuations and variations for the simulation results at $r = 238$ mm on plane C1.	151
8.9	Kinetic energies of various velocity fluctuations and variations for the experimental results at $r = 238$ mm on plane C1.	152
8.10	Ensemble average of tangential velocity at $r = 238$ mm on plane C1.	153
8.11	Comparison of predicted and measured (Göttlich et al. (2004)) non-dimensionalized turbulent kinetic energies $k_{a\theta}/\bar{U}_{a\theta}^2$ on plane C1.	154
8.12	Predicted and measured non-dimensionalized turbine power output.	155
9.1	Relative positions of rotor blades for the three simulation cases at 4 time steps and at 50% span; one blade that serves as reference in the discussion (rb1) is indicated as darker than the others; stator wakes are shown as areas of large entropy (entropy increases from blue to red).	157
9.2	Locations of planes used for visualization of isocontours.	160
9.3	Comparison of non-dimensionalized unsteady turbine power output between cases.	162
9.4	Time averaged surface pressure distributions over a rotor blade for the three cases considered.	164
9.5	Time averaged pressure distribution over a rotor blade at 10% span.	165
9.6	Time averaged pressure distribution over a rotor blade at 50% span.	166
9.7	Time averaged pressure distribution over a rotor blade at 90% span.	166
9.8	Pressure fluctuation amplitude over the surface of a rotor blade for the 3 cases.	168
9.9	Isocontours of the time-varying surface pressure coefficient $P/(\frac{1}{2}\bar{\rho}_{Bulk}\bar{U}_{Bulk}^2)$ on the rotor blade at 10% and 50% spans; positive values of x/c_a indicate the pressure side of the rotor blade while negative values of x/c_a indicate the suction side of the rotor blade.	169
9.10	Isocontours of the time-varying surface pressure coefficient $P/(\frac{1}{2}\bar{\rho}_{Bulk}\bar{U}_{Bulk}^2)$ on the rotor blade at 90% span; positive values of x/c_a indicate the pressure side of the rotor blade while negative values of x/c_a indicate the suction side of the rotor blade.	170
9.11	Power spectral density (PSD) of pressure fluctuations on the suction side of a rotor blade at 10% span and 10% axial chord.	171
9.12	Power spectral density (PSD) of pressure fluctuations on the suction side of a rotor blade at 10% span and 10% axial chord. Detail of fundamental frequency and first harmonic.	172

9.13	Time averaged surface pressure $\overline{P}/\frac{1}{2}\overline{\rho}_{Bulk}\overline{U}_{Bulk}^2$ distribution on a stator vane for the three cases considered.	174
9.14	Pressure fluctuation amplitude $P_{SD}/\frac{1}{2}\overline{\rho}_{Bulk}\overline{U}_{Bulk}^2$ over the surface of a stator vane for the 3 cases.	175
9.15	Isocontours of the time-varying surface pressure coefficient $P/\frac{1}{2}\overline{\rho}_{Bulk}\overline{U}_{Bulk}^2$ on the stator vane at 10% and 50% spans and from 60% to 100% of the stator vane axial chord on the suction side of the stator vane.	177
9.16	Isocontours of the time-varying surface pressure coefficient $P/\frac{1}{2}\overline{\rho}_{Bulk}\overline{U}_{Bulk}^2$ on the stator vane at 90% span and from 60% to 100% of the stator vane axial chord on the suction side of the stator vane.	178
9.17	Power spectral density (PSD) of pressure fluctuations on the suction side of a stator vane at 10% span and 85% axial chord.	179
9.18	Power spectral density (PSD) of pressure fluctuations on the suction side of a stator vane at 10% span and 85% axial chord. Detail of fundamental frequency.	179
9.19	Predicted isocontours of the amplitude of pressure fluctuations $P_{SD}/\overline{P}_{01}$ on plane B1.	181
9.20	Predicted isocontours of the amplitude of pressure fluctuations $P_{SD}/\overline{P}_{01}$ on plane B3.	181
9.21	Predicted isocontours of the amplitude of pressure fluctuations $P_{SD}/\overline{P}_{01}$ on plane C1.	182
9.22	Predicted isocontours of entropy loss coefficient $\overline{\zeta}_s$ on plane B3.	183
9.23	Predicted isocontours of entropy loss coefficient $\overline{\zeta}_s$ on plane C1.	183
9.24	Predicted isocontours of time-averaged turbulent kinetic energy $\overline{k}/\overline{U}_{Bulk}^2$ on plane C1.	185
9.25	Predicted isocontours of time-averaged dimensionless vorticity magnitude $\overline{\omega}_{mag}c/\overline{U}_{Bulk}$ on plane C1.	186
9.26	Instantaneous isosurfaces with $Q_m = 1$ used to identify coherent structures.	187
9.27	Predicted isocontours of pressure $P/\overline{P}_{01}$ at 10% span. Shock is graphically approximated by a black line that was manually added.	189
9.28	Predicted isocontours of pressure $P/\overline{P}_{01}$ on plane B1.	191
9.29	Predicted isocontours of pressure $P/\overline{P}_{01}$ on plane C1.	192
9.30	Predicted isocontours of the entropy loss coefficient $\overline{\zeta}_s$ on plane C1.	194
.1	Side profile of a TTM rotor blade and scaled cases with variable tip gap	211
.2	Predicted isocontours of entropy loss coefficient on plane C1	212

List of Tables

4.1	Grid labelling for a verification study with only 2 grids	66
4.2	Grid labelling for a verification study with 3 grids	66
4.3	Grid labelling for a verification study with only more than 3 grids	66
5.1	T106 low pressure turbine cascade specifications	77
5.2	TTM turbine stage properties	82
5.3	TTM ambient experimental conditions	83
5.4	TTM experimental boundary conditions	84
5.5	TTM LDV optical beam system	88
6.1	Summary of cases investigated	99
6.2	T106A simulation conditions	103
6.3	Gas properties for air	104
6.4	TTM simulation conditions	105
6.5	Simulation parameters	113
7.1	Predicted time-averaged parameters for the different grids using the SST turbulence model	123
7.2	Error estimates for the T106 study	124
7.3	Comparison of tangential and axial force coefficients predicted using different turbulence models	125
8.1	Comparison of simulation results using different grids for the TTM turbine	135
9.1	Flow properties used for normalization	159
9.2	Time averaged turbine performance parameters for scaled and unscaled cases	162
.1	Time averaged turbine performance parameters for scaled and unscaled cases	211

Chapter 1

Introduction

Gas turbines are used extensively in transportation systems, including aeroplanes, trains and ships. They are also used for power generation, particularly in applications where portability is required along with a large power output. Improvements in the design and efficiency on these machines have significant implications on many other industries. Improved efficiency results in lower fuel consumption, lower operating costs, and less adverse environmental impact. Better designs lead to lower manufacturing and maintenance costs, longer engine life, larger power output and reduced size and weight of the engine.

1.1 Motivation

Before the application of *Computational Fluid Dynamics* (CFD), turbomachinery designers relied on flow models which were based on many simplifying assumptions. This was necessary because performance calculations were done manually and the use of complex models would have required exceedingly long times to complete. In addition, complexity

would increase the probability of error. With the advent of CFD, it became possible to develop more complex models, which would presumably improve the prediction of flow through the turbomachine. The availability of higher computing power permitted the use of even more complex models.

In principle, CFD analysis with the most accurate models could be done on an entire engine, including the compressor, combustion chamber and turbine(s). However, present levels of computing power are not sufficient for the accurate simulation of entire engines, at least not on a routine basis, as required for industrial design purposes. Yao et al. (2002) completed a simulation of a 1.5 stage turbine with 6 rotor blades and 7 stator vanes in the first stage and 6 rotor blades in the following half stage. In addition, only 1/6th of the turbomachine was modeled. This domain required 13 505 762 nodes to discretize. The simulation was run on 187 processors on the US Department of Energy Blue Pacific IBM SP2 at Lawrence Livermore Laboratory. Even with simplifications and the use of a supercomputer, the simulation required 27.3 minutes per time step. Considering that accurate resolution of unsteady effects requires 700 time steps per 1/6th rotation of the machine, one may estimate that the simulation of a 180° rotation cycle would require 80 days of CPU time. Such requirements make it impossible to use this approach for design purposes, for which many alternative designs have to be evaluated in a short period of time. Therefore, it is obvious that simulation time needs to be reduced drastically for CFD to serve as a practical tool early in the design process of turbomachines. Simulation time can be reduced by the following approaches:

- increase in computer speed

- increase in the number of computers operating in parallel
- improvement in algorithms
- reduction in the number of nodes used to discretize the domain
- reduction of domain size.

Although the speed and efficiency of computers have increased steadily over the past decades, they have not yet reached levels that would permit the practical simulation of a full gas turbine. The connection of many computers in clusters also has its limitations, because, as the number of connections increases, communication time between computers also increases, thus affecting adversely the simulation time. In addition, the cost of acquiring additional computers and software licences may pose a limit to the use of this approach by industry.

The development of more efficient algorithms and codes is an ongoing process, but not normally among the tasks of industrial designers and analysts, who rely on well-tested in-house or commercial software. General-purpose, commercial CFD codes that can solve many different problems may not be entirely up-to-date and may be less efficient in some aspects than specialized codes, but are often preferred by users familiar with them because of the convenience of having access to extensive documentation and technical support.

Reduction in the number of discretization nodes generally results in lower simulation accuracy, so that it can only be applied up to a limit. It may also be achieved by the use of higher-order discretization elements, as, for example, polyhedra cells. This may not be an option, however, because some CFD software may not offer higher order cells for mesh

generation and also because the use of such elements may be subjected to some restrictions which make them unsuitable for certain simulations (for example, mesh adaptation may not be an available option when polyhedra cells are used).

Reduction in the domain size is an attractive option, because it is done prior to the application of the mesh generation and solution algorithms and so it is within the powers of the CFD user to apply. In turbomachinery simulations, reduction of the domain size is typically accomplished with the use of rotational periodic boundary conditions. For example, if a turbine stage were composed of 36 stator vanes and 36 rotor blades, it would be sufficient to compute the flow through a single stator and rotor passage, namely a 10° sector and apply rotational periodic boundary conditions on the two side surfaces of this domain. In actual turbine stages, however, the number of rotor blades is not equal to or a multiple of the number of stator vanes (typically, the number of blades far exceeds the number of vanes). In many cases, turbomachinery designers reduce resonance effects on turbine blades by constructing gas turbines with blade rows containing a prime number of blades or with a low common factor between the number of rotor blades and stator vanes (Arnone and Pacciani (1996)). If this is the case, then one would need to model the entire turbomachine, or at least a significant portion of it taking advantage of rotational periodicity. Because these large computational domains require significant computational time and resources, methods of reducing the computational domain have been developed.

A popular domain reduction method involves changing the number of blades in the simulation geometry, while at the same time “scaling” the rotor blades (i.e., modifying their size, while leaving their shape unaffected). Because of the required geometrical changes,

one would expect that the simulation results would be distorted as well. The amount of distortion is a topic which has been given rather little attention in the literature and mostly as a side study, rather than being the focus of a specific investigation. A literature survey identified a single article that specifically compares results of scaled and unscaled simulations and this was done using a simplified, two-dimensional geometry. Furthermore, this study focused on torque fluctuations over the rotor blade. Studies of scaling effects on the local pressure fluctuations on the rotor blade have only been performed for very small amounts of rotor blade scaling. One particular issue of concern to high-pressure turbine manufacturers, namely the dependence of vibratory stresses on the blade upon blade scaling used during simulations, has not been investigated at all. This is the primary focus of the present study

The current study will expand on these studies using 3-D simulations, by including the magnitude and location of local pressure fluctuations on both the rotor blades and stator vanes and with much larger scaling.

1.2 Objective and Scope

The objective of this study is to compare the results of three 3-D unsteady simulations of a single-stage high-pressure turbine, all having the same geometry of the stator vanes and the same shapes of rotor blades but each having a different number and a different size of rotor blades. This approach produces three different pairs of simulations that can be compared to each other to evaluate the effect of blade scaling. One case has a ratio of 3 blades to 2 vanes and is thus the one with the smallest periodic domain, which makes it

the most economical to simulate numerically. The reason that it is chosen as the reference case is the availability of experimental results, which may serve to validate the simulations. This will be referred to as the “unscaled case”, although in practice the simulation of such a turbine would not require any scaling at all. The other two cases have blade-to-vane ratios equal to 4:3 and 5:3, respectively, and blades which have been upscaled by 12.5% and downscaled by 10%, respectively. This amount of scaling is larger than those in previous studies and will provide information into the change in vibratory stresses that occur due to scaling as well as the change in the location of maximum vibratory stresses. Criteria used to compare the various cases will include gas turbine efficiency, losses, time-mean and unsteady global and local pressure distributions, turbomachine power output and coherent structure characteristics. A commercial CFD code was chosen for all simulations. Nevertheless, the accuracy of the results depends on the choice of options (e.g., turbulence models) and numerical procedures offered by this code. In order to establish confidence in the analysis and evaluate the performance of the various available options in a timely manner, preliminary studies utilizing a 2-D turbine cascade will be completed first.

1.3 Outline

The following Chapter provides a literature review of previous work done in the area of domain scaling as well as other methods commonly used to eliminate the problem of non-integral blade counts. Chapters 3 and 4 provide background into turbomachinery flows and numerical methods and models, respectively, to aid the reader in the understanding of the thesis. An overview of the available experimental results that have been used in the

validation of the present simulations is contained in Chapter 5, whereas the computational geometry, domain discretization, boundary conditions and other numerical conditions are given in Chapter 6. The results of 2-D simulations used to select the most suitable turbulence model are given in Chapter 7. The main results are described in Chapter 8, which contains mesh independence studies, simulation results for the unscaled case, and comparison with available measurements, and in Chapter 9, which contains a detailed comparison of results using the unscaled and two scaled geometries. Chapter 10 follows with conclusions and recommendations for future work.¹

¹After simulations conducted in the earlier stages of this work were completed, it was observed that they produced some unrealistic results, for example turbine efficiencies that were significantly different among the three cases with different blade counts (i.e., the unscaled case and the two scaled cases). Closer examination of the specified conditions indicated that the meshes were not sufficiently refined near the walls and some inconsistencies in the application of boundary conditions. Moreover, the computational geometry in these early simulations included the exit diffuser, which imposes an adverse pressure gradient with potential problems. To overcome these limitations, the simulations were repeated by Dr. D. Chang after the following changes were made:

- refined meshes were defined near all solid walls.
- instead of dividing the rotor hub section into three parts, only the central one of which was set to rotation, the entire rotor hub was rotated; this approach is consistent with industrial CFD practices.
- the diffuser was removed from the geometry; instead of specifying the static pressure at the diffuser exit, the experimental value of static pressure was specified at the rotor hub as exit boundary condition, accompanied by the radial equilibrium assumption.
- some inconsistencies in the specification of inlet boundary conditions were corrected.

A complete version of the entire thesis was produced reporting the earlier simulations, and the new results became available at a later time. To minimize the effort in reassembling the thesis, some earlier work (e.g., mesh independent study) has been included in the final version, but all reported main results correspond to the improved simulations.

Chapter 2

Literature Review

The literature review focuses on computational methods used in modelling turbomachinery flow and on results obtained by previous studies involving the use of the domain scaling method in order to reduce the computational domain size. These results are restricted to the comparison between cases that use the domain scaling method and cases which do not. The goal of which was to analyze the effect of domain scaling on the results. However, in many studies the comparison between cases using domain scaling and cases which did not was a side study and was not the focus of the paper.

2.1 Computational Methods for Turbomachinery

Various computational methods have been developed over the years to model turbomachinery flows. Steady state approaches require lower computational time than unsteady ones, however, the former are incapable of simulating several important unsteady effects. Steady state methods include the *through-flow* (Dawes (1992a), Dawes (1992b)),

mixing-plane and *average-passage* (Adamczyk (1985), Adamczyk et al. (1990)) methods. The through-flow method determines unsteady interactions using the numerical solution on a 2-D axisymmetric plane (through-flow plane) which encompasses the entire turbomachine. The 3-D flow field is then determined from the numerical solution on a 3-D domain that only encompasses one blade row (Sleiman (1999)). The information from one solution is passed to the other until convergence is reached. A disadvantage of this method is that it smears the flow field between stages (Sleiman (1999)). The mixing-plane method “mixes” the flow between blade rows using circumferential averaging. This model is widely used by industry due to its relatively low computational requirements and is available in FLUENT 6.3.26. In the average-passage method, the domain of the rotor is expanded to include the domain of upstream and downstream blade rows (without the blades). Unsteady effects are taken into account by “deterministic” stresses which are determined from the difference between the phase-averaged and average passage velocities (Meneveau and Katz (2002)). However, this method does not deal well with strong non-linearities (Nadarajah and Jameson (2006)) and the presence of stator vanes tends to lead to unrealistically high “deterministic” stresses (Meneveau and Katz (2002)).

Unsteady methods include *sliding meshes*, the *harmonic balance technique* and the *time spectral method*. The sliding mesh approach passes information from a stationary domain to a moving domain via an interface plane. The method starts by obtaining a solution to the unsteady Navier Stokes equations with the moving and stationary domains in an arbitrary position. Once the solution for that time step is completed, the moving domain is rotated, for turbomachine systems, relative to the stationary domain by an angle

$\Delta\theta = \Omega \times \Delta t$ (Ω is the rotational speed of the rotating part and Δt is the time step used in the CFD simulation) and new flow information is exchanged across the interface plane. In the harmonic balance technique, the flow is modelled using a Fourier series with terms having frequencies that are harmonics of the primary fluctuation frequency, namely the blade passing frequency (Hall et al. (2002)). This method is twice as fast as time domain solvers and can capture strong non-linear fluctuations, but can not be used with commercial solvers, as it would require substantial alterations to the CFD code. Time spectral methods use the Fourier method to discretize the governing equations in time. With this method the problem is solved in the time domain, so that it requires no significant modification to the code (Van der Weide et al. (2005)).

Many methods have been developed to account for non-integral blade counts. Most common are the following.

- Phase lagged methods, including the direct storage method (Erdos et al. (1977)), the phase lagged method with a time-inclined computational plane (Giles (1990), Giles (1988)) and the phase lagged method with shape correction (He (1990), He and Denton (1994)).
- The gradient scaling method (Sleiman (1999), Paulon et al. (1992)).
- The domain scaling method (Rai and Madavan (1990), Rai (1987)).

Erdos et al. (1977) was the first to develop a numerical method to compensate for non-integral blade counts without modelling the entire turbine. This method, called the *phase lagged method*, uses the values of flow parameters at the outlet of the stator domain at

a previous time step as values at the inlet to the rotor domain at the current time step. This requires large amounts of memory to store the required data at many previous time steps. In addition, this method assumes that the flow is temporally periodic over an assumed time interval, which may not be the case in viscous flows (Giles (1988)). Giles (1988) modified the method developed by Erdos et al. (1977) and eliminated the assumption of temporal periodicity by employing a time inclined computational plane. He (1990) and He and Denton (1994) applied shape correction and reduced the storage requirements of the method by Erdos et al. (1977) by assuming that fluctuations follow a specific mathematical model (Fourier series) at the interfaces between the rotor and stator. Therefore, only Fourier coefficients are passed from one interface to the other. The calculation of the Fourier coefficients increases computational time by approximately 15% with respect to the direct storage method (He (1990)), however, overall storage requirements are reduced. The gradient scaling method scales the circumferential gradients of flow variables at the interface between the rotor and stator domains such that they are inversely proportional to the sector angle size of the considered domain.

2.2 Domain Scaling Method

The domain scaling method presented here is based on the method developed by Rai (1987) and Rai and Madavan (1990). The problem of non-integral blade counts is overcome by scaling the blade pitch while maintaining a constant solidity ratio (ratio between rotor blade chord and rotor blade pitch). A result of the blade scaling is that the number of rotor blades change as well. The user can then select a scaling ratio that

adds or subtracts additional blades such that the number of stator vanes and rotor blades are integral numbers of each other. This method has been found to produce results with discrepancies in various parameters and flow patterns in turbomachinery flow, including the following ones.

- The time-mean blade surface pressure distribution.
- The unsteady blade surface pressure distribution.
- Passage vortices.
- Secondary flows.
- Pressure losses.

Studies using the domain scaling technique were carried out by Arnone and Pacciani (1996), Clark et al. (2000) and Yao et al. (2002).

Arnone and Pacciani (1996) performed a 2-D unsteady analysis of the first stage of a two-stage transonic 2MW turbine produced by Nuovo Pignone (PGT2). The inlet temperature of this turbine is 1370 K and its pressure ratio is 12.5:1. The stage in question has 22 stator vanes and 38 rotor blades. Therefore, without domain scaling the smallest numerical domain would contain 11 stator vanes and 19 rotor blades. Arnone and Pacciani (1996) compensated for the non-integral blade count using the method developed by Rai and Madavan (1990) and Rai (1987). Besides the unscaled case, the scaled configurations that were analyzed included a 1:2 (1 vane - 2 blades; 0.87 scaling ratio), 3:5 (1.036 scaling ratio), 4:7 (0.987 scaling ratio), 7:12 (1.0076 scaling ratio).

Clark et al. (2000) completed two 3-D unsteady simulations on the first 1.5 stages of a turbine from the PW6000 engine. The number of airfoils in the first stator, first rotor, and second stator were 36, 56, and 36, respectively. The simulation domain was simplified through rotational periodic boundary conditions. This resulted in the first simulation domain that was a 1/4th sector of the entire turbine. In the second simulation, rotor blade scaling reduced the domain size to a 1/18th sector of the entire turbine, while the airfoil was shifted so that the axial gap remained identical to the one in the unscaled case.

Yao et al. (2002) simulated 3-D unsteady flow in the Aachen 1.5 stage axial flow turbine. The airfoil counts for the first stator, first rotor, and second stator are 36, 41, and 36, respectively. Because 41 is a prime number, the simulation domain can not be simplified by the application of rotational periodic boundary conditions. Therefore, Yao et al. (2002) simulated two scaled cases: one with a stator/rotor/stator configuration of 6:7:6 (0.9762 scaling ratio) and the other with a 1:1:1 (1.1389 scaling ratio) configuration.

2.2.1 Time-mean Rotor Blade Surface Pressure

Arnone and Pacciani (1996) found that a 13% reduction in the rotor blade pitch (1:2 stator/rotor configuration) led to premature choking in the rotor passage and a lower stator vane exit velocity. As a result, the time averaged stator vane and rotor blade pressure distributions were significantly affected such that the lift forces on the vanes and blades were reduced. This effect was largely reduced in the 3:5 configuration, and was essentially nonexistent in the 4:7 configuration with respect to the 1:2 configuration.

It was determined by Clark et al. (2000) that a blade scaling ratio of 1.03 did not significantly affect the time-averaged pressure distributions over rotor blades and stator

vanes.

Yao et al. (2002) found slight discrepancies between the 1:1:1 configuration and the 6:7:6 configuration in the pressure profile over both surfaces of rotor blades and the suction side of stator vanes. On the suction surface of the stator vanes of the 1:1:1 configuration, they predicted higher pressures, than those for the 6:7:6 configuration, forward of the region of lowest pressure. Aft of the region of lowest pressure, the 1:1:1 configuration had lower predicted pressures than the 6:7:6 configuration. On the suction surface of the rotor blades, the 1:1:1 configuration had lower predicted pressures than the 6:7:6 configuration, from the leading edge to the point of lowest pressure. On the pressure surface of the rotor blade, the 1:1:1 configuration had lower predicted pressures than the 6:7:6 configuration, over the entire surface.

2.2.2 Unsteady Rotor Blade Surface Pressure

Although the time-mean pressure distributions remain, essentially, unaffected by blade scaling, the unsteady pressure distributions are significantly affected. Clark et al. (2000) were able to prove that 90% of the temporal fluctuations in static pressure are associated with the fundamental blade passing frequency and its first harmonic.

With respect to the unsteady lift coefficient, Arnone and Pacciani (1996) found that the 1:2 configuration was significantly different in amplitude, mean, and major fluctuation frequency when compared to the 11:19 configuration. The 3:5 and 4:7 configurations had some discrepancies compared to the 11:19 configuration as well, although these were substantially lower than discrepancies in the 1:2 configuration. For the 3:5 configuration, the lift coefficient spectral power was underpredicted at 1, 3 and 5 times the rotor blade

passing frequency (BPF). For the 4:7 configuration, the lift coefficient spectral power was overpredicted at the BPF and underpredicted at 4 times the BPF. Differences between the temporal variations of the lift coefficients in the 7:12 configuration and the 11:19 configuration were insignificant. Arnone and Pacciani (1996) concluded that a change in the rotor pitch of the unscaled case by less than 1% has a negligible effect on the simulation results.

Clark et al. (2000) found that, with a blade scaling ratio of 1.03 and at the fundamental blade passing frequency, the major differences in the pressure fluctuations between the scaled and unscaled models, occurred at the tip of the rotor. Moreover, the scaled model predicted smaller pressure fluctuations forward of the mid-chord and greater pressure fluctuations aft of the mid-chord. The difference in pressure fluctuations between scaled and unscaled models was found to be even greater at the first harmonic of the blade passing frequency. The scaled model predicted high pressure fluctuations in the tip region of the rotor blade and lower pressure fluctuations near the root. Clark et al. (2000) concluded that the scaled model underpredicts the overall pressure fluctuations, which can lead to underprediction of stresses on rotor blades.

Yao et al. (2002) determined that, whereas the overall maximum amplitude of pressure fluctuations did not vary a lot, the local amplitude of pressure fluctuations in the 1:1:1 configuration could vary significantly from that in the 6:7:6 configuration. Blade scaling with a scaling ratio of 1.1389 reduced the spectral energy of pressure fluctuations at higher harmonics of the blade passing frequency. Pressure fluctuations at frequencies lower than the fundamental blade passing frequency were attenuated as well. The largest differences in pressure fluctuations were located in the end wall regions, with the strongest

fluctuations occurring on the suction side of the blades.

Conclusions drawn by Yao et al. (2002) were similar to those by Clark et al. (2000). As blade scaling was increased, pressure fluctuations became underpredicted at blade passing frequency harmonics above the fundamental frequency. However, Yao et al. (2002) found that pressure fluctuations were overpredicted at the fundamental blade passing frequency, as blade scaling increased.

2.2.3 Passage Vortices

Blade scaling also changes the location of passage vortices. Yao et al. (2002) speculated that this is due to the change in the size of rotor blades. It has been shown by Yao et al. (2002) that, in the 1:1:1 configuration, the location of the stator vane tip passage vortex was predicted to be closer to the hub than in the 6:7:6 configuration. Similarly, predictions using the 1:1:1 configuration place the locations of the rotor trailing edge vortices and the rotor tip vortex closer to the hub than predictions with the 6:7:6 configuration. The locations of passage vortices in the stator and rotor of the 6:7:6 configuration agreed with experiments more closely than those in the 1:1:1 configuration.

2.2.4 Secondary Flows

Different types of secondary flows are encountered in turbomachinery and the accuracy of their prediction is difficult to quantify. Yao et al. (2002) defined secondary flow as the difference between the local flow vector and an average flow vector. They also determined visually that, as blade scaling is increased above from the 6:7:6 configuration to the 1:1:1 configuration, secondary flow features, such as tip and passage vortices, were

poorly captured or not captured at all. Yao et al. (2002) suggested that this is due to the increased azimuthal pitch that occurs as the blade scaling increases.

2.2.5 Pressure Losses

The scaled model's inability to capture secondary flow features affects the prediction of pressure loss as well. Yao et al. (2002) showed that the distribution and magnitude of pressure loss depends on the secondary flow field, which in turn depends on blade scaling. Therefore, a poorly predicted secondary flow field leads to a poorly predicted pressure loss.

2.2.6 Summary

It was determined that time-averaged pressure distributions over blades and vanes are not significantly affected by blade scaling with ratios of 1.14 (Yao et al. (2002)) and 1.03 (Clark et al. (2000)). Arnone and Pacciani (1996) found that blade scaling ratios of 1.036, 1.0076 and 0.987 did not appreciably affect the unsteady lift coefficient, whereas the unsteady lift coefficient was affected by a scaling ratio of 0.87. Unsteady pressure fluctuations are strongly affected by 14% blade scaling (Yao et al. (2002)) and 3% blade scaling (Clark et al. (2000)). Yao et al. (2002) and Clark et al. (2000) have also shown that blade scaling with ratios of 14% and 3% respectively, underpredict pressure fluctuations at harmonics of the fundamental blade passing frequency. At the fundamental blade passing frequency, scaling led to overprediction of pressure fluctuations. Arnone and Pacciani (1996) found that the lift coefficient spectral power at the fundamental blade passing frequency was overpredicted in the 1:2 and 4:7 configurations, whereas it was underpredicted in the 3:5 configuration. When the rotor blade was scaled such that its chord was shorter than

in the unscaled case, as was the case with the studies conducted by Yao et al. (2002) and Clark et al. (2000), the results of Arnone and Pacciani (1996) agreed with the results of Yao et al. (2002) and Clark et al. (2000). When the scaled rotor blade chord was longer than that in the unscaled configuration, no conclusions can be drawn, with respect to agreement between the results from Arnone and Pacciani (1996), Yao et al. (2002) and Clark et al. (2000), because Yao et al. (2002) and Clark et al. (2000) did not present any results for such cases. Furthermore, Arnone and Pacciani (1996) also found that the lift coefficient amplitude was underpredicted for all configurations at harmonics of the fundamental blade passing frequency, in agreement with the results of Yao et al. (2002) and Clark et al. (2000).

Blade scaling also affects passage and tip vortices, pressure loss, and secondary flow features. The error in pressure loss is linked to the failure of the scaled model to capture secondary flow. Yao et al. (2002) concluded that this is due to a change in the size of the rotor blades.

In conclusion, the results of previous investigations indicate that blade scaling should be kept as closely to 1 as possible, so that the secondary flow features and the unsteady pressure fluctuations closely agree with results in the unscaled cases. Modifying blade size excessively, in order to reduce computational time, could lead to inaccurate simulation results.

Chapter 3

Fluid Mechanical Background

This chapter introduces some basic concepts of flows in gas turbines, definitions of losses, and the domain scaling method. A thorough understanding of these topics was necessary in order to analyze the simulation results. Therefore, the goal of this chapter is to establish a firm basis on which the information in the following chapters can be communicated effectively.

3.1 Flows in Gas Turbines

Flows in gas turbines are extremely complex due to their three-dimensionality, compressibility effects resulting from high flow velocity, and unsteady stator-rotor interactions. Three-dimensionality in the flow field is a result of many factors, including the following ones (Lakshminarayana (1996)).

- Radial pressure gradients.
- Radial variations in blade shape (blade twist).

- Compressibility effects.
- Blade camber.
- Variation in annulus area.
- Rotor rotation.
- Tip and axial gaps.
- Non-uniformities in the inlet velocity and temperature profiles.
- Viscous effects in wall regions.

Much of the three-dimensionality of the flow is associated with inviscid mechanisms. However, secondary flows are, primarily, the result of viscous effects. Individual phenomena listed in the previous list interact with each other in many ways to generate even higher complexity. Many separation zones, vortices, 3-dimensional boundary layers, wakes, and secondary flows are present in gas turbine flows. The major flow features can be categorized as *tip leakage flow*, *shock waves*, *wakes*, *secondary flows* and *hub passage vortices*.

3.1.1 Tip Leakage Flow

Tip leakage flow occurs when a gap exists between the blade tip and the casing or hub. In rotors, a gap exists between the rotor blade tip and the casing. In stators, a gap sometimes exists between the stator vane tip and the hub. The addition of secondary flows and hub passage vortices complicate the tip leakage flow problem.

The driving force behind tip leakage flow is the pressure difference between the suction side and the pressure side of the blade or vane. This pressure difference forces fluid to move from the pressure side of the blade to the suction side of the blade.

Due to the fact that tip leakage depends on the pressure on the suction and pressure sides of the blade, blade loading is a major factor in determining the amount of fluid involved in tip leakage flows. The amount of tip leakage flow also depends on the gap height. If the tip gap were sufficiently narrow, viscous effects would limit the fluid flow through it. In many cases, the tip gap needs to remain large enough to prevent the rotor tip from contacting the casing when the rotor undergoes expansion due to high operating temperatures. Consequently, the tip leakage flow rate may be substantial and it increases as the gap widens.

In many cases, the leakage flow will roll up and form a vortex on the suction side of the blade. However, the tip vortex may not form if the turbulence is very high, if the mean flow speed is very high or if a corner separation zone, on the blade suction side, diffuses the tip leakage flow before it forms a vortex (Lakshminarayana (1996)).

Secondary flows and leakage flows tend to have opposite directions. Leakage flows are beneficial to gas turbines performance when there is a corner separation region because the leakage flow pushes the separation region into the main flow field and eliminates it.

Rotor rotation direction has a significant impact on the tip vortex. In a compressor, the rotation assists the tip leakage flow and therefore the tip leakage vortex moves closer to the centre of the blade passage. In contrast, in a turbine, the tip leakage flow moves in the same direction as the rotor rotation, and the tip leakage vortex remains close to the suction

side of the blade.

In many gas turbine flows, the tip leakage flow is more important than the secondary flows. The most significant effects of tip leakage flows and tip vortices are the following (Lakshminarayana (1996)).

1. They produce a three-dimensional flow field, which can spread away from the tip region, by a distance of approximately 10-30% of the blade span.
2. Tip leakage flow and the associated vortices produce losses, which reduce the device's efficiency.
3. The unloading of the blade tip, due to the leakage flow, reduces the pressure change through the rotor and reduces the angle through which the flow turns.
4. They cause unsteadiness, which can lead to higher stresses on downstream blades, vibration, flutter, and noise production.
5. They introduce complications to the heat transfer process.

3.1.2 Shock Waves

In modern high pressure gas turbine flows (shock waves do not exist in low pressure turbines), shock waves can occur in the turbine passage. However, they are typically oblique shocks and as a result produce little energy loss. The shock system of greatest importance is the trailing edge shock system (Denton (1993)). A small recirculation zone is formed directly behind the rounded trailing edge. The flow, from the pressure and suction sides of the rotor blades, undergoes expansion as the flow travels around the trailing edge and

into this region. In addition, at the point where the suction-side flow and the pressure-side flow meet, a strong shock wave is formed. The total energy loss due to shocks accounts for approximately 18% of the total loss in two-dimensional flows in a high pressure gas turbine (Payne (2001)).

The interaction between shock waves and boundary layers is of importance as well. In the interaction zone, a separation bubble is likely to occur (Denton (1993)). As a consequence of this, additional energy losses occur, due to increased dissipation in the separation bubble and downstream of the bubble. In addition, laminar flow will likely undergo transition as it passes through the separation bubble and strong shocks may cause “total” separation of the boundary layer (Denton (1993)).

3.1.3 Wakes

The loss in this region has been found to depend on the shape of the trailing edge (Payne (2001)). Elliptical trailing edges reduce the loss with respect to square or circular trailing edges. It has been determined, for two-dimensional flows, that approximately 15 to 22 % of the total loss is due to the wake behind a rotor blade with a trailing edge of zero thickness (Payne (2001)). The addition of a thick trailing edge increases this loss by approximately 12% up to a total wake loss of 27% to 34% (Payne (2001)).

3.1.4 Secondary Flows

Secondary flows are the result of viscous effects in the presence of velocity, pressure and temperature gradients introduced by boundary layers on the walls, upstream rotors and stators, or the combustion chamber. Lakshminarayana (1996) gives a simplified description

of the generation of secondary flow by viscous effects in a boundary layer for a steady incompressible 2-D cascade. Figure 3.1 shows the simplified blade channel used in this explanation. The line AAA is a streamline in the inviscid region and line BBB is another streamline, directly below AAA , in the boundary layer region. For simplicity, we will neglect velocity gradients and viscous effects in the n direction. If the flow is incompressible and steady (no rotation), the pressure gradient in the normal direction n at a point along streamline AAA will be proportional to the centripetal acceleration, as

$$\left(\frac{\partial P}{\partial n}\right)_A = \rho \frac{u_A^2}{R_A} \quad (3.1)$$

where R is the local radius of curvature. If we assume that the pressure is constant across the boundary layer, then the normal pressure gradient would have the same value for streamlines AAA and BBB , so that

$$\left(\frac{\partial P}{\partial n}\right)_A = \left(\frac{\partial P}{\partial n}\right)_B = \rho \frac{u_A^2}{R_A} > \rho \frac{u_B^2}{R_B} \quad (3.2)$$

However, because $u_B < u_A$ and $R_B = R_A$, the normal pressure gradient along streamline BBB would not be proportional to the centripetal force that a fluid particle would experience moving with a velocity u_B at a radius of R_B . As a result, the streamline BBB would be deflected to $BB'B''$, to permit the normal pressure gradient force to balance the centripetal force. This results in a cross flow v , which is known as secondary flow.

An effect of the secondary flow is the generation of vorticity. In incompressible flow, the continuity equation is

$$\frac{\partial u}{\partial s} + \frac{\partial v}{\partial n} + \frac{\partial w}{\partial b} = 0 \quad (3.3)$$

In steady flow, along the mean line in a curved duct (neglecting the streamwise pressure

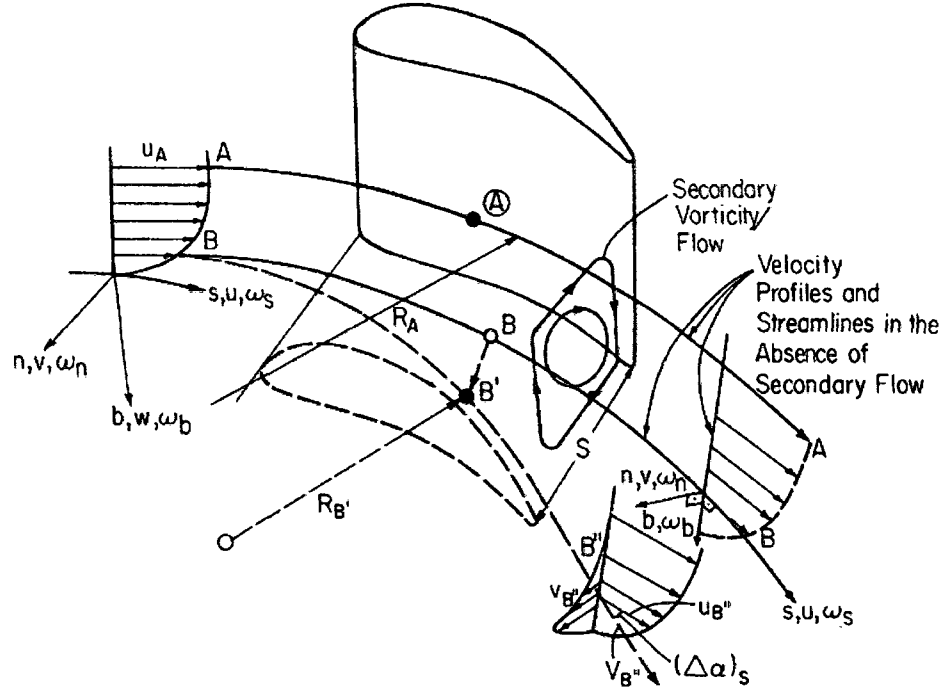


Figure 3.1: Schematic illustration of the development of secondary flow in a rotor blade passage (Lakshminarayana (1996)).

gradient in the s direction) $\partial u / \partial s = 0$ (Lakshminarayana (1996))

$$\frac{\partial v}{\partial n} = -\frac{\partial w}{\partial b} \quad (3.4)$$

$$\omega_s = -\frac{\partial v}{\partial b} + \frac{\partial w}{\partial n} \quad (3.5)$$

This relation shows how the viscous effects in the boundary layer give rise to additional velocity components (v and w), which lead to secondary flow and streamwise vorticity ω_s (equation 3.5) in a gas turbine blade passage.

It is clear that a secondary flow would develop when a flow with a velocity gradient (e.g., a boundary layer or a vortex) moves through a curved path, such as a blade passage. Secondary flow can also occur when viscous flow interacts with the leading edge of a blade or

vane or when a radial temperature gradient at the inlet exists. This can be demonstrated by using a generalized analysis of the vorticity equations (Lakshminarayana (1996)), although the detailed analysis was omitted here. The generalized equations for vorticity give the following expression modeling compressible flow, with some simplifying assumptions, for the vorticity in the streamwise direction s

$$\frac{\partial}{\partial s} \left(\frac{\omega_s}{\rho u} \right) = \frac{2\omega_n}{\rho UR} + \frac{1}{\rho^2 R} \frac{\partial \rho}{\partial b} = \frac{2\omega_n}{\rho UR} - \frac{1}{\rho RT} \frac{\partial T}{\partial b} \quad (3.6)$$

where ρ is density, ω_n is the vorticity in the n direction and T is temperature. The first term on the right hand side shows the contribution of the velocity gradient in the spanwise direction b to the streamwise vorticity, as discussed previously. The second term shows the effect of a spanwise temperature gradient on the streamwise vorticity. This was not observed in equation 3.5. The above equation can be simplified to

$$\frac{\partial}{\partial s} \left(\frac{\omega_s}{\rho U} \right) = \frac{2}{\rho \rho_\infty U^2 R} \frac{\partial P_0}{\partial b} \quad (3.7)$$

Equations 3.6 and 3.7 show that it is possible to manipulate the spanwise temperature gradient and the inlet spanwise velocity gradient, such that the two terms on the right hand side of equation 3.6 cancel each other out. This leads to a reduced spanwise stagnation pressure gradient, which reduces the streamwise vorticity and secondary flow, as indicated by equation 3.7.

In addition to the previous effects, secondary flows can be also generated by rotation of the rotor and compressibility effects. However, in axial gas turbines the rotation of the rotor has negligible effect on secondary flows. In contrast, compressibility effects change the inlet vorticity ω_n and the temperature gradient $\partial T/\partial b$, which in turn affect the secondary flow field significantly.

In general, secondary flows have several effects on gas turbine flows, including the following.

1. Generation of cross flow, which results in a three-dimensional flow field.
2. An effect on the flow turning, which affects the pressure change through gas turbines.
3. Secondary flow losses, which results in approximately a 2-4% reduction in efficiency.
4. The generation of a vortex by the cross flow, which may cause flow separation in the corner region where the end wall and the blade suction surface meet.
5. An effect on the flow conditions through downstream rotors and stators, which results in unsteadiness possibly leading to flutter, vibration and noise.
6. Distortion of the temperature field.

3.1.5 The Hub Passage Vortex

The hub passage vortex is generated from the pressure-side leg of the horseshoe vortex that develops due to the approach of the incoming boundary layer upon the junction between the blade or vane tip and the endwall. This is shown in Figure 3.2, which illustrates the secondary flow model of Takeishi et al. (1990). While there are many secondary flow models, many do not agree on the existence of the smaller corner vortices (Goldstein and Spores (1988)) and the path that the suction leg of the horseshoe vortex follows. However, all investigators agree that the passage vortex (i.e., the pressure-side leg of the horseshoe vortex) is the most dominant and strongest feature of this flow, considering that no tip vortex is generated there due to the absence of a gap. Moreover, the pressure-side leg of

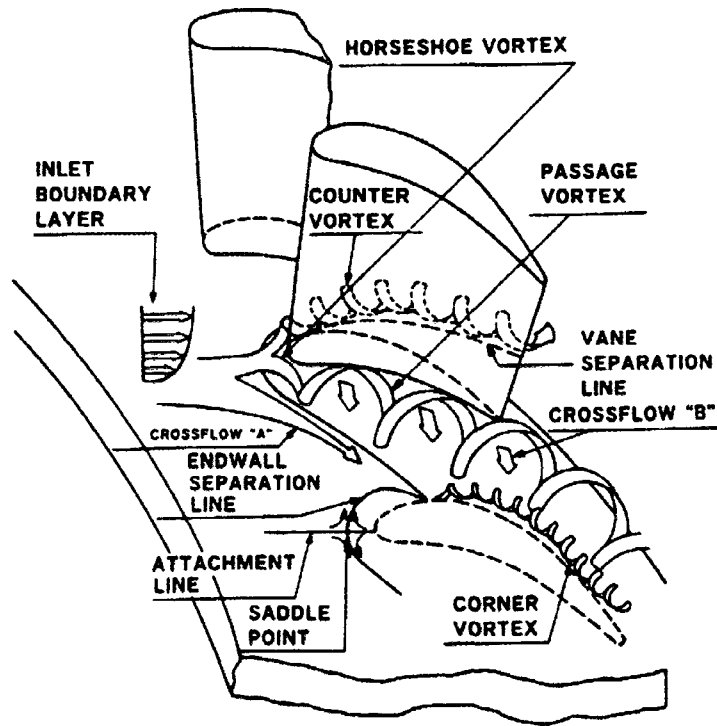


Figure 3.2: Passage vortex model of Takeishi et al. (1990).

the horseshoe vortex is much stronger than its suction-side leg due to the amplification of the former by the crossflow component near the endwall, whereas crossflow weakens the latter. As seen in Figure 3.2, the passage vortex travels across the blade passage towards the suction side of the adjacent blade. The suction-side leg of the horseshoe vortex (counter vortex in Figure 3.2) is thought to either orbit around the passage vortex (Sharma and Butler (1987)) or travel outwards, in the spanwise direction, towards the casing remaining above (Goldstein and Spores (1988); Takeishi et al. (1990)) or below the passage vortex (Langston (1980)).

The horseshoe vortex develops due to the incoming boundary layer coupled with

a spanwise increase in pressure along the leading edge of the vane or blade as the radius increases. This spanwise pressure gradient drives the fluid, at the leading edge, down into the boundary layer and forms the horseshoe vortex. The strength and structure of the horseshoe vortex have been found to depend on the leading edge radius of the blade or vane. If the leading edge radius is greater than the boundary layer thickness, the horseshoe vortex is formed as a large single vortex, as shown in Figure 3.3a, whereas, if the leading edge radius is smaller than the boundary layer thickness, then the horseshoe vortex consists of a combination of smaller vortex structures, as shown in Figure 3.3b (Xun et al. (2005)). The single large vortex is stronger than the combination of smaller vortices. Therefore, a large leading edge radius on a blade or vane will lead to a single strong horseshoe vortex and high losses. If the leading edge radius is reduced sufficiently, the horseshoe vortex may be close enough to the endwall surface to be diffused by viscous stress in the boundary layer and become eliminated (Bradshaw (1987)), reducing the losses. Roulund et al. (2005) found that a horseshoe vortex may not form around a cylindrical pile if the boundary layer thickness is less than 1% of the diameter of the pile. Therefore, accelerating the flow through a subsonic nozzle may reduce the horseshoe vortex by thinning the boundary layer. One must recognize, however, that the leading edge radius is usually maintained sufficiently large in order to reduce sensitivity to incidence and improve heat transfer to increase blade cooling.

It is generally agreed upon that the passage vortex is a dominant structure in turbomachine flows. In rotating turbine blade rows, the passage vortex strength is increased (Payne (2001)). The fact that losses in a turbine blade row are larger than in a

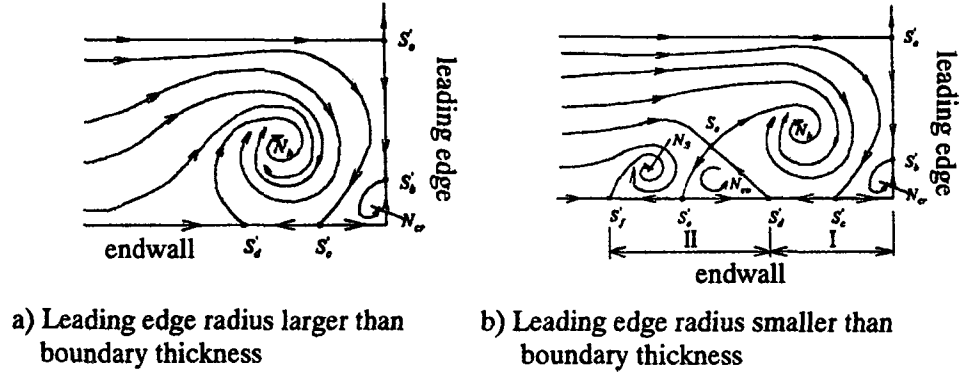


Figure 3.3: Horseshoe vortex topology (Xun et al. (2005)).

stationary blade row, demonstrates that the passage vortex contributes significantly to the turbomachine losses.

3.2 Quantification of Losses

In practice, the prediction of losses is usually accomplished through the use of empirical correlations. The present study endeavours to predict the losses directly from the numerical solutions and without the use of correlations. The losses typically represented by the *total pressure loss coefficient* (Schobeiri (2005))

$$Y_p = \frac{P_{01} - P_{03}}{P_{03} - P_3} \quad (3.8)$$

where the subscripts 01, 03, and 3 indicate the stagnation pressure at state 1, the stagnation pressure at state 3 and the static pressure at state 3, respectively. Figure 3.4 shows the state numbers with respect to a turbine stage. State 1 is before the stator, state 2 is between the stator and rotor and state 3 is after the rotor. This parameter is widely used because it is easily determined from experimental data. However, a more appropriate loss coefficient,

for a turbine blade, is (Denton (1993))

$$\zeta_h = \frac{h_3 - h_{3s}}{h_{03} - h_3} \quad (3.9)$$

where h is specific enthalpy and the subscript $3s$ indicates the static property at state 3 after an isentropic process. Although the loss coefficients in equations 3.8 and 3.9 are commonly used and are adequate for preliminary design purposes, one should be cautioned that rotation can cause changes in the relative stagnation pressure and enthalpy, even in the absense of any losses (Denton (1993)).

It has been suggested by Denton (1993) that the estimation of losses and efficiency be accomplished through the use of entropy, because the entropy value is independent of the frame of reference. However, in order to determine the efficiency, the entropy s_1 at a reference state is required. This requires the measurement of many variables, which may not be easy to do experimentally. This is not an issue for a numerical study, which can provide all necessary data for any location in the domain. The specific entropy change can be determined as (Van Wylen and Sonntag (1978))

$$\frac{s_3 - s_1}{\mathbf{R}} = \left(\frac{\gamma}{\gamma - 1} \right) \ln \left(\frac{T_{03}}{T_{01}} \right) - \ln \left(\frac{P_{03}}{P_{01}} \right) \quad (3.10)$$

where $\mathbf{R}$ is the gas constant for air and γ is the ratio of specific heats for air. The *total-to-total (isentropic) efficiency* η_{tt} is defined as the ratio of the actual turbine work output and the isentropic turbine work output

$$\eta_{tt} = \frac{h_{01} - h_{03}}{h_{01} - h_{03s}} \quad (3.11)$$

$$\eta_{tt} = \frac{T_{01} - T_{03}}{T_{01} \left[1 - \left(\frac{P_{03}}{P_{01}} \right)^{\frac{\gamma-1}{\gamma}} \right]} \quad \text{if } \mathbf{C_p} \text{ is constant} \quad (3.12)$$

where the subscript 03s indicates the stagnation property at state 3 after an isentropic process. This is true if no heat is transferred to the surroundings. In many cases, the kinetic energy can be neglected and the isentropic efficiency can be simplified to

$$\eta_{tt} \simeq \frac{h_1 - h_3}{h_1 - h_{3s}} \quad (3.13)$$

The isentropic work output can be written in terms of entropy and temperature instead of enthalpy change after an isentropic process, if the temperature is assumed to be constant between states 3s and 3 (Denton (1993))

$$\eta_{tt} \simeq \frac{h_1 - h_3}{h_1 - h_3 + T_3(s_3 - s_1)} \quad (3.14)$$

Denton (1993) states that this assumption is not likely to result in large errors. The *total-to-static efficiency* η_{ts} is defined as the actual turbine work output divided by the isentropic turbine work output and the kinetic energy of the exit flow of an ideal turbine

$$\eta_{ts} = \frac{h_{01} - h_{03}}{(h_{01} - h_{03s}) + \frac{1}{2}U_3^2} = \frac{h_{01} - h_{03}}{h_{01} - h_{3s}} \quad (3.15)$$

$$\eta_{ts} = \frac{T_{01} - T_{03}}{T_{01} \left[1 - \left(\frac{P_3}{P_{01}} \right)^{\frac{\gamma-1}{\gamma}} \right]} \quad \text{if } \mathbf{C_p} \text{ is constant} \quad (3.16)$$

where U is the flow velocity magnitude. A comparison of the two efficiencies shows that the total-to-static efficiency treats the kinetic energy contained in the exit as a loss, whereas the total-to-total efficiency does not do so. As a result, the total-to-total efficiency equation is a more accurate representation of the efficiency of multistage gas turbines and the total-to-static efficiency is a better representation of the efficiency of single stage turbines or of the last stage of a multi stage gas turbine (Lakshminarayana (1996)).

The entropy loss coefficient, defined by Denton (1993), is

$$\zeta_s = \frac{T_3(s_3 - s_1)}{h_{03} - h_3} \quad (3.17)$$

$$\zeta_s = \frac{s_3 - s_1}{C_p \left(\frac{T_{03}}{T_3} - 1 \right)} \quad \text{if } C_p \text{ is constant} \quad (3.18)$$

Entropy generation in fluid processes is also usually inversely proportional to the local temperature, as shown by Denton (1993)

$$T\Delta s = \zeta_s \frac{1}{2} U^2 \quad (3.19)$$

In addition, because the loss coefficient is essentially independent of temperature, flows with constant velocity and a constant entropy loss coefficient produce less entropy at higher temperatures than the same fluid process occurring at a lower temperature. Therefore, it is more beneficial for a gas turbine to operate at higher temperatures.

In order to quantify the energy losses separately, they are divided into groups, each of which contains a major loss feature. Denton (1993) has used a long standing method, in which the losses are divided into three groups:

- Profile losses,
- Secondary flow and endwall losses and
- Leakage losses.

Profile losses contain losses due to blade boundary layers and mixing losses near the trailing edge. Secondary flow losses occur in the entire blade passage and endwall losses occur near the hub and casing walls. Secondary flow and endwall losses are difficult to separate from the profile losses and leakage losses because they are generated by many

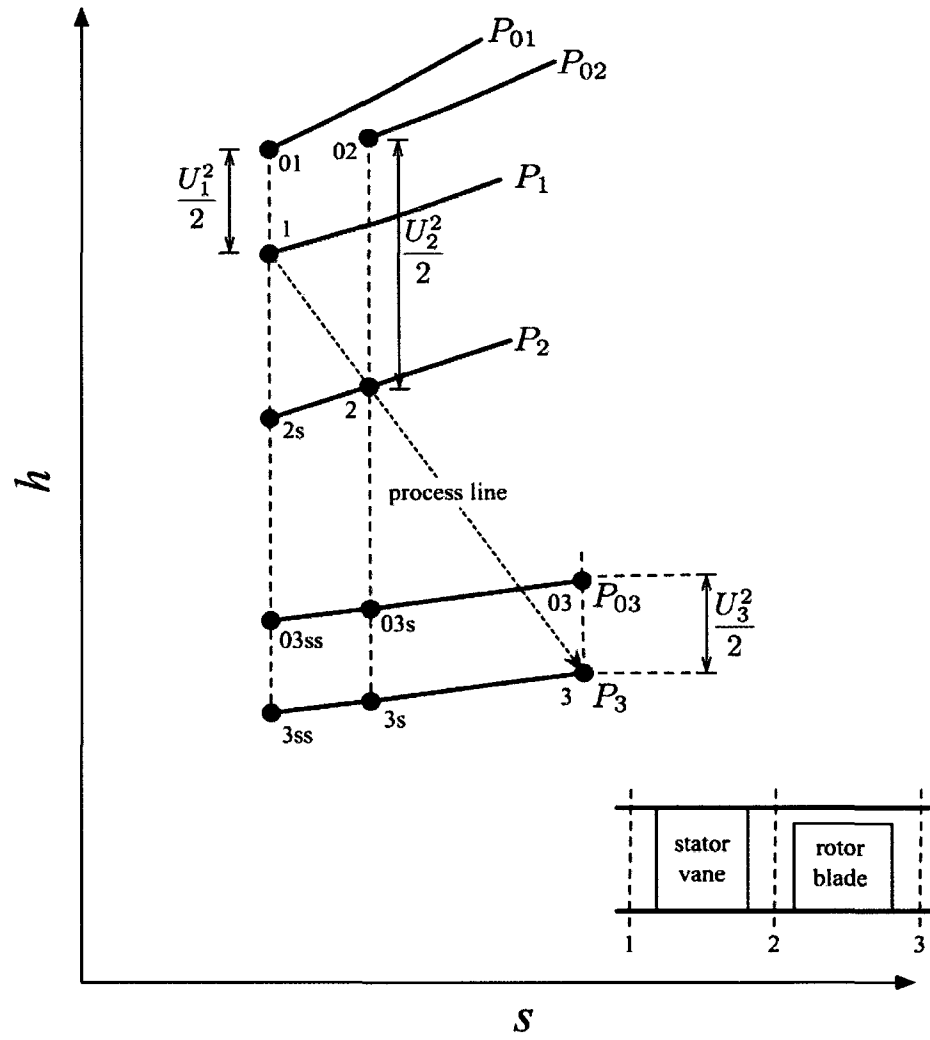


Figure 3.4: Enthalpy-entropy diagram for the expansion process in a single-stage turbine (based on Lakshminarayana (1996)).

different mechanisms. Therefore, the secondary flow and endwall losses category is often used to account for losses not captured by the profile and leakage losses. Leakage losses include losses due to leakage of the flow through the tip gap region between blade or vane tips and endwalls. In many machines, the magnitudes of the three types of losses are approximately equal, so that each type accounts for approximately 1/3 of the total losses (Denton (1993)).

Other methods of loss breakdown have been developed as well. Payne (2001) separated the losses into five categories, instead of three, as follows:

- Profile losses,
- Wake/mixing losses,
- Shock losses,
- Secondary flow and endwall losses and
- Leakage losses.

This method allows a more detailed description of the losses in a gas turbine, however, it is very difficult to determine the magnitude of each loss, due to interactions between different loss types. For example, the interaction between shocks and boundary layers generates additional losses, which can not be classified in any of the previously mentioned groups. However, there have been many loss correlations developed, which attempt to predict the losses associated with the above processes.

Many correlations and theories exist for predicting the losses associated with the profile, wake and shocks. However, when the flow is three-dimensional, the inter-

connectivity between various losses makes it very hard to determine the amount that each loss mechanism contributes to the overall loss. Denton (1993) accomplished this by considering numerical results at a plane after the rotor of interest. This plane was split into three zones:

- The primary flow field,
- Tip leakage flow and
- The remaining secondary flow field.

The primary flow field consists of low entropy fluid. This region is defined as all fluid with a specific entropy value that is lower than a threshold value. The threshold value separates the low entropy fluid (primary flow field) from the high entropy fluid (tip leakage flow and remaining secondary flow). Once the three regions have been determined, the mass-averaged entropy can be computed as (Payne (2001))

$$\frac{\tilde{s}}{\mathbf{R}} = \frac{\int \left(\frac{s}{\mathbf{R}}\right) \rho U dA}{\int \rho U dA} \quad (3.20)$$

This value can then be used to determine the loss coefficients and the efficiency of the stage. This method does not separate all major loss mechanisms. However, this was the best available method which separates the losses due to the three major mechanisms from numerical results, without the use of loss correlations.

3.3 Domain Scaling Method

A common approach in computational fluid dynamics is to take advantage of existing geometric periodicity in order to reduce computer memory, storage space and com-

putational time requirements. In such cases, only one of the repeated sections of the domain is considered and periodic boundary conditions are applied at its boundaries. In the case of turbine stages, which consist of successive rows of stator vanes and rotor blades, geometric periodicity would require that the number of rotor blades be equal to an integral multiple of the number of stator vanes. In practice, however, this is not always the case, which means that accurate simulation of turbine stages would require consideration of the entire stator and rotor. To circumvent this problem, gas turbine analysts use several different methods, as indicated by (Mårtensson et al. (2003)), including rotor blade scaling, phase lagged periodic boundary conditions, and time inclined boundary conditions. This report will focus on the rotor blade scaling technique in view of the fact that a commercial code will be used to run the numerical simulation. The other methods require extensive modification to the underlying code, used for CFD, and therefore, are not practical for use with commercial CFD programs.

The following method was proposed by Rai and Madavan (1990) and Rai (1987) to reduce the computational time required for the numerical simulation of flow in a turbine consisting of 22 stator vanes and 28 rotor blades.

The ratio of the numbers of rotor blades and stator vanes in an actual turbine stage is defined as

$$\Lambda = \frac{N_{rt}}{N_{st}} \quad (3.21)$$

where N_{rt} and N_{st} are the numbers of rotor blades and stator vanes, respectively. The objective of rotor blade scaling is to simplify the computational geometry by changing the

number of blades such that it is equal to the number of vanes or such that the blade-to-vane ratio Λ_{sc} in the scaled geometry is equal to the ratio of two small integers (e.g., $1/3$, $2/3$, $3/4$ etc.). During scaling, it is desirable to maintain the airfoil shape intact as well as to preserve the power generated by the original rotor. This requires that the rotor blade pitch-to-chord ratio (solidity ratio) be kept constant, which is achieved by scaling the rotor blade by the scaling ratio factor

$$q = \frac{\Lambda}{\Lambda_{sc}} \quad (3.22)$$

To do so, the chord length c_{sc} of the scaled rotor blade and the scaled rotor pitch $L_{rp,sc}$ are set as, respectively,

$$c_{sc} = qc \quad (3.23)$$

and

$$L_{rp,sc} = qL_{rp} \quad (3.24)$$

where c is the chord length of the unscaled rotor blade and L_{rp} is the unscaled rotor pitch, as shown in Figure 3.5.

The main idea behind the blade scaling method is to change the number of rotor blades so that a periodic boundary condition can be applied on a sector of the domain in order to reduce the computational resource requirements. However, changing only the pitch would change the number of rotor blades and, therefore, the lift force per blade, thus altering its lift coefficient. Scaling of the rotor blades changes the area of the blade. As a

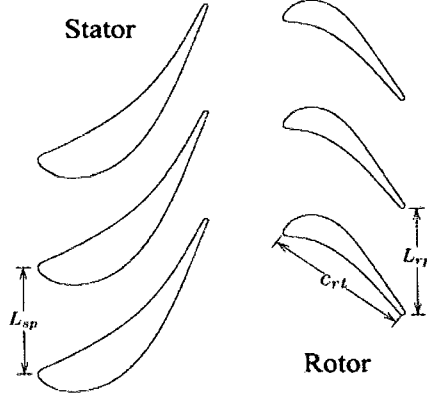


Figure 3.5: Schematic diagram of a single stage turbine.

result, the lift force per blade changes in proportion to the blade area, as

$$L = \int_{\text{blade span}} \frac{C_l}{2} \rho_{\infty} U_{\infty}^2 c ds \quad (3.25)$$

where ds represents an infinitesimally small portion of the span of the rotor blade. Moreover, as will be shown in the following, the total torque generated by the rotor remains constant if the blade scaling method is used.

While it was stated earlier that the lift coefficient changes as the pitch changes, Clark et al. (2000) and ?) both found that the time-mean rotor blade pressure distribution changes very little when the rotor blade is scaled between 1 and 1.14. Therefore, if the blade is scaled by less than 14% while maintaining a constant shape, the coefficient of lift C_l , defined as

$$C_l = \frac{1}{c} \int_0^c (C_{p,suc} - C_{p,prs}) dx \quad (3.26)$$

would remain constant. This condition holds only if it is assumed that the change in the number of blades from N_{rt} in the unscaled rotor to $N_{rt,sc}$ in the scaled rotor does not modify the pressure distribution over the rotor blades. The torques M generated by the unscaled and scaled rotors are, respectively,

$$M = N_{rt} \int_{r_{hub}}^{r_{cas}} \frac{C_l}{2} \rho_{\infty} U_{\infty}^2 c r dr \quad (3.27)$$

and

$$M_{sc} = N_{rt,sc} \int_{r_{hub}}^{r_{cas}} \frac{C_l}{2} \rho_{\infty} U_{\infty}^2 c_{sc} r dr \quad (3.28)$$

where r is the radial distance from the axis of rotation, with r_{hub} and r_{cas} being the radii of the hub and casing, respectively.

To obtain an accurate numerical simulation with a scaled rotor, kinematic similarity, dynamic similarity and torque equality are required. Kinematic similarity is only maintained if the speeds of sound in the scaled and unscaled cases are the same, which may not be the case, especially in the scaling ratio is large. Dynamic similarity in the entire stage cannot be not maintained, because enforcing dynamic similarity of the flow over the rotor would compromise dynamic similarity of the flow over the stator. The current scaling method maintains torque equality. This can be shown by substituting the scaling relations, Equations 3.21, 3.23, and 3.24, into Equation 3.28.

$$M_{sc} = \frac{2\pi}{L_{rp,sc}} \int_{r_{hub}}^{r_{cas}} \frac{C_l}{2} \rho_{\infty} U_{\infty}^2 c_{sc} r dr = \frac{2\pi}{\Lambda L_{rp}} \int_{r_{hub}}^{r_{cas}} \frac{C_l}{2} \rho_{\infty} U_{\infty}^2 \Lambda c r dr =$$

$$N_{rt} \int_{r_{hub}}^{r_{cas}} \frac{C_l}{2} \rho_{\infty} U_{\infty}^2 c r dr = M \quad (3.29)$$

Therefore, the scaling method presented by Rai (1987) and Rai and Madavan (1990) maintains torque similarity between the actual turbine and the model turbine. In addition, if

the rotational speed is kept constant, then the power output from the model turbine would match the power of the actual turbine.

3.4 Coherent Structure Identification

Due to the highly complex and turbulent flow that exists in turbomachines, it is quite difficult to distinguish *coherent structures* (CS) from non-coherent turbulence and mean vorticity patterns (Chang and Tavoularis (2007)). For this study CS were identified using the Q -criterion of Hunt et al. (1988). This method allows one to capture large-scale vortical structures whose core fluid has small, if any, strain. For convenience in writing equations containing vectors and other tensors, we shall adopt the index notation, by which the velocity vector is written as u_i , $i = 1, 2, 3$. In this notation, the Einstein summation notation is adopted, by which an index repeated in the same term of an equation is summed over its range. The rate of strain tensor is defined as

$$S_{ij} = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad (3.30)$$

where $\partial u_i / \partial x_j$ is the velocity gradient tensor at a given point. Similarly, the rotation tensor is defined as

$$\Omega_{ij} = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i} \right) \quad (3.31)$$

The parameter Q is defined as the second invariant of the velocity gradient tensor

$$Q = -\frac{1}{2} \frac{\partial u_i}{\partial x_j} \frac{\partial u_j}{\partial x_i} = \frac{1}{2} (\Omega_{ij} \Omega_{ij} - S_{ij} S_{ij}) \quad (3.32)$$

Positive values of Q indicate regions where vorticity overcomes strain. The advantage of this method is that the value of Q is not affected by wall-generated shear because the latter has equal components of rotation and strain rate. CS have been identified as surfaces on which Q maintains a constant positive value, called the threshold (Chang and Tavoularis (2007)). Applications of this method to the present simulation results identified a large number of CS in the near-wall regions but did not clearly identify the tip-gap vortices or the passage vortices. To resolve this issue, a modified parameter Q , developed by Chang and Tavoularis (2007), was used as criterion; this was defined as

$$Q_m = \frac{1}{2}(\Omega_{ij}\Omega_{ij} - c_q S_{ij}S_{ij}) \quad (3.33)$$

where c_q is an empirical factor of order one that reduces the effect of the rate of strain on the coherent structure identification. By selecting $c_q = 1.3$ and $Q = 1$, the tip-gap vortices and the passage vortices were identified, while the small-scale, near-wall CS were not.

Chapter 4

Numerical Background

This chapter introduces turbulence models and relevant methods of numerical analysis, spatial discretization and simulation verification. These topics are essential in understanding the simulation procedures.

4.1 Governing Equations

Gas turbine flows are highly complex, transient and turbulent and as a result the turbulence must be accounted for accurately. *Direct numerical simulation* (DNS) and *large eddy simulation* (LES) are not feasible to use due to the high mesh density that these approaches require. DNS requires a mesh density that is related to the Reynolds number by $Re^{9/4}$ (Blazek (2005)). Therefore, high-speed gas turbine flow with a Reynolds number of the order 10^6 would require a grid with $10^{13.5}$ nodes. LES requires approximate node count that is given by $Re^{0.4}$ in the outer regions and $Re^{1.8}$ in the viscous sublayer (Blazek (2005)). For the same gas turbine and Reynolds number given above, this yields a mesh

with $10^{13.2}$ nodes. Both of these node counts are far in excess of the available computing power. A less computationally intensive approach is the solution of the *Unsteady Reynolds Averaged Navier Stokes* (URANS) equations with the use of turbulence models.

4.1.1 Reynolds Averaged Navier-Stokes Equations

In this approach, Reynolds averaging is used, namely the flow properties are considered as composed of a mean component, averaged over an ensemble of realizations (denoted by angle brackets) and a fluctuating component (denoted by a prime), as for example

$$u = \langle u \rangle + u' \quad (4.1)$$

Ensemble averages in the present flows, which are periodic on the average, are obtained as phase averages, that is by averaging all values of the corresponding property at the same relative time instant during each cycle. The time-dependent property u is the “resolved” property, provided as the solution of the URANS equations. Furthermore, a time average (to be denoted by an overbar) can be defined over the cycle as

$$\overline{\langle u \rangle} = \frac{1}{\tau_p} \int_0^{\tau_p} \langle u \rangle dt \quad (4.2)$$

where τ_p is the period. In addition to the previously defined fluctuations, the actual flow contains velocity fluctuations, to be denoted by braces, as $\{u''^2\}$ etc., which are not resolved by the solution of the URANS equations. Instead, the use of a specific turbulence model in the URANS equations provides estimated statistical properties of these unresolved velocity fluctuations, such as the time-dependent turbulent kinetic energy k and Reynolds stresses. One may represent symbolically this as

$$k = \frac{1}{2} [\{u''^2\} + \{v''^2\} + \{w''^2\}] \quad (4.3)$$

The total kinetic energy of all velocity fluctuations at any given phase in the cycle would be

$$\langle k_{tot} \rangle = \langle k \rangle + \frac{1}{2} [\langle u'^2 \rangle + \langle v'^2 \rangle + \langle w'^2 \rangle] \quad (4.4)$$

The conservation of momentum equation, solved by FLUENT 6.3.26, can be written as (Ansys (2006))

$$\frac{\partial}{\partial t} (\rho u_i) + \frac{\partial}{\partial x_j} (\rho u_i u_j) = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial u_k}{\partial x_k} \right) \right] + \frac{\partial}{\partial x_j} (-\rho \{u_i'' u_j''\}) \quad (4.5)$$

The term $-\rho \{u_i'' u_j''\}$ indicates the Reynolds stresses which must be solved for using turbulence models. The term δ_{ij} is the Krönecker delta, equal to 1 when $i = j$ and 0 when $i \neq j$.

The conservation of energy equation, in index notation, is (Ansys (2006))

$$\frac{\partial}{\partial t} (\rho e) + \frac{\partial}{\partial x_j} (u_j [\rho e + P]) = \frac{\partial}{\partial x_j} \left[\kappa_{eff} \frac{\partial T}{\partial x_j} \right] + \frac{\partial}{\partial x_j} \left[u_i \mu_{eff} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial u_k}{\partial x_k} \right) \right] \quad (4.6)$$

The specific internal energy e , the effective thermal conductivity κ_{eff} and the effective dynamic viscosity μ_{eff} are defined as

$$e = h - \frac{P}{\rho} + \frac{U^2}{2} \quad (4.7)$$

$$\kappa_{eff} = \kappa + \kappa_t \quad (4.8)$$

$$\mu_{eff} = \mu + \mu_t \quad (4.9)$$

where h , κ , κ_t , μ and μ_t are the specific enthalpy, the fluid thermal conductivity, the

turbulent thermal conductivity, the fluid dynamic viscosity and the turbulent dynamic viscosity, respectively. The velocity magnitude is

$$U = (u_i u_i)^{\frac{1}{2}} \quad (4.10)$$

The first term in Equation 4.6 represents the change in energy over time, while the second term represents the convection of energy by the flow. The two terms on the right hand side of equation 4.6 represent molecular diffusion of heat and viscous heating, respectively. The viscous heating term models the thermal energy created by the viscous shear and is always included when the coupled solver is used. In addition, viscous heating becomes important in high Mach number flows (Clark et al. (2000)).

The equation expressing conservation of mass for compressible flow is (Ansys (2006))

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i) = 0 \quad (4.11)$$

4.1.2 Favre Averaged Navier-Stokes Equations

Considering that the flows in the devices of interest are compressible, the density based solver in the utilized commercial CFD code FLUENT 6.3.26 will be implemented. This, in conjunction with the requirement to model turbulence and unsteadiness, requires the use of the *Unsteady Favre Averaged Navier Stokes* (UFANS) equations. The UFANS equations are derived from the Navier Stokes equations by using Favre averaging, namely

by using the density-weighted mean velocity (Hinze (1975)), defined as

$$\tilde{u} = \frac{\langle \rho u \rangle}{\langle \rho \rangle} \quad (4.12)$$

$$\rho = \langle \rho \rangle + \rho' \quad (4.13)$$

In the UFANS equations, Reynolds averaging is done for density and pressure, while Favre averaging is done for velocity, internal energy, enthalpy and temperature.

After Favre averaging the conservation of momentum is

$$\frac{\partial}{\partial t} (\rho \tilde{u}_i) + \frac{\partial}{\partial x_j} (\rho \tilde{u}_i \tilde{u}_j) = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial \tilde{u}_k}{\partial x_k} \right) \right] + \frac{\partial}{\partial x_j} (-\rho \{u_i'' u_j''\}) \quad (4.14)$$

However, in the case of FLUENT 6.3.26, it does not appear that Favre averaging is completed. Instead, for variable density flow, Equations 4.5, 4.6 and 4.11 are “interpreted as Favre-averaged Navier-Stokes equations, with the velocities representing mass-averaged values”. Therefore, the actual process of mass averaging is not done. The flow properties are simply calculated as with incompressible flows, but with the variable density determined from the perfect gas law, and the results simply represent mass averaged values. Therefore, in FLUENT 6.3.26 the same forms of the mass, momentum and energy equations are used for both compressible and incompressible flows, although the density is variable in the compressible flow case.

4.1.3 Modelling Rotation

FLUENT 6.3.26 has several methods to model flows with rotating components. The model under study contains a stationary component and a rotating component. This

problem can be handled using the mixing plane approach, the multiple reference frame approach, or with a sliding mesh model. However, as the interactions between the rotor blades and stator vanes are expected to be strong, the multiple reference frame approach is not recommended (Ansys (2006)). Therefore, only the mixing plane approach and the sliding mesh model will be used.

The conservation equations differ slightly in a rotating domain, than in a stationary domain. Figure 4.1 shows a schematic diagram of a rotating frame with respect to a stationary frame. If the rotating frame is rotating around the rotation axis with an angular velocity ω_i , then the relative velocity $u_{rel,i}$ (the velocity with respect to the rotational frame) is given by

$$u_{rel,i} = u_i - u_{spin,i} \quad (4.15)$$

where u_i is the absolute velocity (the velocity with respect to the stationary frame) and $u_{spin,i} = \epsilon_{ijk}\omega_j r_k$ is the spinning velocity (the velocity due to the motion of the rotating frame; ϵ_{ijk} is the Levi-Civita tensor; which is 0 if any two indices are the same, +1 if i, j, k have an even permutation of 1, 2, 3, or -1 if i, j, k have an odd permutation of 1, 2, 3).

If the density based solver is used for the simulations, FLUENT 6.3.26 modifies the momentum equations using the absolute velocity, rather than the relative velocity. If the pressure based solver is used, then the user has a choice between the absolute velocity formulation or the relative velocity formulation of the momentum equations.

Mixing Plane Model

The mixing plane model is the preferred choice for gas turbine simulations when computational resources are limited. This model does not require the interface to be iden-

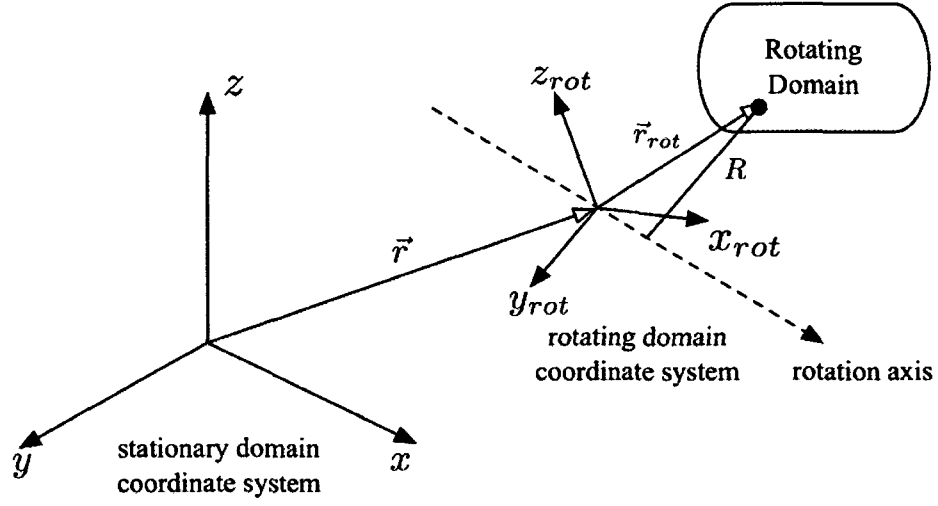


Figure 4.1: Schematic diagram of a rotating domain (based on Ansys (2006)).

tical on both sides, only one rotor blade and one stator vane from each stage need to be modelled. This greatly reduces the computational domain. In addition, the mixing plane model employs a steady solver, therefore, reducing the computational time required to obtain a solution. However, the mixing plane model cannot model unsteady effects, which makes it impossible to determine unsteady interaction effects and unsteady blade loading, as required for flutter predictions.

The mixing plane model artificially mixes the flow at the interface regions by circumferential averaging of the flow variables. The flow variables, passed from the upstream domain to the downstream domain, are the total pressure, flow direction cosines, total temperature and turbulence properties. The flow variables, passed from the downstream domain to the upstream domain, are static pressure and the flow direction cosines.

Sliding Mesh Model

A sliding mesh was implemented to allow the rotor domain to rotate while the stator domain remained stationary. As a result, unsteady simulations could be run and the rotor-stator interactions could be analyzed with minimal assumptions.

Two interface planes are required to employ the sliding mesh model. In addition, both planes have to be geometrically identical, within a certain tolerance. However, the meshes on each of the faces need not be identical or conformal (if two meshes are conformal, then they are identical in all aspects, including size, shape, node locations, and connectivity between nodes). FLUENT 6.3.26 is capable of dealing with non-conformal interfaces when passing information from one domain to the next. An example of a sliding mesh interface is shown in Figure 4.2. When the sliding mesh model is used, an interior zone (a new zone with fluid cells in both domains) and periodic zones are generated. In this example, a sliding mesh interface would consist of an interior zone (d-b-e-c), where the two domains overlap, and a periodic zone pair (a-d and c-f), where the domains do not overlap. The fluxes of flow properties are calculated using the faces that result from the intersection of domain cell faces, such as d-b, rather than the actual cell zone face, such as A-B. Therefore, in Figure 4.2, the flux into cell 4 would be determined using information from cells 1 and 3. This information would be passed through faces d-b and b-e, in the interior zone, in the same manner as if the problem were steady. Faces a-d and c-f are used to form a periodic pair of faces and information is transferred in a similar manner as with the interior zone.

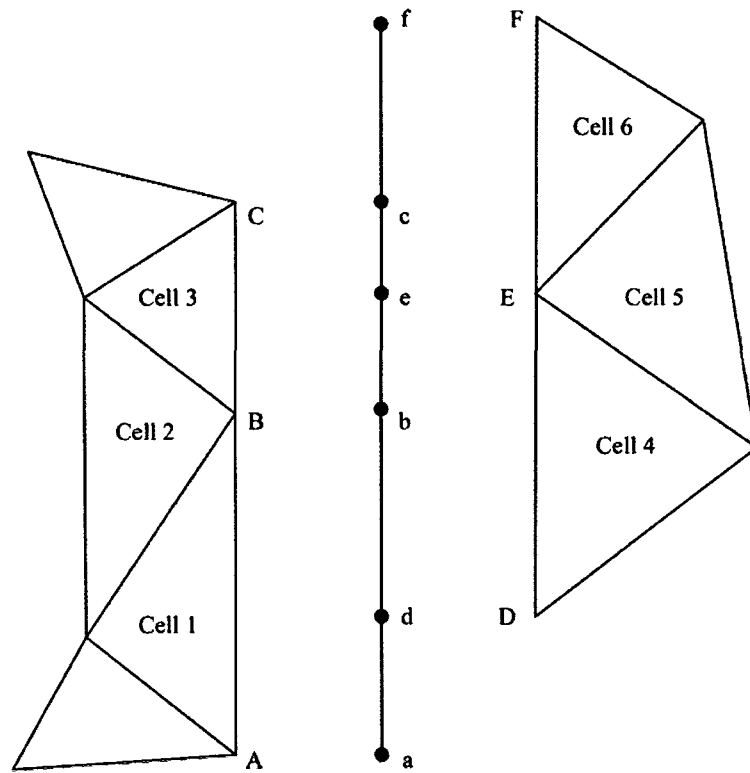


Figure 4.2: Schematic of a sliding mesh interface (based on Ansys (2006)).

4.2 Turbulence Modelling

Reynolds or *Favre* averaging introduces additional unknowns, *Reynolds stresses*, which may be treated as additional stresses that arise from the randomness of the flow. Solution of the *Reynolds Averaged Navier Stokes* (RANS) equations or *Favre Averaged Navier Stokes* (FANS) equations requires the use of turbulence models for the Reynolds stresses. Such models have been introduced in the forms of algebraic or differential equations. No available RANS/FANS model can be used to solve all types of turbulent flows, because, in order to get the optimized prediction of the flow field, there is a need to adjust the empirical coefficients in the model by direct experimental input or by trial-and-error adjustment (Bradshaw (1997)). Turbulence models have found successful applications in engineering design and analysis, even though there is no universal RANS/FANS model for all applications. Compared to the alternative approaches of LES (Large Eddy Simulations) and DNS (Direct Numerical Simulations), RANS/FANS simulations have the advantages of lower computing time and computer resource requirements.

RANS/FANS models have been developed to incorporate non-local and flow history effects in the eddy viscosity. The simplest turbulence models introduce a single additional relationship for the turbulent kinetic energy per unit mass k . Depending on how they achieve the closure of the turbulent kinetic energy equation, turbulence models can be classified into two categories. The first category includes the turbulent viscosity models, which employ the gradient transport assumption, proposed by Boussinesq in 1877

$$-\rho \{u_i'' u_j''\} = \mu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \left(\rho k + \mu_t \frac{\partial u_i}{\partial x_i} \right) \delta_{ij} \quad (4.16)$$

This category is further subdivided into *one-equation* and *two-equation* models,

The second category includes the Reynolds stress models (RSM), which determine the turbulent shear stresses by solving transport equations for each stress component: 7 equations in a two-dimensional isothermal flow, and 15 equations in a three-dimensional non-isothermal flow.

Currently, FLUENT 6.3.26 has ten available turbulence models, among which five have been selected for testing and comparisons. These include the *Spalart-Allmaras* model (S-A), the *RNG (Renormalization group) k - ϵ* turbulence model, the *realizable k - ϵ* turbulence model, the *shear stress turbulence* (SST) model and the *Reynolds stress model* (RSM).

4.2.1 Spalart-Allmaras (S-A) model

One-equation models have shown limited capabilities in their predictive ability because the turbulent length scale, which is strongly dependent on the flow configuration and not universal in three-dimensional turbulent flows, is determined empirically (Speziale (1996)). The one-equation model of Spalart and Allmaras (1992), which determines the turbulent viscosity directly from a single transport equation, has shown better performance on airfoil and wing applications than the standard one-equation model (Blazek (2005)), because the former does not require a length scale as an input like the latter. However, the S-A model is unable to predict accurately the spreading rates of plane and round jets (Bardina et al. (1997)).

4.2.2 Standard $k - \varepsilon$ model

The standard $k - \varepsilon$ model requires two partial differential equations, one for the turbulence kinetic energy and another for the turbulence dissipation rate ε , to determine the turbulence viscosity, which is then used with Boussinesq's relation to determine the Reynolds stresses. The two-equation turbulence models have been the most commonly used in industrial applications. The most popular one is the standard $k - \varepsilon$ turbulence model (Launder and Spalding (1974)). This model is widely used in industrial processes due to its numerical stability, simplicity and overall reliability of the solutions (Menter (1991)). It is also well known that the standard $k - \varepsilon$ turbulence model has shown low accuracy for the simulation of boundary layers in adverse pressure gradients (Bradshaw (1997)). The standard $k - \varepsilon$ turbulence model also has difficulty in integrating through the viscous sublayer and requires a viscous correction to reproduce the law of the wall for incompressible flat-plate boundary layers (Wilcox (2000)).

4.2.3 RNG $k - \varepsilon$ model

The performance of standard two-equation turbulence models is fairly good at high Reynolds number flows, in which the turbulence is nearly homogeneous and the turbulence production is nearly balanced by dissipation. At relatively low Reynolds numbers, however, the difference between turbulent kinetic energy production and dissipation rates may depart from their equilibrium value of zero, thus the ad hoc adjustment of each empirical coefficient and the turbulent Prandtl number in the standard turbulence model is inevitable. The RNG $k - \varepsilon$ turbulence model was introduced to avoid that adjustment by implementing additional

equations based on renormalization group theory, which allow one to analyze changes in turbulent flow as seen from various length scales (Yakhot and Orszag (1986)). For this reason, the RNG $k - \varepsilon$ turbulence model has shown good performance in highly strained flows and swirling flows.

4.2.4 Realizable $k - \varepsilon$ model

The term realizable indicates that this turbulence model partially accounts for the physics of turbulent flow by following constraints placed on the Reynolds stresses (Shih et al. (1994)). The realizable $k - \varepsilon$ model determines the turbulent viscosity with a new function and the dissipation rate is determined by a transport function which is derived from the exact equation for the mean-squared vorticity fluctuation (Shih et al. (1994)). This results in a turbulence model which more accurately determines the flow field in flows with rotation, boundary layers in adverse pressure gradients, and separation and recirculation zones (Shih et al. (1994)).

4.2.5 Shear Stress Turbulence (SST) model

The second most widely used two-equation model is the $k - \omega$ model, where $\omega \propto \varepsilon/k$ is a characteristic frequency of the large eddies (Wilcox (2000)). A transport equation for ω is solved and the turbulent viscosity is then determined as a function of turbulent kinetic energy and dissipation rate. The advantages of the $k - \omega$ model, developed by Wilcox (2000), include numerical stability, simplicity and more accurate results in the viscous sublayer. The simplicity of the model is based on the fact that it requires no damping function (Menter (1994)). Dirichlet boundary conditions can then be used at the wall, in place of the damping

functions. In addition, it has been shown that, in the logarithmic region, the $k - \omega$ model is superior to the $k - \varepsilon$ model when adverse pressure gradients exist and when the flow is compressible (Menter (1992), Menter (1994)). Therefore, the $k - \omega$ model is used in the viscous sublayer and logarithmic regions of the boundary layer. However, the $k - \omega$ model has a strong dependence on the free stream value of ω . The eddy viscosity in the boundary layer has been found to change by more than 100% if one reduces the free stream value of ω (Menter (1991)).

Many two-equation turbulence models suffer from an inability to correctly model flows with an adverse pressure gradient or a recirculation zone. The goal of the SST model, developed by Menter (1994), is to eliminate this problem. It does this by blending the $k - \omega$ model, applied for near-wall region, and the high Reynolds number $k - \varepsilon$ model, applied for core flow region, then modelling the transport of turbulent shear stress in a similar manner to the Johnson-King (JK) turbulence model (Menter (1994)).

Previous two-equation models, which do not account for turbulent shear stress transport, determine the turbulent shear stress τ_t by (Menter (1994))

$$\tau_t = \rho \sqrt{\frac{k_{prod}}{k_{diss}}} a_1 k \quad (4.17)$$

where a_1 is a constant. It has been shown that the ratio of production k_{prod} and dissipation k_{diss} of turbulent kinetic energy can be much greater than 1 in flows with adverse pressure gradients (Menter (1992), Blazek (2005)). Therefore, the above equation leads to an over-prediction of turbulent shear stress, which in turn yields higher values of turbulent eddy viscosity. As a result, the predictions show that the flow has a tendency to adhere to walls in

situations at which it would actually separate. The SST model eliminates this problem with the application of the following equation for the turbulent eddy viscosity (Menter (1994))

$$\nu_t = \frac{a_1 k}{\max\left(a_1 \omega, \frac{\partial u}{\partial y} G\right)} \quad (4.18)$$

The variable G is 0 in free shear flows and 1 in boundary layer flows. Therefore, in the free shear regions, where Bradshaw's assumption ($\tau_t = \rho a_1 k$) may not be valid, the relationship for the turbulent eddy viscosity reverts to the expression used by previous two-equation turbulence models.

The SST model has been found to be the most suited turbulence model for turbomachinery flows, because of its accurate prediction of separation and flows in adverse pressure gradients. In addition, the improved accuracy comes with little computational cost. Better models, like RSM, do exist, however, in many cases the higher computational cost and the instability of RSM models often outweigh the possible benefits that come with their use.

4.2.6 Reynolds Stress Model (RSM)

Unlike the previous two-equation models, the RSM model determines the turbulent stresses by solving a transport equation for each stress component, without the need to rely on the Boussinesq eddy-viscosity approximation. Unlike two-equation and one-equation models, the RSM accounts for the effect of flow history due to terms representing the convection and diffusion of shear stress tensors. The effects of streamline curvature, system

rotation and stratification can be taken into account through convection and production terms (Wilcox (2000)). The RSM model also has modelled terms, which are derived from a combination of theoretical assumptions and empirical results. One of the important modelled terms in the RSM is the pressure-strain term, which plays an important role in determining the structure of turbulent flows, distributing turbulent energy among the Reynolds stress components and accounting for flow anisotropy. This term was initially modelled by considering homogeneous flows. More recently, different approaches to model the pressure-strain term have also been made. The RSM may need careful fine tuning of their coefficients to avoid solution divergence. The complex Reynolds stress models are known to give results superior to those of one- and two-equation turbulence models for flows with streamline curvature, sudden change in strain rate and secondary motions of the second kind, although at the cost of increased computing time (Bradshaw (1997)).

4.3 Spatial Discretization

The process of spatial discretization is important in determining the accuracy of the simulation. If regions with high flow variable gradients are not refined to a sufficient degree, inaccuracies in the solution are sure to occur. In some cases, the solution may even diverge. Therefore, a substantial effort must be dedicated to the spatial discretization of the model.

There are many grid generation programs commercially available. In the present work, the model was meshed with the commercial software GAMBIT 2.3.16 and TGrid 4.0.24. GAMBIT 2.3.16 is a surface and volume meshing program, whereas TGrid 4.0.24

is strictly a volume meshing program, but TGrid 4.0.24 contains a more robust prism boundary layer mesh generator than GAMBIT 2.3.16. To optimize the mesh, the surface mesh for the domain was generated in GAMBIT 2.3.16 and then imported into TGrid 4.0.24 for volume grid generation.

The main grid types used in modeling gas turbine flows include *H-type structured* grids and tetrahedral *unstructured* grids. A compromise between the two is a *hybrid* grid, which uses structured grids in the boundary layer regions and unstructured grids in the other regions.

Structured grids use quadrilaterals to form the surface mesh and hexahedra to form the volume mesh. These grids typically take a long time to generate using algebraic grid generation or grid generation using PDEs (Blazek (2005)), and make mesh adaption (the term adaption is used in FLUENT 6.3.26 documentation instead of the more widely used term adaptation) rather difficult to implement. However, the solution time on structured grids is lower, when compared to that for unstructured grids. One major disadvantage of structured meshes is that the physical space must be mapped onto the computational space. This may require that the physical space be altered by splitting or sectioning the domain so as to make it “mappable”. For a surface to be “mappable”, it must be composed of four or more faces (Ansys (2008)). If a surface is composed of more than four faces, the additional faces must be combined until the surface contains four faces. This can be seen in Figure 4.3, where the suction or pressure surface side of the blade is combined with the attached periodic boundary, upstream and downstream of the blade, to form one face. Therefore, the H-type structured grid is formed in a mappable surface with four sides; at the inlet, at

the outlet, one formed by the suction side of the blade and two periodic boundaries and one formed by the pressure side of the blade and two periodic boundaries. As a result, it can be very difficult to generate a structured grid when the model geometry is complex, as in the case of a blade with a large twist or a non-uniform blade profile or a tip gap region. In some cases, the domain can be subdivided into smaller mappable regions, but this can be time consuming and difficult to accomplish. Another disadvantage of structured meshes is that refinement in regions where it is required may lead to unwanted mesh refinement in other parts of the domain, as seen in Figure 4.3. This Figure shows that local refinement near the leading edge and the trailing edge of the rotor propagates into the domain, where refinement is unnecessary. Therefore, local mesh refinement can be difficult.

Unstructured grids are composed of triangular elements on the surface of the model and tetrahedral elements in the volume. The main advantage of these types of grids is that they make it very easy to discretize complex domains and meshing can be done very quickly. However, as the mesh does not need to have any order, accuracy in the wall regions is reduced. Achieving the required mesh resolution to decrease the errors in the near wall region can lead to extremely large meshes. In many cases, it is worth the extra time to develop a hybrid mesh, as it would have a smaller node count with respect to an unstructured mesh, thus reducing the computational time. Unlike the structured grids, unstructured grids cannot be mapped onto a computational space. Therefore, unstructured grids require a connectivity matrix to inform the solver of which nodes are neighbours. This requires additional computational memory and can lead to increased computational time with respect to structured grids. However, a distinct advantage of unstructured meshes,

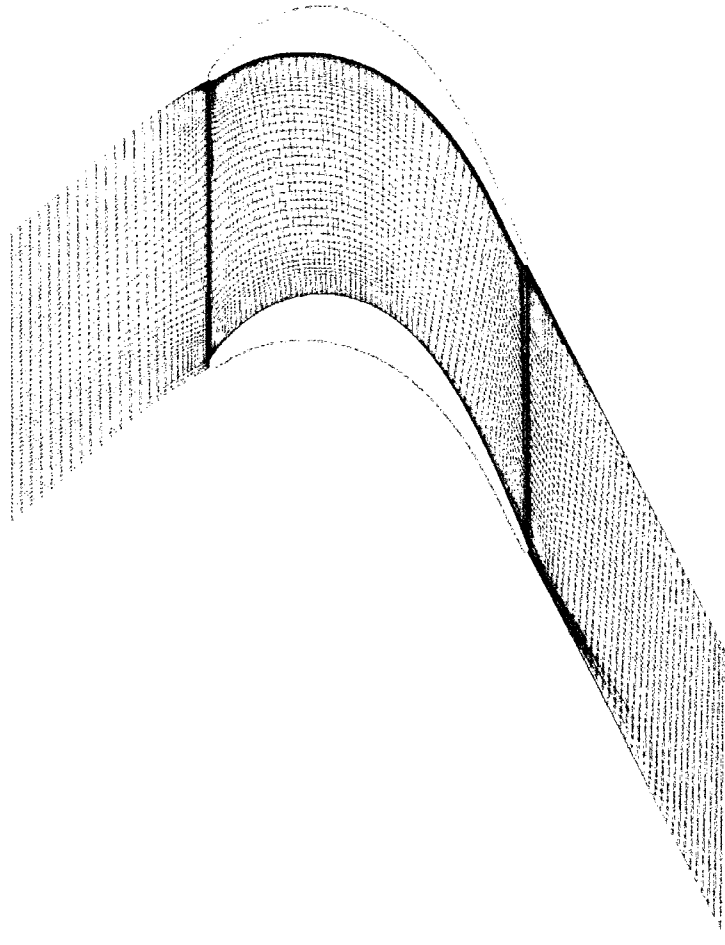


Figure 4.3: H-type structured grid.

over structured meshes, is that they can easily be refined locally with little effect on the rest of the domain, as shown in Figure 4.4.

A compromise between the unstructured mesh and the structure mesh is the hybrid mesh. This type of mesh allows control of the mesh in the near-wall regions, while also allowing easy meshing of complex geometries. In addition, local refinement does not propagate into the domain and an excessive number of nodes is not required in the boundary layer to achieve the desired dense meshing near the wall.

The generation of a structured mesh in a 3-D gas turbine requires specialized meshing software. However, for 2-D simulations, a hybrid mesh (Figure 4.5) can be constructed using only GAMBIT 2.3.16. GAMBIT 2.3.16 has the ability to generate an unstructured mesh and a 2-D structured mesh in the boundary layer simultaneously. For the 3-D simulations, an unstructured mesh was first generated using GAMBIT 2.3.16 and TGrid 4.0.24 was used to generate the 3-D structured mesh in the boundary layer after the unstructured mesh was created.

4.4 Simulation Verification

Among the several methods that were found to estimate the uncertainty in the simulation results, two were used in the current study: the *Grid Convergence Index* (GCI) method of Roache (1998) and the *Approximate Error Spline* (AES) method of Celik and Li (2005). The GCI method gives a 90% confidence estimated error band in which the true solution lies. This method requires that the solutions on consecutive grids achieve monotonic convergence. The AES method uses parts of the GCI method to also give a 90%

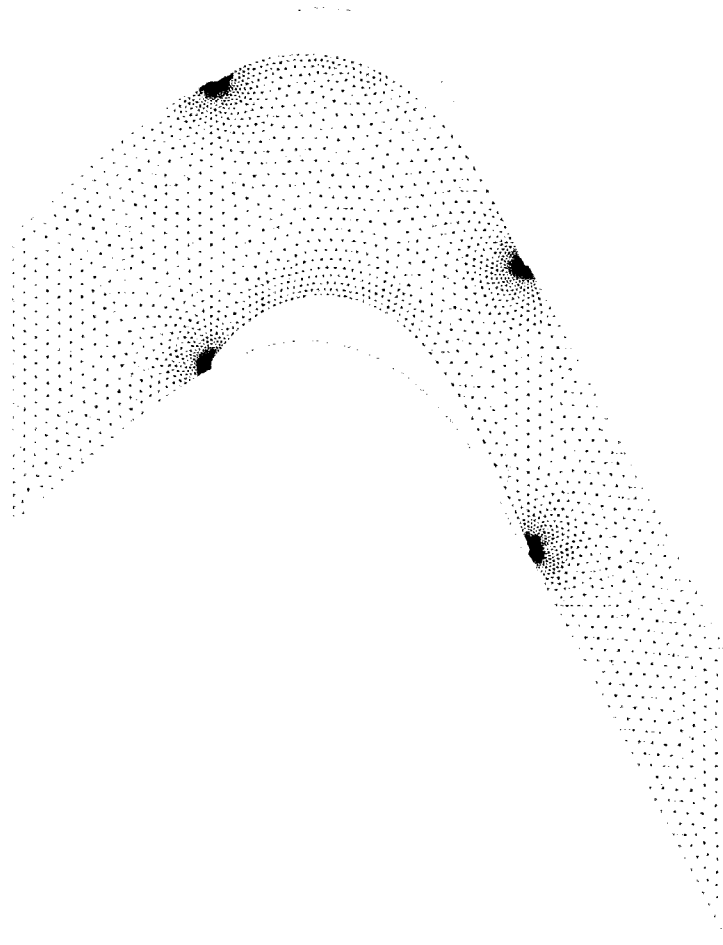


Figure 4.4: Unstructured tetrahedral grid.

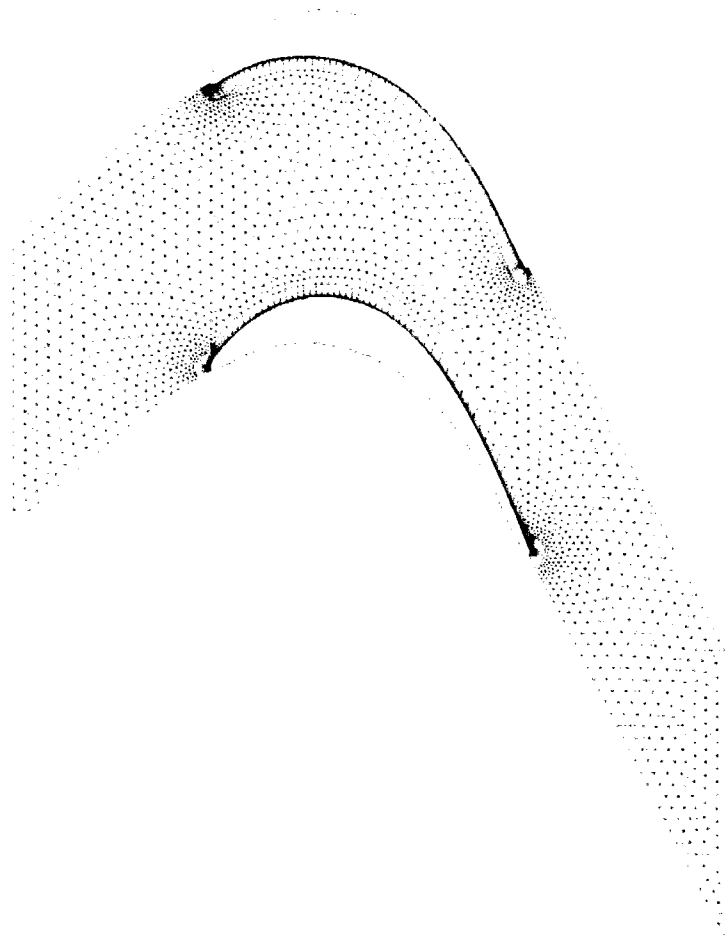


Figure 4.5: Hybrid mesh with structured cells near wall regions and unstructured cells elsewhere.

Table 4.1: Grid labelling for a verification study with only 2 grids

Grid	2	1
Cell spacing	Coarse	Fine

Table 4.2: Grid labelling for a verification study with 3 grids

Grid	3	2	1
Cell spacing	Coarse	←	Fine

confidence estimated error band, but it can be implemented in cases in which the solutions on consecutive grids follow either monotonic or oscillatory convergence. Grid numbering, shown in Tables 4.1, 4.2 and 4.3, was used for this study. As shown in these tables, the grids with the finest mesh (highest number of nodes or cells) were assigned the number 1, with all subsequent meshes being assigned higher numbers as the mesh became coarser.

4.4.1 Grid Convergence Index (GCI) Method

In order to estimate the relative error of the numerical solution by comparison to the estimated true value, a method developed by Roache (1998) is used. This method is based on Richardson Extrapolation, which estimates the exact value of any computational variable g_{exact} using the grid refinement ratio σ and the order p of the computational method through

$$g_{exact} \cong g_I + \frac{g_I - g_{I+1}}{\sigma^p - 1} \quad (4.19)$$

Table 4.3: Grid labelling for a verification study with only more than 3 grids

Grid	n	←	3	2	1
Cell spacing	Coarse	←	Intermediate 2	Intermediate 1	Fine

where g is the value of a computational parameter and I , in the subscripts, is the grid number as shown in Tables 4.1, 4.2 and 4.3.. For unstructured grids, σ can be estimated by

$$\sigma = \left(\frac{m_I}{m_{I+1}} \right)^{1/D} \quad (4.20)$$

where m_1 and m_2 are the numbers of nodes on the fine and medium grids, respectively, D is the number of spatial dimensions in the problem. With equation 4.20, an estimated fractional error E_f , which is a reliable estimate of the actual fractional error when $E_f \ll 1$, can be determined, for the fine grid (grid 1) from (Roache (1998))

$$E_f^1 = \frac{\delta_{12}}{\sigma^p - 1} \quad (4.21)$$

$$\delta_{12} = \frac{g_2 - g_1}{g_1} \quad (4.22)$$

A requirement of this procedure is to estimate the “error band”, namely the region around the computational solution within which the estimated true solution resides with a high degree of confidence. In the CFD community, it is accepted that a practical (90%) confidence level would be achieved for δ if the simulation were run at second order accuracy ($p = 2$) and grid doubling were used ($\sigma = 2$). If the numerical study has not been performed under these specific conditions, e may not be a good indication of the error band with sufficient confidence (Roache (1998)). Consequently, Roache (1998) introduced the Grid Convergence Index (GCI), which relates the value of δ from the grid convergence study done by the user to the value of δ of a grid convergence study that would use $\sigma = 2$ and $p = 2$. The GCI for the consecutive grids (grids I and $I + 1$) are defined as

$$\text{GCI}^I = K \frac{|\delta|}{\sigma^p - 1} \quad (4.23)$$

$$\text{GCI}^{I+1} = K \frac{|\delta| \sigma^p}{\sigma^p - 1} \quad (4.24)$$

$$\delta = \frac{g_{I+1} - g_I}{g_I} \quad (4.25)$$

where $K = 3$, when the order of the simulation p is estimated from a grid convergence study with 2 grids, and $K = 1.25$, when p is estimated using 3 grids (Roache (1998)).

For practical problems, the order of convergence can be estimated from computational values on three different grids. If the grid refinement ratio has the same value for the fine and intermediate grids (σ_{12}) and for the intermediate and coarse grids (σ_{23}), then p can be easily determined as

$$p = \ln \left(\frac{g_3 - g_2}{g_2 - g_1} \right) / \ln \sigma \quad (4.26)$$

where the subscripts 1, 2 and 3 indicate the values of g on the fine, and intermediate/coarse grids, respectively. If σ is not constant, then p must be determined by solving the following equation

$$\frac{\delta_{23}}{\sigma_{23}^p - 1} = \sigma_{12}^p \left[\frac{\delta_{12}}{\sigma_{12}^p - 1} \right] \quad (4.27)$$

$$\delta_{12} = \frac{g_2 - g_1}{g_1} \quad (4.28)$$

$$\delta_{23} = \frac{g_3 - g_2}{g_2} \quad (4.29)$$

To solve equation 4.27, a direct substitution iteration code has been written, using Matlab,

which utilized the following iteration equation (Roache (1998))

$$p_l = \xi p_{l-1} + (1 - \xi) \frac{\ln(\lambda)}{\ln(\sigma_{12})} \quad (4.30)$$

$$\lambda = \frac{(\sigma_{12}^{p_{l-1}} - 1) \delta_{23}}{(\sigma_{23}^{p_{l-1}} - 1) \delta_{12}} \quad (4.31)$$

The parameter ξ is the under-relaxation factor, which prevents divergence of the solution. An under-relaxation parameter value of 0.5 was used, following the suggestion of Roache (1998).

In many cases, comparisons between grids are done with more than one parameter. If this is the case, then the orders of convergence may not be identical for all parameters. A question arises then: which parameter should determine the order of convergence? In fact, there is no overall convergence rate. Each parameter has a specific convergence rate that is unique, therefore the order of convergence is calculated separately for each flow parameter. The GCI is then calculated using the order of convergence specific to each flow parameter.

The Roache (1998) method assumes that the solutions on the two grids, used to estimate the GCI, are in the asymptotic range. One is reminded that the difference between simulation parameters on consecutive grid decreases by a minimum amount as defined by Equation 4.32, as grid refinement increases. In order to determine whether the solutions are in the asymptotic range, one is required to use a third grid. To determine whether a parameter on two grids is in the asymptotic range, the GCI for the fine grid (GCI_{12}^1) and the GCI for the nearest intermediate/coarse grid (GCI_{23}^2) are calculated. The subscripts 12 and 23 indicate which grids were used for the calculation of the GCI (ie. GCI_{12}^1 indicates that the solutions on grids 1 and 2 were used to calculate the GCI for grid 1, whereas

GCI_{23}^2 indicates that the solutions on grids 2 and 3 were used to calculate the GCI for grid 2). If the fine and intermediate grids are in the asymptotic range, then the following would be true

$$\frac{GCI_{23}^2}{GCI_{12}^1} \approx \sigma_{12}^p \quad (4.32)$$

Alternatively, if GCI_{12}^2 is comparable to GCI_{23}^1 , then the two grids would be in the asymptotic range. However, the grids may not always lie in the asymptotic region ($p < 1$), especially if further refinement of the grid is not possible due to limitations in computational resources or time constraints.

The GCI gives a good estimate of the error band, while maintaining an acceptable (90%) confidence level in the estimate. Figure 4.6 shows the 90% confidence error band that the GCI represents with respect to the exact solution g_{exact} and the numerical solution g . In addition, because the GCI is representative of the error relative to a grid convergence study completed with grid doubling and second-order accurate simulations, error bands from different grid convergence studies can be compared directly.

4.4.2 Approximate Error Spline (AES) Method

The AES method accounts for the possibility that the solutions on various grids used in the verification study may follow an oscillatory convergence pattern. Therefore, this method is more robust and can be used in many more situations than the GCI method.

This method estimates the exact value of a simulation parameter by first estimating the approximate error E_a of the simulation parameter on the finest grid (grid 1). The approximate error is used to estimate the true error E_t , which is used to find the exact

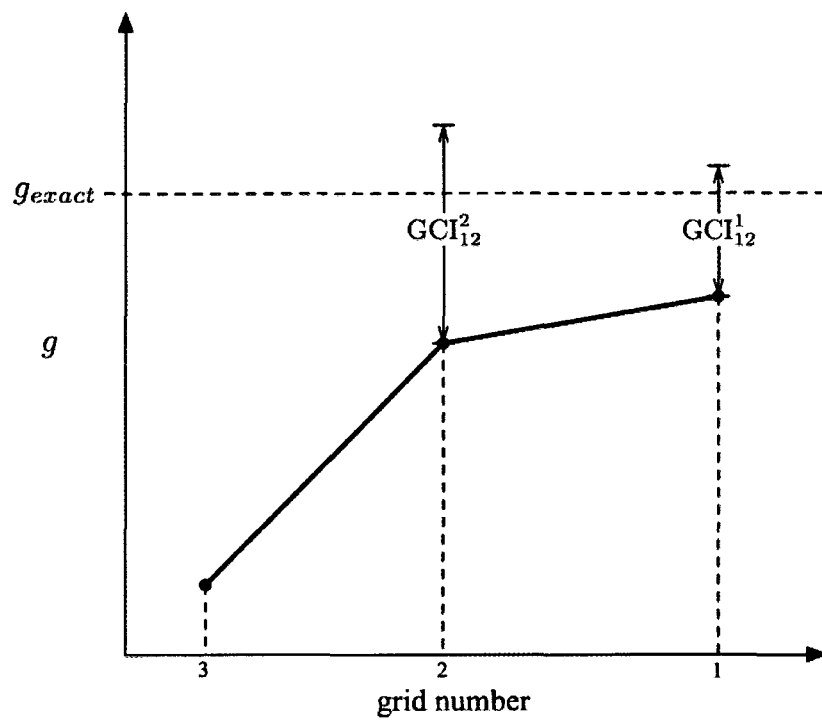


Figure 4.6: Schematic representation of the GCI.

value of the simulation parameter. The exact value can be used to determine the GCI factor.

To find the approximate error on the finest grid, the approximate errors for all coarser grids are plotted vs. the average grid size d . Normally the three finest grids are used for the verification study in order to reduce the simulation uncertainty. However, for generalization, if any three consecutive grids were used for the verification study, d and E_a would be defined as (Celik et al. (2005))

$$d = \sqrt[3]{\frac{V}{m_{cells}}} \text{ for a 3-D case} \quad (4.33)$$

$$d = \sqrt{\frac{A}{m_{cells}}} \text{ for a 2-D case} \quad (4.34)$$

$$E_a(d_{I+2}) = g_{I+2} - g_{I+1} \quad (4.35)$$

$$E_a(d_{I+1}) = g_{I+1} - g_I \quad (4.36)$$

where V and A are the volume and the area of the simulation domain, respectively, and m_{cells} is the total number of cells in the simulation domain. It is assumed that the approximate error at $d = 0$ (infinitely small cell size) is 0. Therefore, the only unknown approximate error is $E_a(d_I)$. The approximate error at d_I can be found by fitting cubic splines through the known data points. Cubic spline fitting also requires the endslopes at $d = 0$ and $d = d_{I+2}$. These endslopes are given by (Celik et al. (2005))

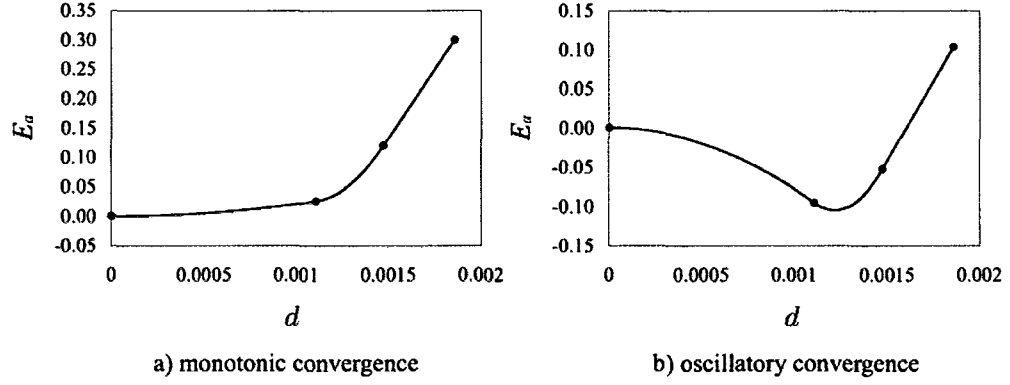


Figure 4.7: Plots of approximate error as a function of cell size for monotonic and oscillatory convergence.

$$\frac{dE_a}{dd}(0) = 0 \quad (4.37)$$

$$\frac{dE_a}{dd}(d_{I+2}) = \frac{E_a(d_{I+2}) - E_a(d_{I+1})}{d_{I+2} - d_{I+1}} \quad (4.38)$$

At $d = 0$ the assumption that the endslope is 0 is valid for all numerical schemes with an order greater than 1. If the method is of first order, then the endslope at $d = 0$ may not be 0, however, this assumption would still produce good results (Celik et al. (2005)). Figure 4.7 shows a completed plot of E_a as a function of d for a monotonic convergence case and an oscillatory convergence case.

The true error is found from a complex function that involves the approximate error, the average cell size and the grid refinement ratio σ , which for the AES method is defined as (Celik et al. (2005))

$$\sigma = \frac{d_{I+2}}{d_{I+1}} = \frac{d_{I+1}}{d_I} \quad (4.39)$$

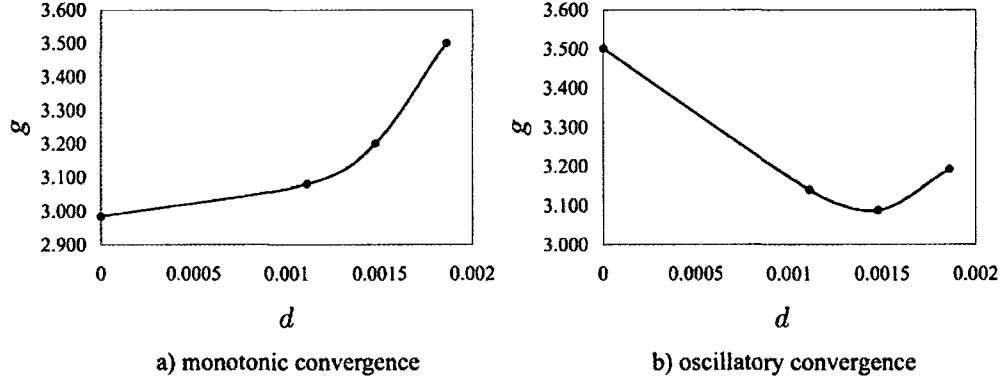


Figure 4.8: Plots of flow parameter as a function of cell size for monotonic and oscillatory convergence.

The details of the function relating E_t and E_a can be found in Celik et al. (2005). Once the true error is found, the exact value of the simulation parameter in question g_{exact} can be found as (Celik et al. (2005))

$$g_{exact} = g_I - E_t(d_I) \quad (4.40)$$

Figure 4.8 shows the corresponding plot of g as a function of d for the cases in Figure 4.7.

The GCI for grid I is found using the expression

$$\text{GCI}^I = K \left| \frac{g_{exact} - g_I}{g_I} \right| \quad (4.41)$$

The main advantage of the AES method is that it can be used to estimate the GCI in cases for which a simulation parameter shows non-monotonic convergence as the grid is refined. Celik and Li (2005) also showed that the AES method had the best performance among four different methods of estimating the uncertainty in a simulation parameter. This comparison was done with flow parameters that exhibited monotonic and oscillatory

convergences. The AES method was also found to be exceptionally superior when the simulation parameter exhibited oscillatory convergence and three grids were used for the simulation verification.

Chapter 5

Overview of Relevant Experimental Studies

5.1 T106 Low Pressure Turbine Cascade

Experiments on the T106 low pressure turbine cascade were conducted at the Whittle Laboratory at Cambridge University in Cambridge, U.K. (Stieger et al. (2003)). The experimental setup is shown in Figure 5.1 with a detail of the cascade shown in Figure 5.2. Table 5.1 summarizes the geometrical properties of this model shown in Figure 5.3. The T106 blade has a chord $c = 198$ mm and an axial chord length $c_a = 170$ mm. The blade pitch L_{rp} and the span S are 92.9% and 220.6% of the axial chord length, respectively. The inlet flow angle is 37.7° and the design outlet flow angle is 63.2° .

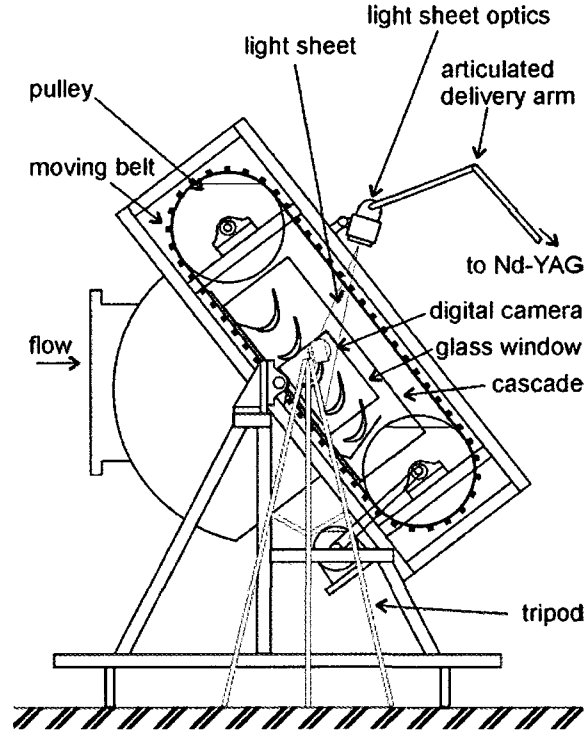


Figure 5.1: Experimental setup for T106 experiments (Stieger et al. (2003)).

Table 5.1: T106 low pressure turbine cascade specifications

Number of blades	5
Chord, c (mm)	198
Axial chord, c_a (mm)	170
Blade stagger ($^\circ$)	59.3
Pitch, L_{rp} (mm)	158
Span, S (mm)	375
Suction surface length (mm)	264.7
Pressure surface length (mm)	230
Inlet flow angle, α ($^\circ$)	37.7
Design exit flow angle, β ($^\circ$)	63.2

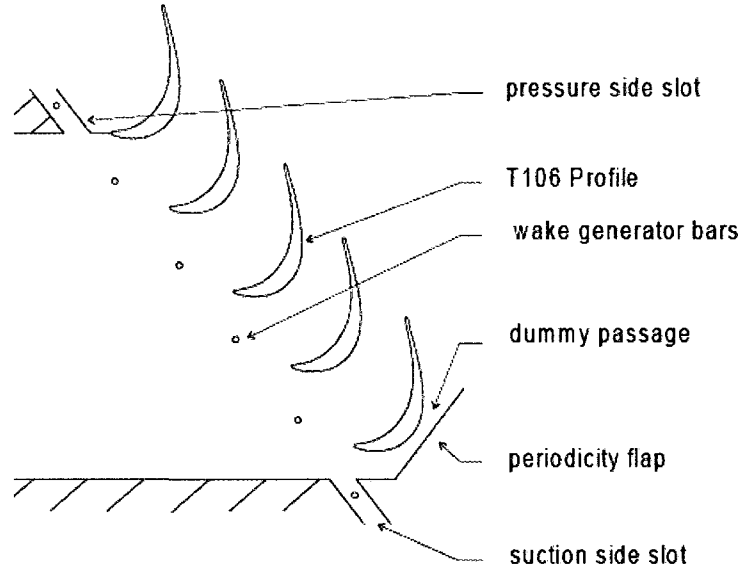


Figure 5.2: Detail of T106 cascade (Stieger (2002)).

5.1.1 Experimental Conditions

The flow over the T106 airfoil is incompressible and in some cases incoming wakes were simulated through moving bars upstream of the cascade. However, the current study is only concerned with the results of the steady state case (no moving bars or incoming wakes). Only the exit Reynolds number, based on chord $Re_{2,c} = 1.6 \times 10^5$, was available for the steady case (Stieger (2002), Stieger et al. (2003)). No data were available for the inlet conditions (especially the inlet turbulent kinetic energy) which makes conducting simulations for this case difficult.

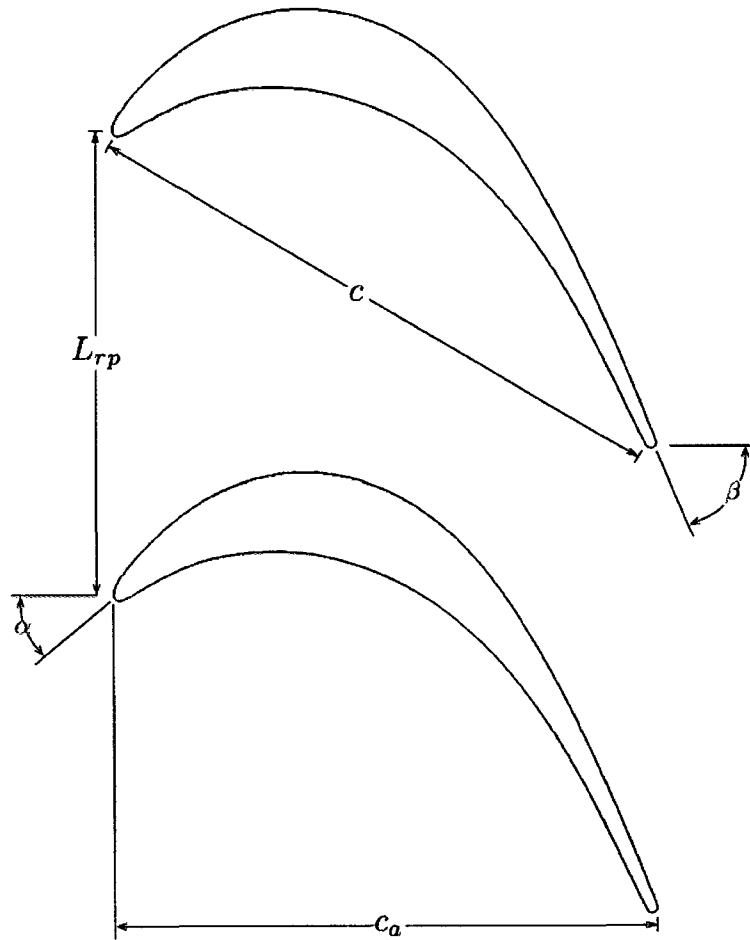


Figure 5.3: T106 blade profile.

5.1.2 Static Surface Pressure Measurement

Static pressure measurements were made using surface static pressure taps. Tapping was accomplished through 0.3 mm holes drilled normal into the surface of the T106 blade at 25% span (Stieger (2002)). Oil and dye flow visualizations showed that the flow at 25% span was in the two-dimensional flow region in the cascade (Stieger et al. (2003)). The taps were connected to a Scanivalve with 1.0 mm ID PVC tubing. Pressure measurements were obtained using a Druck PDCR 22 pressure transducer with a 3700 Pa gauge range. Transducer output voltage was amplified using a PCI-MIO-16E-1 A/D card. Linear calibration of the transducer was accomplished using a Druck DPI520 ATE pressure controller. Pressure measurements were non-dimensionalized to give the coefficient of pressure for the T106 experiments as

$$C_p = \frac{P_{01} - P}{P_{01} - P_{2s}} = \left(\frac{U}{U_{2s}} \right)^2 \quad (5.1)$$

where P_{01} is the total inlet pressure, P is the measurement pressure on the blade surface, P_{2s} is the station isentropic exit pressure, U is the velocity at the pressure measurement location, and U_{2s} is the isentropic exit velocity.

The Scanivalve system had a ± 2490 Pa (10 in H₂O) range and its output was discretized using a 16-bit A/D converter. This gives a discretization error of 1.0 Pa, which is equivalent to 1.0% exit dynamic head at $Re_{2,c} = 1.6 \times 10^5$.

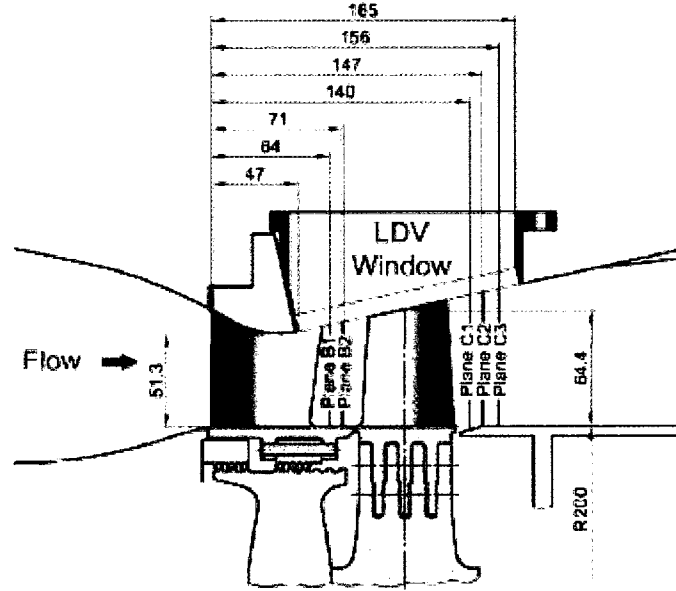


Figure 5.4: Profile of the TTM test section. Stator vanes and rotor blades are shown on the left and right respectively; Dimensions are in mm (Göttlich et al. (2004)).

5.2 TTM High Pressure Turbine

Experiments with the TTM single-stage high pressure turbine were conducted at the Institute for Thermal Turbomachinery and Machine Dynamics (TTM) at Graz University of Technology in Austria (Göttlich et al. (2004)). A side view of the TTM turbine is shown in Figure 5.4 and an isometric view is shown in Figure 5.5. The turbine stage consists of 24 nozzle guide vanes and 36 rotor blades. Additional stage variables are given in Table 5.2.

5.2.1 Experimental Conditions

In the experimental setup, the intake fluid was not the product of a combustion process therefore, temperatures were much lower than temperatures in a commercial gas

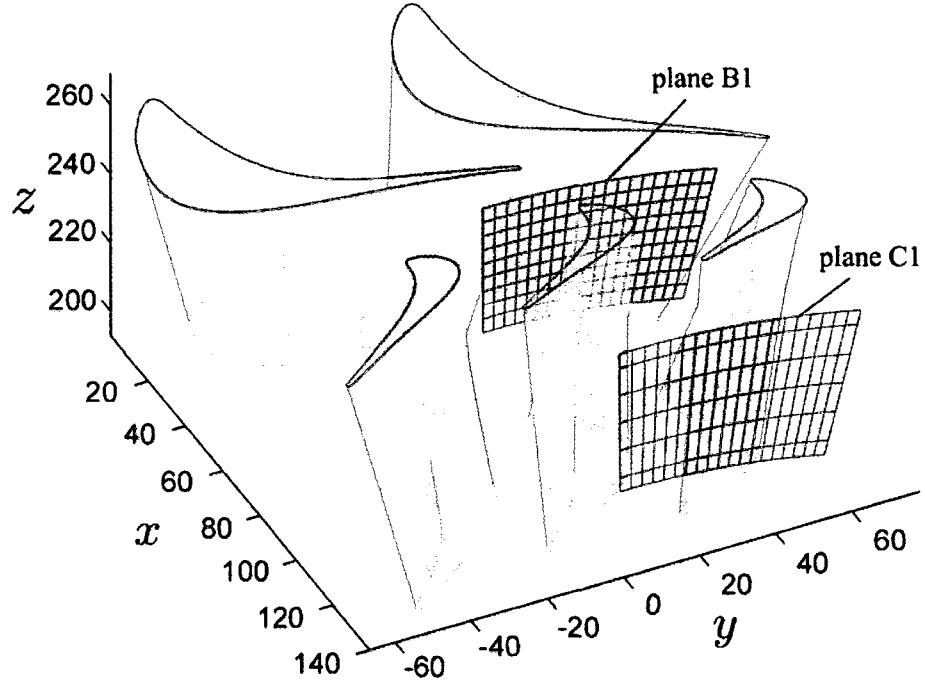


Figure 5.5: Three-dimensional view of 2 stator vanes and 3 rotor blades in the TTM turbine stage and relative locations of planes B1 and C1 (Göttlich et al. (2004)).

Table 5.2: TTM turbine stage properties

Number of nozzle guide vanes	24
Number of rotor blades	36
Midspan nozzle chord (mm)	78.9
Midspan axial nozzle chord (mm)	56.1
Geometric turning angle of nozzle (°)	70
Midspan blade chord (mm)	55.9
Midspan axial blade chord (mm)	46.8
Geometric turning angle of blade (°)	107
Nozzle height at exit (mm)	55.1
Rotor blade height at exit (mm)	69.2
Rotor tip clearance/span (%)	1.4
Vane-blade spacing (% of midspan axial nozzle chord)	47

Table 5.3: TTM ambient experimental conditions

Mean value	T_{amb} [°C]	P_{amb} [kPa]	Humidity [%]
Plane B1	3.4 to -3.1	97	46 to 77
Plane C1	32.1 to 15.8	97 to 96	41 to 71
Plane B1 & C1	32.1 to -3.1	97	41 to 77

turbine engine. In addition, the air used in the experiment is not mixed with any combustion products. Experiments also show that the flow through the TTM turbine was found to be compressible.

Experimental data collected consisted of the ambient temperature T_{amb} , the ambient pressure P_{amb} , the turbine rotor rotational speed Ω , the mixing chamber total pressure $P_{0,MC}$, the total inlet temperature $T_{0,in}$, the pressure expansion ratio $P_{0,MC}/P_{exit}$, the total inlet pressure $P_{0,in}$, and the static output pressure P_{out} . Data were collected on planes B1 and C1 at different times of the year, as shown in Table 5.3. During the winter, data were collected along plane B1, whereas data on plane C1 were collected in the summer time. After the data were collected during different seasons, the results were averaged and are presented in Table 5.4 under the heading Plane B1 & C1. Figure 5.6 shows the location of pressure measurements that are summarized in Table 5.4. Pressure measurements downstream of the rotor was accomplished with static pressure taps located in the hub. The reduced pressure in the exhaust chamber P_{exit} was obtained with the use of a suction blower.

5.2.2 Measurement Planes

Velocity data were collected for the axial u_a and circumferential u_θ components, at each point separately, along radial lines of the measurement grids shown in Figure 5.5. Along

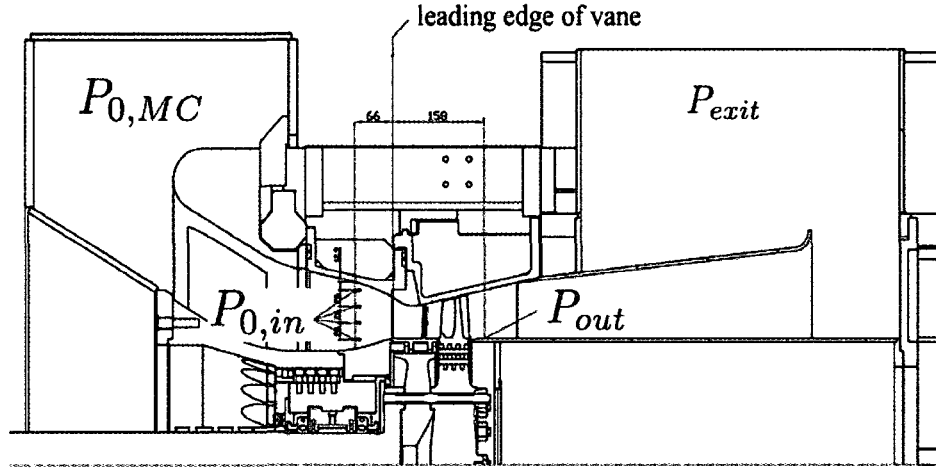


Figure 5.6: TTM experimental boundary conditions (Göttlich et al. (2004)).

Table 5.4: TTM experimental boundary conditions

	Ω [RPM]	W [kW]	$P_{0,MC}$ [bar]	$T_{0,in}$ [K]	$\frac{P_{0,MC}}{P_{exit}}$	$P_{0,in}$ [bar]	P_{out} [bar]
Plane B1							
Mean value	10621	1838	3.390	403.0	3.511	3.387	0.915
Variance	5.395	3.297	0.006	0.170	0.009	0.011	0.010
Error [%]	0.051	0.179	0.174	0.042	0.257	0.322	1.055
Plane C1							
Mean value	10614	1785	3.280	402.4	3.484	3.278	0.885
Variance	24.290	16.116	0.022	1.663	0.007	0.006	0.003
Error [%]	0.229	0.903	0.680	0.413	0.206	0.188	0.356
Plane B1 & C1							
Mean value	10617	1811	3.335	402.7	3.497	3.332	0.900
Variance	17.594	11.632	0.016	1.182	1.182	0.009	0.007
Error [%]	0.166	0.642	0.489	0.294	0.294	0.266	0.798

plane B1, data at 9 radial locations were obtained, while along plane C1, data at only 7 radial locations were obtained. The measurement points do not extend to the endwall because laser reflections from the hub and window prevented data collection in these regions. The circumferential measurement position was adjusted by rotation of the stator (the positive rotation direction is indicated in Figure 5.7). A trigger mechanism ensures that the LDV collects data when the rotor and stator are in the same relative position, which allows for phase-averaging of the results. In order to keep flow velocities in the turbine within the measurement range of the LDV system, the LDV laser was rotated by 24° in plane B1 as shown in Figure 5.8. Data for both planes were obtained 120 times per periodic passing period ($\tau_p \approx 4.71 \times 10^{-4}$ s) and a total of approximately 80000 samples were collected for each point for phase-averaging. Using these data, the ensemble average velocities $\langle u_a \rangle$ and $\langle u_\theta \rangle$ and the variances $\langle u_a'^2 \rangle$ and $\langle u_\theta'^2 \rangle$ were determined for different rotor positions.

The uncertainty, with 95% confidence, in the mean velocity and variance is 3.5 m/s in the rotor wake and 0.9 m/s outside of the rotor wake for the ensemble averaged velocity (Göttlich et al. (2004)). For the unresolved velocity component the uncertainty, with 95% confidence, is 11 m/s in the rotor wake and 2.5 m/s outside the rotor wake (Göttlich et al. (2004)).

5.2.3 LDV System

LDV and pneumatic flow field results were obtained for the TTM turbine stage along planes B1 and C1, shown in Figure 5.5. Göttlich et al. (2004) specify that plane B1 is located 64 mm behind the leading edge of the stator vanes and plane C1 is located 140 mm behind the leading edge of the stator vanes. The LDV system used back-scattering from

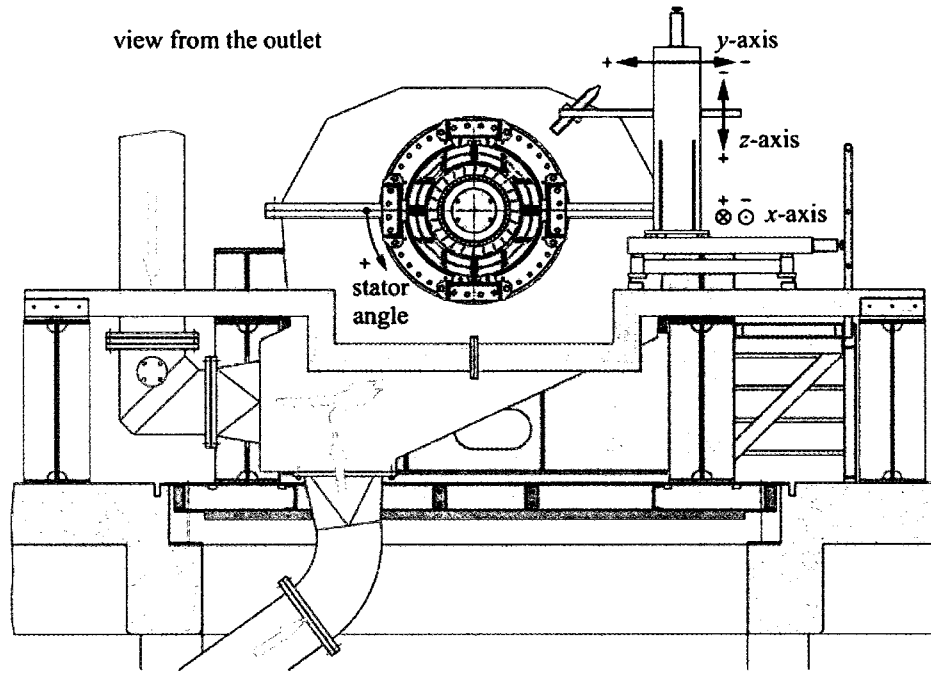


Figure 5.7: TTM LDV traverse support (Göttlich et al. (2004)).

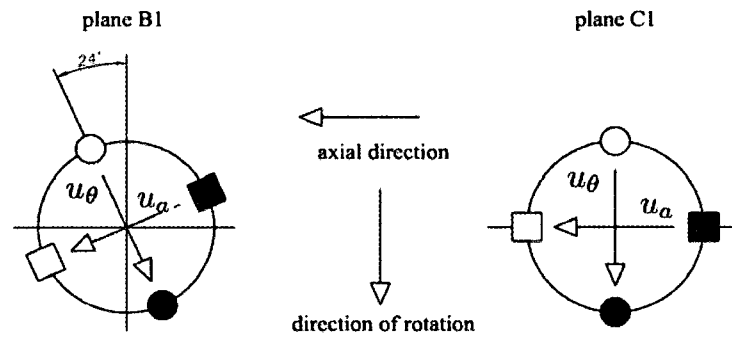


Figure 5.8: LDV velocity measurement directions (Göttlich et al. (2004)).

seeding particles (fine oil particles of DEHS; Di-Ethyl-Hexyl-Sebacin-ester supplied by a PALLAS AGF 5.0 seeding generator) to determine flow velocities in the turbomachine. The seeding particle size was kept small (0.3-0.6 μm), because high particle response (inversely proportional to particle size) is necessary for transonic flows. If the seeding particle is relatively large, then the shock will appear smeared by an amount equal to $3\tau_c U$, where U is the velocity downstream of the shock and the time constant τ_c is given by Zwart (1995)

$$\tau_c = \frac{d_p^2 \rho_p}{18\mu} \quad (5.2)$$

Therefore, increasing the particle size would increase the distance over which the shock would be smeared, whereas keeping the particle size and density as small as possible would reduce this smearing effect. The rotor blades, stator vanes and endwalls were also coated with a high-temperature flat black paint to reduce reflections from the surfaces.

The LDV system accessed the flow field through a 9 mm thick window (shown in Figure 5.4 and 5.9) with a surface area of 120 mm $\times$ 23 mm. The LDV was a two-component system (DANTEC Fibre Flow with BSA processors) which used an argon-ion laser from COHERENT. The LDV traverse was mounted on a vibration isolation support, as shown in Figure 5.7. Table 5.5 contains details of the LDV system settings.



Figure 5.9: Photograph showing the window in the casing of the TTM turbine used for the LDV measurements (Göttlich et al. (2004)).

Table 5.5: TTM LDV optical beam system

Velocity component	u_a (BSA1)	u_θ (BSA2)
Velocity directions along plane B1	see Figure 5.8	see Figure 5.8
Velocity directions along plane C1	axial	circumferential
Focal length [mm]	400	400
Wavelength [nm]	514.5	488
Beam spacing [mm]	38	38
Beam diameter [mm]	2.2	2.2
Focal spot diameter [μm]	119.1	112.9
Number of fringes	21	21

Chapter 6

Numerical Conditions and Procedures

Two simplified gas turbine models were used during the course of this study. The models are the 2-D *T106* cascade and the 3-D *TTM* turbine. The 2-D T106 cascade was obtained from the *Whittle Laboratory* at Cambridge University in Cambridge, England (Stieger et al. (2003)). The blade profile designation of the Whittle Laboratory low pressure cascade is T106 and was chosen because studies of this cascade are well documented. The 3-D TTM turbomachine geometry was obtained from the *Institute for Thermal Turbomachinery and Machine Dynamics* (TTM) at Graz University of Technology in Austria (Göttlich et al. (2004)). This high pressure turbine has 36 rotor blades and 24 stator vanes. This chapter summarizes the simulation geometry, its discretization, and the numerical conditions and schemes used for the simulation of the T106 cascade and the TTM turbine.

6.1 Hardware and Software

The simulations were performed on three different systems: a) an in-house personal computer, b) an in-house cluster of Sun Fire workstations, or c) the *High Performance Computing and Virtual Laboratory* (HPCVL). Available in-house computing resources consisted of an HP Pavilion a1730n workstation with an 2.6 GHz AMD Athlon 64 X2 Dual Core Processor 5000+ and 3.5 GB of RAM. The local Sun Fire cluster consists of six Sun Fire X2200 M2 x64 servers, each with $2 \times 2.2\text{GHz}$ AMD Opteron 2214 dual core processors and 8 GB of RAM. The resources available at HPCVL consisted of six Sun Fire 25000 Nodes with $72 \times$ dual-core UltraSPARC-IV+ 1.5 GHz processors with 576 GB of RAM each and one Sun Fire 25000 Node with $72 \times$ dual-core UltraSPARC-IV+ 1.8 GHz processors with 576 GB of RAM.

The software GAMBIT 2.3.16 was used for computer model generation and mesh generation. TGrid 4.0 was used for enhanced mesh control and for 3-D structured wall meshing. The commercial code FLUENT 6.3.26 was used for all CFD simulations, whereas Matlab and Tecplot 360 were used for postprocessing and visualization.

6.2 Computational Domain and Grid Generation

6.2.1 T106 Cascade

The original experimental apparatus had 5 blades, however, the numerical model was reduced to only 1 blade and periodic boundary conditions were used to model adjacent blades as shown in Figure 6.1.

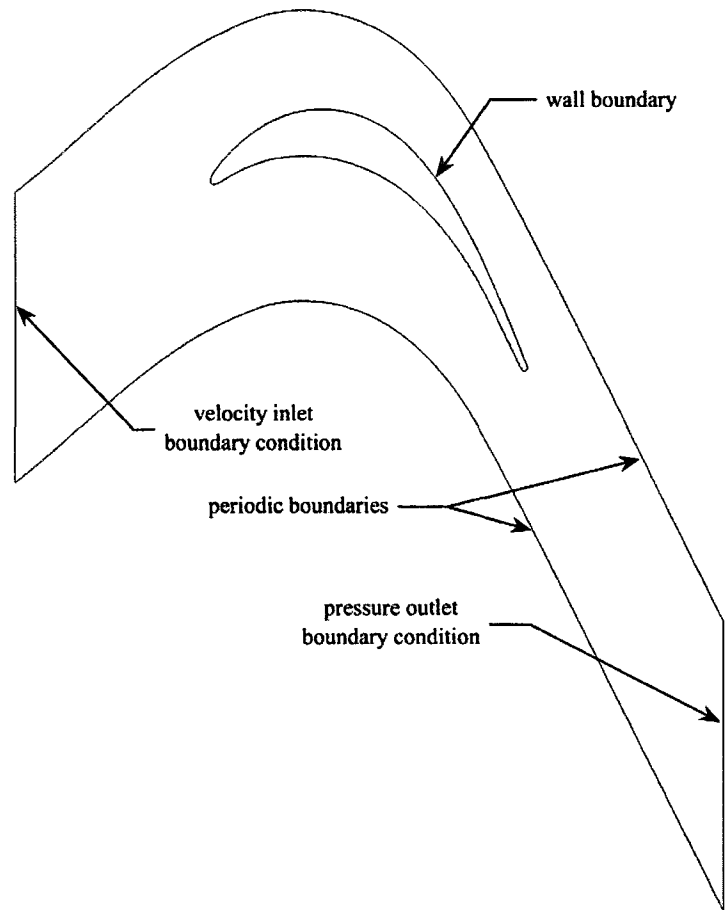


Figure 6.1: T106 computational domain with boundary conditions.

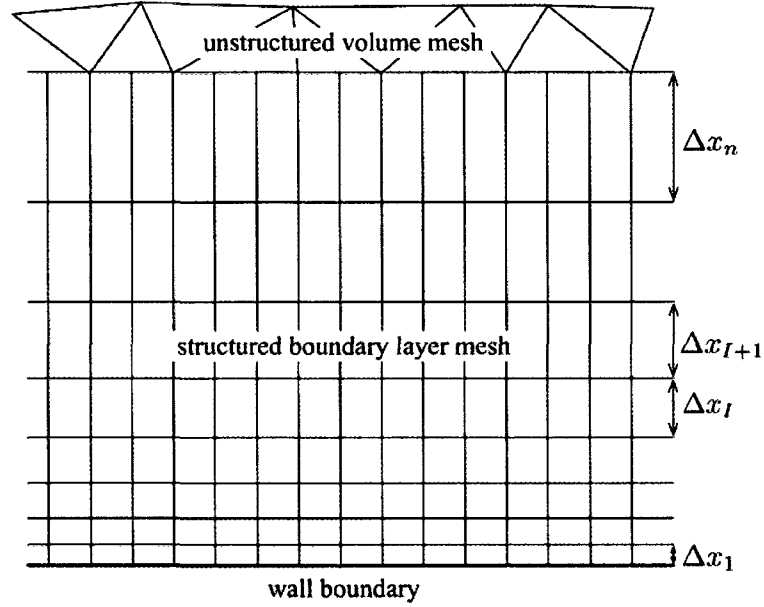


Figure 6.2: Schematic of the structured mesh in the near wall regions.

The T106 domain was meshed with a hybrid grid using GAMBIT 2.3.16. The unstructured portion consisted of triangular elements and the structured portion, in the near wall region as shown in Figure 6.2, consisted of 15 layers with a growth rate ($\Delta x_{I+1}/\Delta x_I$) of 1.2. The height of the first row was $\Delta x_1 = 0.01$ mm. The resulting y^+ value (i.e. the distance from the wall normalized by wall variables) never exceeded 0.4, which allowed the use of the enhanced wall treatment in FLUENT 6.3.26. Three meshes were developed for the T106 domain, all with the same structured mesh. They are shown in Figure 6.3. A grid independence study was conducted to determine which grid is adequate without incurring unnecessarily large computational costs.

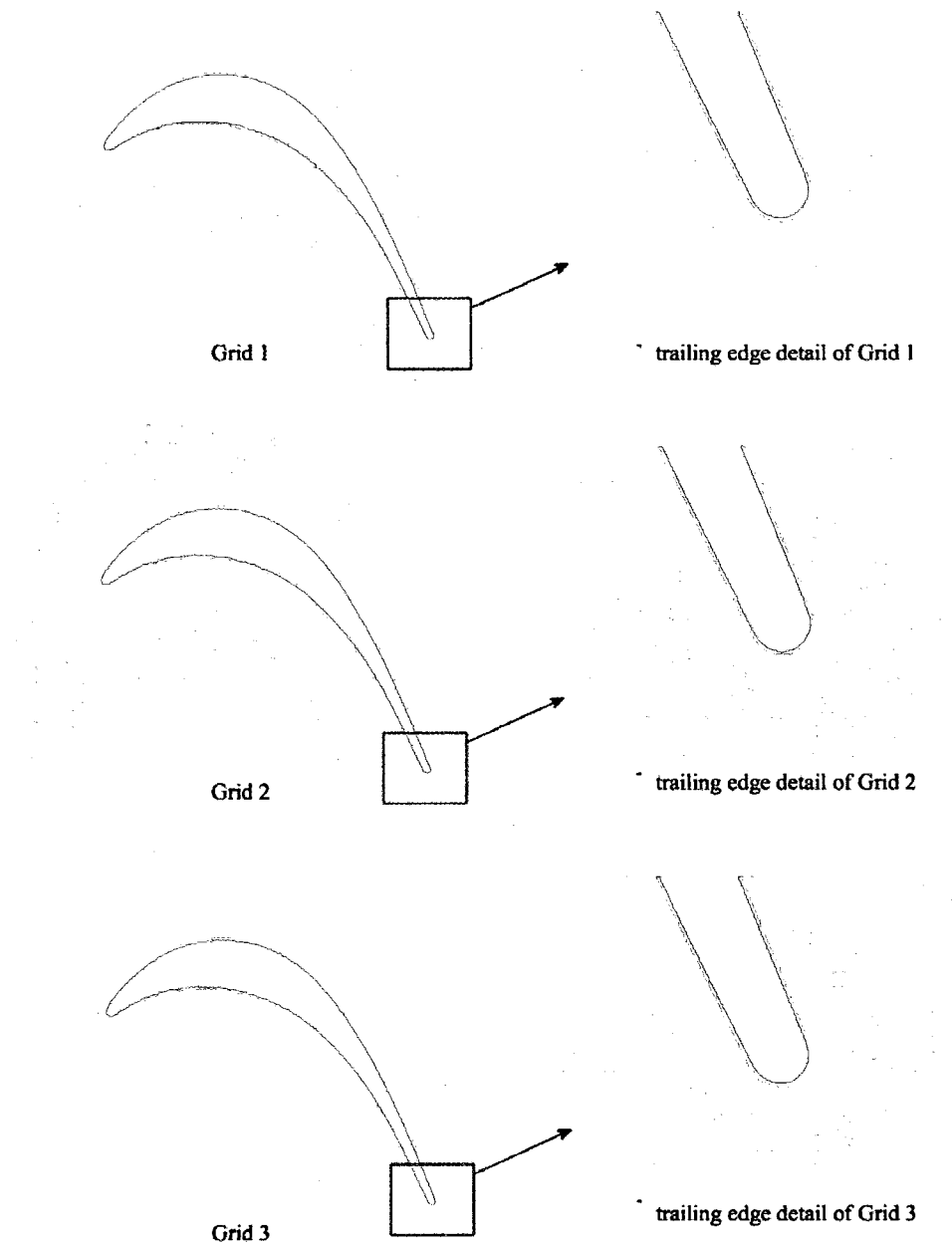


Figure 6.3: Various grids used to model the T106 cascade.

6.2.2 TTM Turbine

Because the TTM geometry is azimuthally periodic with a period equal to an 1/12th sector, only 1/12th of the full stage needed to be modelled. As a result, computational time requirements were much lower than those corresponding to modelling the full geometry.

The 3-D TTM model was developed using geometry data obtained from TTM and is shown in Figure 6.4. The rotor shape was generated using 5 profiles equally spaced from the hub to the tip. Similarly, the stator vanes were constructed with 2 profiles, one at the hub and another at the casing. In addition, the domain was initially split into three separate domains, namely the stator, rotor and diffuser sections for the grid independence study. However, after inconsistencies in the results were found with the initial meshes, the diffuser domain was removed to reduce the mesh requirements. Splitting the domain allowed the rotor domain to move while the stator and diffuser domains remained fixed. Rotational periodic boundary conditions were also used in order to reduce the size of the computational domain. However, the use of a sliding mesh in FLUENT 6.3.26 requires that the interface regions between adjacent domains be identical. Therefore, in this case, rotational periodic boundary conditions could not be applied to only one stator channel and one rotor channel, because this would result in a stator volume that is a 15° sector and a rotor volume that is a 10° sector. In order to obtain identical interface regions, 2 stator vanes and 3 rotor blades were required for the model. This resulted in a 30° sector for all domains or a reduction of the full turbomachine to 1/12th of the original size. The rotor domain also contains a gap region between the rotor blade tips and the casing.

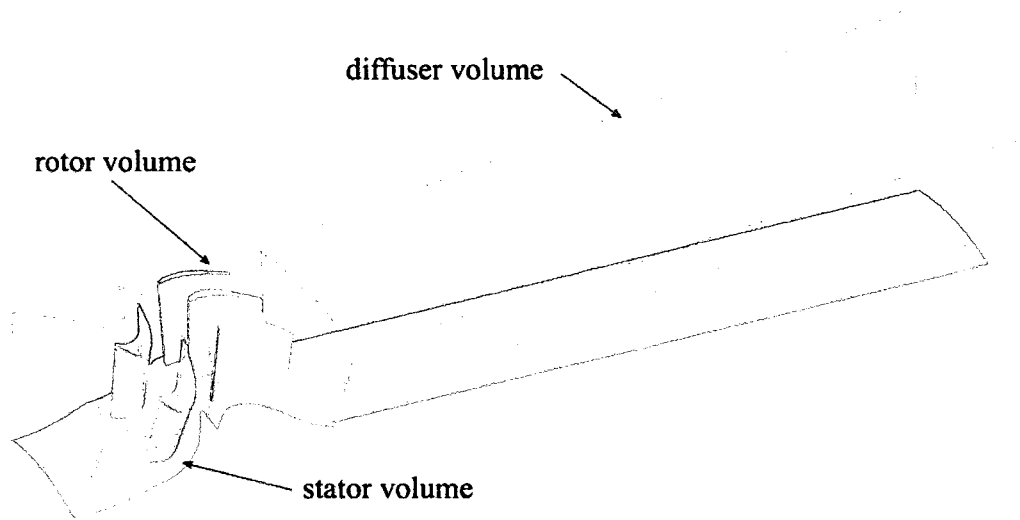


Figure 6.4: TTM model.

As for the T106 domain, a hybrid mesh was used to discretize the TTM domain. However, because the TTM domain is 3-D, the unstructured portion consisted of tetrahedral elements rather than triangular elements. GAMBIT 2.3.16 was unable to generate the required 3-D structured wall mesh. Therefore, an initial unstructured mesh was generated in GAMBIT 2.3.16. This mesh was imported into TGrid, that was used to generate a 3-D structured wall mesh. The 3-D structured wall mesh has a similar cross-section to the structured wall mesh shown in Figure 6.2. The TTM structured wall mesh consisted of 5 layers with a growth rate of 1.3. The height of the first layer was 0.005 mm. The reduced number of layers and an increased growth rate, with respect to the T106 model, were required to maintain a manageable mesh size without sacrificing a suitable y^+ value. The y^+ value, for the TTM model, was kept below 5 on the rotor blades and the stator vanes. This allowed the application of enhanced wall treatment in FLUENT 6.3.26. Three

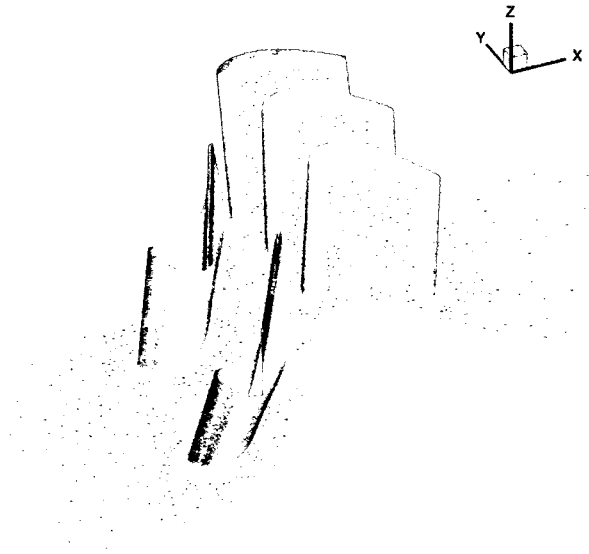


Figure 6.5: TTM surface mesh for grid 3.

meshes were generated for the TTM domain and are shown in Figures 6.5, 6.6 and 6.7 ranging from the coarsest to the finest.

As indicated previously the mesh was refined after the grid independence study. Figure 6.8 shows a comparison of the meshes before refinement (grid 2) and after refinement. The grid independence studies, which were independent of time, showed that grid 2 was adequate for the time varying runs. However this was not the case and the original mesh (before refinement) had to be refined along the flatter portions of the blade wall, as shown in Figure 6.8, in order to capture the time varying flow properties accurately.

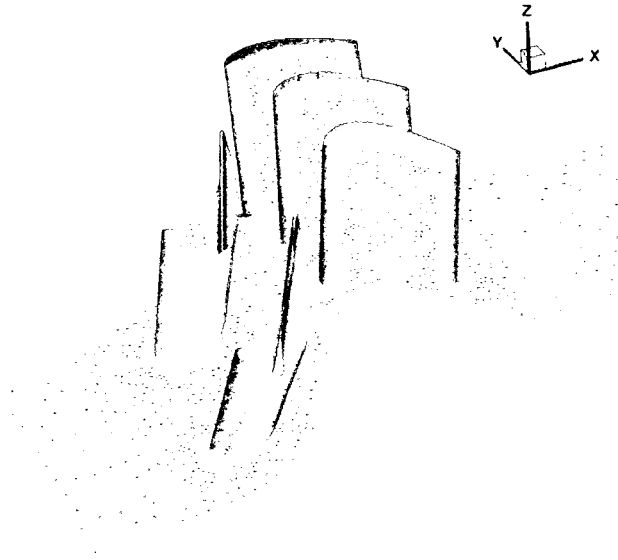


Figure 6.6: TTM surface mesh for grid 2.

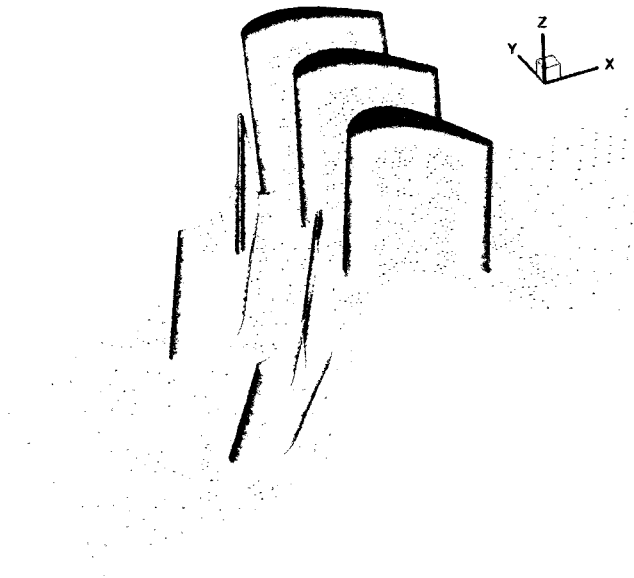


Figure 6.7: TTM surface mesh for grid 1.

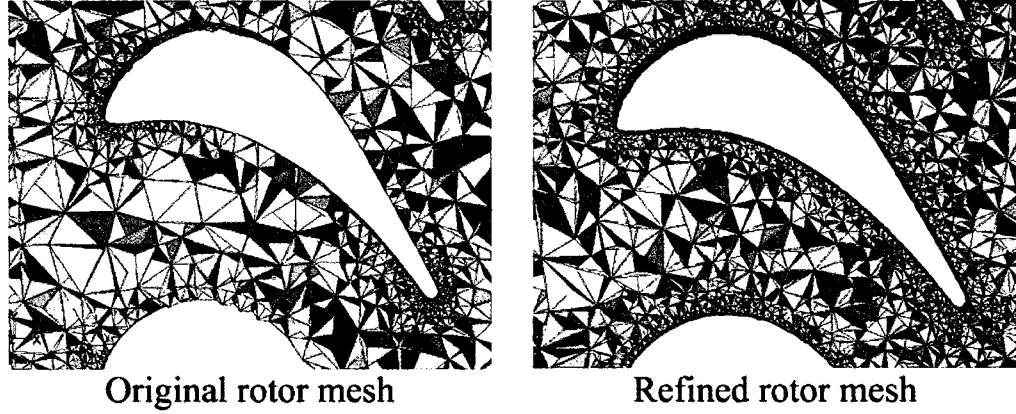


Figure 6.8: A comparison of the grid around the rotor blades for the original mesh (grid 2) and the refined mesh.

Rotor Blade Scaling

For the primary study, whose objective was to investigate the effects of scaling on turbomachinery performance, two scaled rotor geometries were simulated in addition to the unscaled case. For easy reference, the cases were given the abbreviated names TTM3:2 (unscaled case, used for verification and validation and as a reference for comparison of the scaled cases), TTM4:3 (with a rotor blade chord length increased by 12.5%) and TTM5:3 (with a rotor blade chord length reduced by 10%). Their characteristics are summarized in Table 6.1. The scaling was chosen so as to keep the scaling as low as possible without resulting in an excessively large domain that would require long simulation times. In all cases the number of stator vanes N_{st} remained constant, while the number of rotor blades N_{rt} varied.

The scaling ratio q was found by dividing the blade-vane ratio $\Lambda = N_{rt}/N_{st}$ in the TTM3:2 case by the blade-vane ratio in the case in question Λ_{sc} . To obtain the scaled

Table 6.1: Summary of cases investigated

Case	N_{rt}	N_{st}	Λ	q	Scaling
TTM3:2	36	24	3/2 (1.50)	1	unscaled
TTM4:3	32	24	4/3 (1.33)	1.125	+12.5%
TTM5:3	40	24	5/3 (1.67)	0.90	-10%

geometry, the tangential θ and axial a coordinates were multiplied by the scaling ratio

$$\theta_{sc} = q\theta \quad (6.1)$$

$$a_{sc} = qa \quad (6.2)$$

$$r_{sc} = r \quad (6.3)$$

The subscript *sc* indicated the scaled coordinates. The radial coordinates r were left unchanged, because scaling the rotor blades in the radial direction would alter the casing and hub locations. This would change the turbomachine flow path too drastically. Because the leading edge of the rotor blade profiles was not located at the origin of the global coordinate system in the data obtained from TTM, as shown in Figure 6.9, the scaled rotor blade had to be shifted in the axial and tangential directions so that the leading edge of the scaled blade was coincident with the leading edge (subscript *LE*) of the unscaled blade. The shifted coordinates (subscript *shift*) are

$$\theta_{shift} = \theta_{sc} + (\theta_{LE} - \theta_{sc,LE}) \quad (6.4)$$

$$a_{shift} = a_{sc} + (a_{LE} - a_{sc,LE}) \quad (6.5)$$

$$r_{shift} = r_{sc} = r \quad (6.6)$$

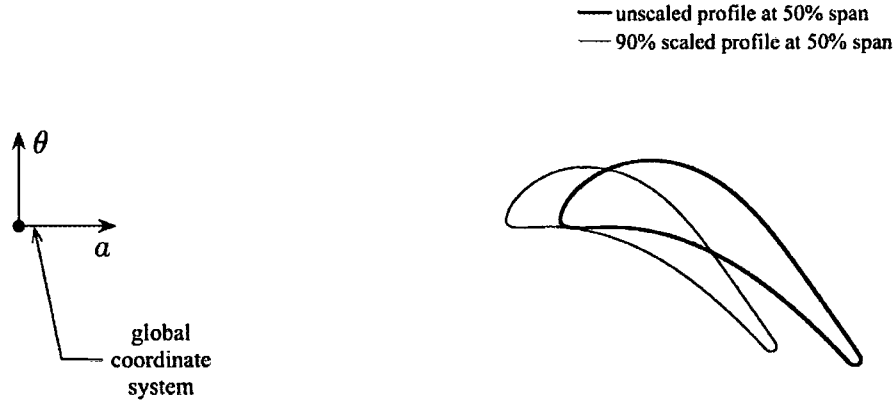


Figure 6.9: Global position of rotor profile data and effect of scaling on the scaled profile location.

In the case shown in Figure 6.9 the shift in the tangential direction was very small because the leading edge of the profile was located at a tangential position very close to 0. With the leading edges of the rotor blades coincident for all cases, the axial gap between the stator vane trailing edges and the rotor blade leading edges remained constant. Therefore, changes to the flow field were attributed to blade scaling only and not to a change in the axial gap spacing. Figure 6.10 shows the three cases after scaling and shifting.

As a result of scaling only in the tangential and axial directions, the rotor tip gaps differed considerably for the three cases, as shown in Figure 6.11a. Therefore, the tip profiles were adjusted for the scaled blades, such that the tip gaps at the leading edge and at the trailing edge were identical to those in the unscaled case. The results are shown in Figure 6.11b. This ensured that the tip leakage flow and tip vortex would not change due

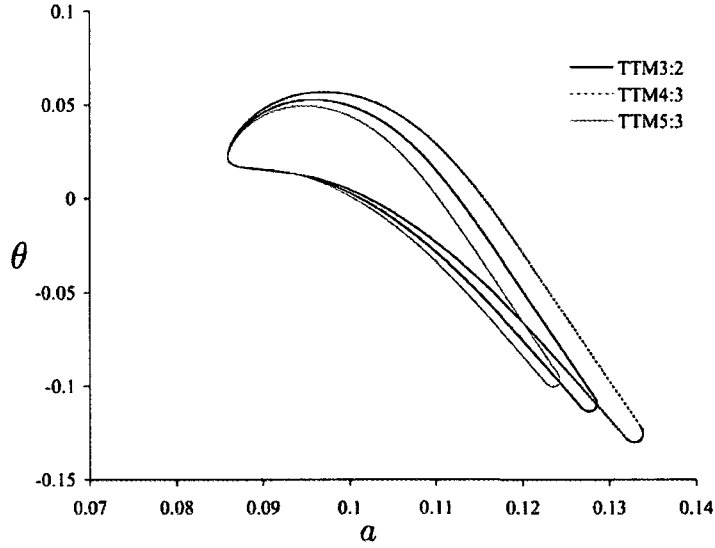


Figure 6.10: Profiles of rotor blade tips for the three cases, viewed in the spanwise direction from the casing to the hub.

a modified tip gap size, thus isolating the effect of scaling on efficiency and losses.

6.3 Computational Conditions

In the present studies, the interest focuses on the steady-state periodic solution, not on the start-up flow field. Therefore, the initial condition did not need to match the actual initial condition. However, the choice of initial condition is important as it affects the convergence rate and the overall success of the simulation. It was determined, after some trial and error, that an initial flow field that is approximately equivalent to the outlet conditions was required. This was especially the case for the initial pressure field, as it was found that the convergence of the simulation was highly sensitive to this choice. It was also determined that the mixing plane model was more sensitive to the initial field than the

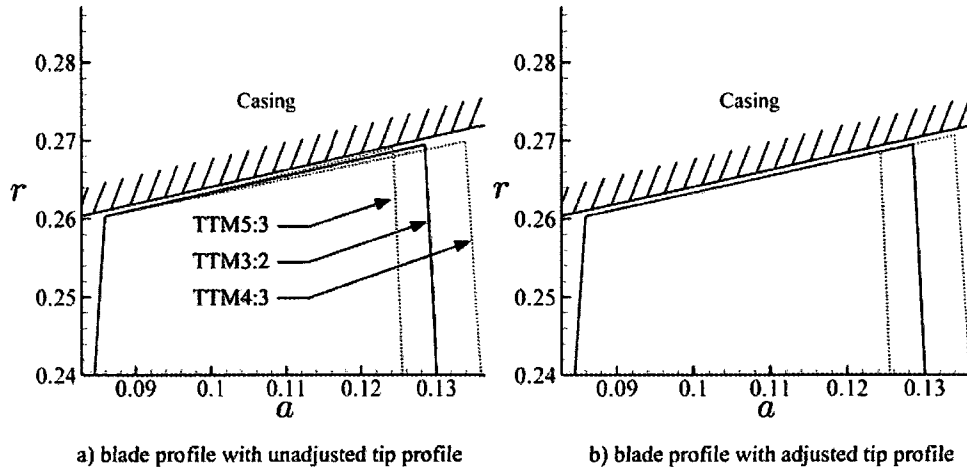


Figure 6.11: Profiles of rotor blades, viewed in the tangential direction and in the direction of rotation.

sliding mesh model.

6.3.1 T106 Cascade

Because the flow through the T106 cascade was incompressible, temperature and variable density effects were neglected. The working fluid for the T106 simulation was air with a density of 1.125 kg/m^3 and a viscosity of $1.7894 \times 10^{-5} \text{ kg/ms}$.

The simulation results and convergence rate were not as sensitive to the initial condition for the T106 case as for the TTM case. This is most likely due to the fact that the T106 case was an incompressible 2-D simulation whereas the TTM case was a compressible 3-D simulation. The initial conditions for the T106 case were calculated from the inlet conditions, shown in Table 6.2, using an internal program built into FLUENT 6.3.26.

Table 6.2: T106A simulation conditions

Inlet flow velocity U_{in} (m/s)	4.45
Inlet turbulence intensity (%)	0.1
Inlet turbulence length scale (m)	0.016
Outlet absolute pressure (Pa)	101325
Inlet Reynolds number based on chord $Re_{1,c}$	5.2×10^4
Exit Reynold number based on chord $Re_{2,c}$	1.6×10^5

The inlet velocity was determined by several iterations of CFD simulations until an inlet velocity was found, in which the outlet Reynolds number, from the simulation, matched the outlet Reynolds number from Stieger et al. (2003). In addition, the inlet turbulence intensity was not available from Stieger et al. (2003). As a result, the inlet turbulence intensity was adjusted until a trailing edge separation bubble was obtained that was similar to that observed in the experiments. This occurred at a turbulence intensity of 0.1%.

Periodic boundary conditions and other boundary conditions are shown in Figure 6.1. The blade surfaces were defined as smooth walls.

6.3.2 TTM Turbine

For the simulation of the TTM turbine, all material properties were assumed to be constant, with the exception of the density. This is acceptable, considering that Northall (2006) found that the application of variable gas properties has little effect on turbine performance prediction. To model compressible flow, the density was assumed to follow the ideal gas law

$$\rho = \frac{P}{\frac{\mathbf{R}}{MW}T} \quad (6.7)$$

Table 6.3: Gas properties for air

Specific heat C_p (J/kg K)	1006.43
Thermal conductivity κ (W/m K)	0.0242
Viscosity μ (kg/m s)	1.7894×10^{-5}
Gas constant R (J/K mol)	8.314
Molecular weight MW (kg/kg mol)	28.966

R is the gas constant; and MW is the molecular weight of the gas. The values of these and other properties for air are given in Table 6.3.

As stated earlier, the initial pressure was set close to the outlet pressure. Therefore, the initial absolute pressure was 110000 Pa, which was slightly higher than the outlet static pressure. The initial velocity in the x -direction was set to 5 m/s, which is non-zero but still close to stagnation conditions. The other two components of velocity were left at 0 m/s. The turbulence values were estimated from the velocity field to be $0.09375 \text{ m}^2/\text{s}^2$ for the turbulent kinetic energy and 34.128 1/s for the specific dissipation rate. The temperature in all domains was set to 300 K, which was equal to the estimated outlet temperature.

The main objective of these simulations was to determine whether the present approach can accurately predict the actual turbine operation, as evidence by an agreement of the numerical results with the experimental LDV data. Therefore, the boundary conditions in the simulations were required to match the experimental boundary conditions specified by TTM (summarized in Table 6.4). Nevertheless, some simplifications and assumptions were made to reduce computational costs.

The model was split into three domains in order to accommodate a rotating rotor, a stationary stator and a stationary diffuser. All domains were meshed separately and were connected together with a sliding mesh interface or a mixing plane. The stationary domains

Table 6.4: TTM simulation conditions

Pressure ratio $P_{0,in}/P_{out}$	3.5
Rotational speed (RPM)	10617
Inlet total temperature $T_{0,in}$ (K)	402.7
Nozzle guide vane exit Reynolds number	2.57×10^6
Rotor blade exit Reynolds number	1.69×10^6

were the stator and diffuser domains, while the rotor domain is rotating with a rotational velocity of approximately 10617 RPM (1111.8 rad/s).

The inlet boundary condition was a pressure-inlet condition 66 mm upstream from the leading edge of the stator vanes. The experimental value for the average total absolute pressure at the inlet was specified as 333200 Pa with a standard deviation of 95000 Pa. The experimental value for the average total temperature at the inlet was specified as 402.7 K with a standard deviation of 1.1 K by the TTM institute. In the present simulations, the inlet parameters were assumed to be constant across the inlet boundary. As, there were no specific inlet turbulence parameters given by the TTM, the inlet turbulence intensity was estimated to be 5% using the following relationship (Ansys (2006))

$$k_{in} = 0.16 (\text{Re}_{D_h})^{-1/8} \quad (6.8)$$

Some iterations of an incompressible simulation were required to approximately determine the inlet flow velocity. The hydraulic diameter D_h at the inlet was calculated to be 0.234 m. The turbulence intensity, along with the hydraulic diameter, were sufficient specifications for the inlet boundary conditions for the turbulence.

Initially, for the grid independence study, the outlet boundary condition was a pressure-outlet condition approximately 700 mm (approximately 15 midspan axial blade

chords) downstream from the leading edge of the stator vane. This length was equivalent to the length of the diffuser section in the experimental apparatus. After the diffuser section, the flow in the experimental apparatus proceeded into a large mixing chamber. Modelling of the mixing chamber would be time consuming work with no benefit to the present study. Therefore, the simulation model was terminated at the exit of the diffuser section. The average static absolute pressure was not known at this location, therefore the static pressure was estimated as being equal to the static pressure in the downstream mixing chamber, which was 95370 Pa. In addition, the pressure was assumed to be constant over the entire outlet boundary. This assumption seems to be appropriate, if one considers that Xiao et al. (2001) found that the flow field downstream from the rotor undergoes rapid mixing between the hub and tip vortices and, as a result, the pressure field is nearly uniform.

As stated previously, the diffuser section was removed for the refined mesh study. Therefore, in this case, the pressure-outlet condition was placed just behind the trailing edge of the rotor blades at P_{out} shown in Figure 5.6. A static pressure of 90 kPa (which was specified in the experimental parameters) was assigned along with a radial equilibrium assumption.

Three periodic boundaries were defined, one pair for each domain. These periodic boundaries were defined as rotational periodic boundaries with an axis of rotation coincident with the turbines axis. Figures 6.12 and 6.13 show the periodic boundaries with respect to the rest of the model.

The rotational periodicity was determined from the number of rotor blades and

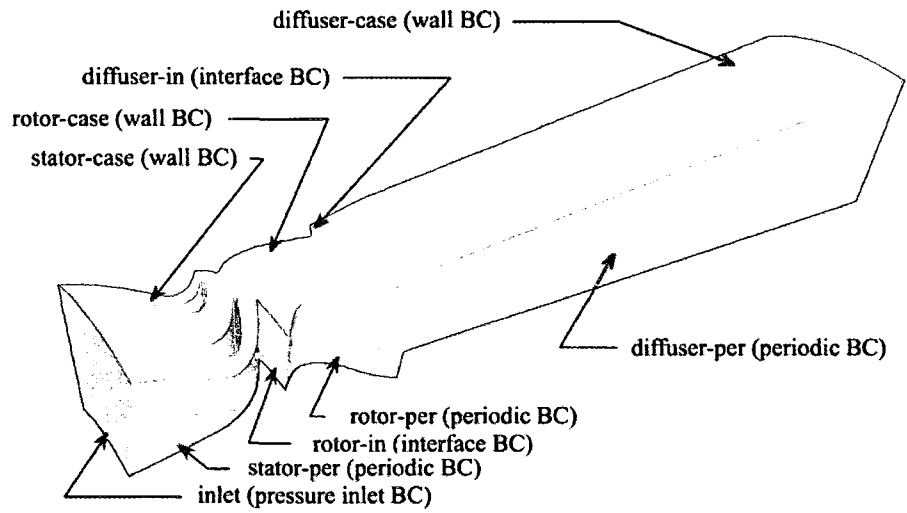


Figure 6.12: TTM boundary conditions.

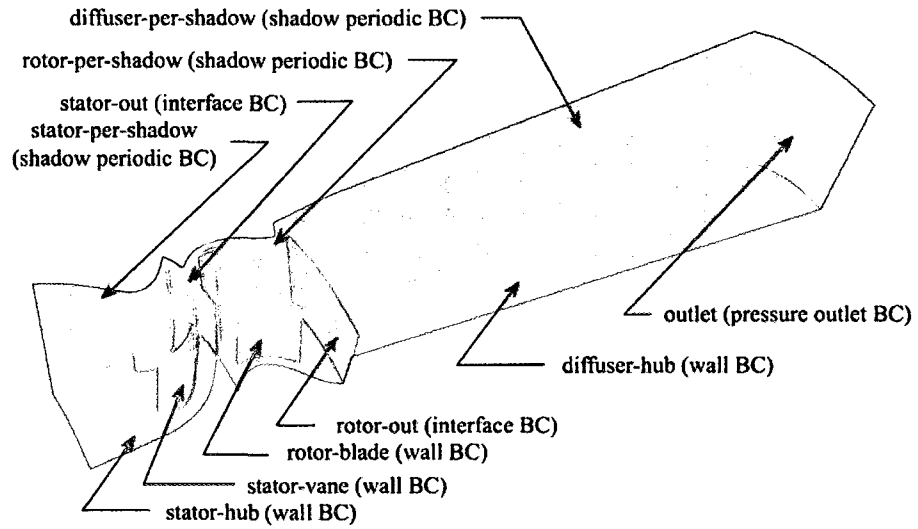


Figure 6.13: TTM boundary conditions.

the number of stator blades. In this case, the number of rotor blades was 36 and the number of stator blades was 24. The ratio of the number of rotor blades to the number of stator vanes is then given by

$$\Lambda = \frac{36}{24} = \frac{3}{2} \quad (6.9)$$

Accordingly, the TTM model contained 3 rotor blades and 2 stator vanes. The angle of rotational periodicity can be determined by

$$\theta_{rp} = \frac{360^\circ}{(N_{rt}/N_{rt,sim})} \quad (6.10)$$

where N_{rt} and $N_{rt,sim}$ represent the number of rotor blades in the actual and model turbines, respectively. In the case of the TTM turbine, there were 36 rotor blades in the actual turbine and 3 rotor blades in the model turbine, which gives a rotational periodicity of 30° .

In order to simulate the motion of the rotor, wall boundaries were assigned rotational speeds. The stator domain was given a rotational speed of 0 rad/s and all walls associated with that domain were assigned the same rotational speed. The rotor domain was assigned a rotational speed of 1111.8 rad/s. However, in the initial runs not all associated walls were assigned the same rotational speed. Three wall boundaries are shown in Figure 6.14. The boundaries rotor-hub1 and rotor-hub3 were assigned a global rotational speed of 0 rad/s, while the boundaries rotor-hub2, rotor-tip and rotor-blade were assigned a rotational speed of 1111.8 rad/s. For the refined mesh study, the rotor hub was not split into 3 distinct surfaces. In this case, the entire rotor hub (or all 3 hub surfaces in Figure 6.14) was assigned a rotational speed of 1111.8 rad/s. The casing wall boundary, which is

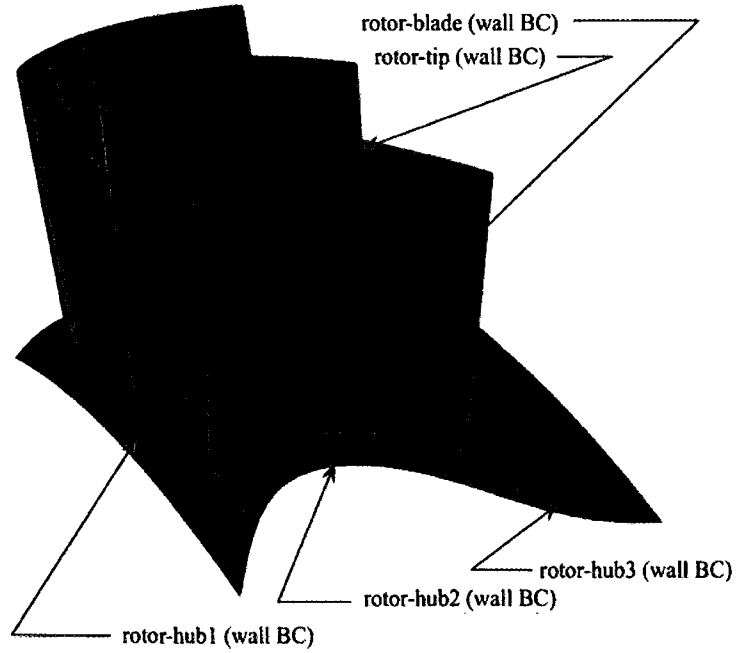


Figure 6.14: TTM rotor boundary conditions.

not shown, was assigned a global rotational speed of 0 rad/s for all simulations. In addition to the above boundary conditions, the walls were assigned boundary conditions, which were applied to the energy equation. As no combustion occurs, the temperature of the air flowing through the turbine is significantly lower than what it would be in operating gas turbines. In the present case, heat transfer with the surroundings is expected to be very small. For simplicity, the wall boundary conditions (wall BC in Figures 6.12 and 6.13) were assumed to be adiabatic.

6.4 Numerical Schemes

In order to achieve high accuracy and to eliminate artificial diffusion associated with first-order spatial discretization, a second-order upwind scheme was used for spatial discretization of all flow variables (Barth and Jespersen (1989)). Gradients and derivatives were calculated using the *Green-Gauss Cell-Based* method. The initial *Courant-Friedrichs-Lewy number* ($CFL = u\Delta t/\Delta x$), which is the ratio of the time step (the internal solver time step between solution iterations at a given time) and the cell residence time, for all problems was 0.0001. This was then gradually increased, in a pattern shown in Table 6.5 (see next section), until convergence was obtained. The simulations were found to become unstable when the CFL number exceeded 0.1, if the explicit solver were used, and when it exceeded 1, if the implicit solver were used.

In the case of unsteady simulations, a second-order implicit scheme was used for time discretization, as this scheme is known to be unconditionally stable. With the explicit formulation of the density-based solver, the second order implicit scheme for time discretization (dual-time formulation) employed a second-order Euler backward differencing scheme. The inner iterations (iterations within each time step) employed explicit pseudo-time marching with a three-stage Runge-Kutta scheme.

6.4.1 T106 Cascade

In this case, the pressure-based solver in FLUENT 6.3.26 was used, as it applies to incompressible flows. The pressure based solver solves each governing equation separately. This reduces the required computational resources, however, it is more prone to divergence.

This instability is more noticeable in compressible than incompressible flows. Pressure-velocity coupling was accomplished through a *SIMPLEC* (Semi – Implicit Pressure Linked Equations - Consistent) algorithm (Vandoormaal and Raithby (1984)).

6.4.2 TTM Turbine

For compressible flow problems, the explicit formulation of the density-based solver was used in FLUENT 6.3.26. The density-based solver works with all governing equations simultaneously. This greatly increases the computational requirements, however, this solver is required for compressible flow problems. Convective fluxes in the density-based solver were evaluated using the *Roe-FDS* (Flux-Difference Splitting) scheme (Roe (1986)).

In the explicit formulation, unknown quantities in each cell were determined using formulations that use only known values. This eliminates the need for a linear equation solver, and a Runge-Kutta solver is used instead. In addition, the explicit formulation allowed for the use of convergence acceleration methods. In basic terms, the explicit formulation solves for all flow parameters in each cell separately, whereas the implicit formulation solves for all flow parameters in all cells simultaneously. Convergence acceleration methods that were implemented included 5 levels of *Full-Approximation Storage* (FAS) multigrid and 2 iterations of residual smoothing with a smoothing factor of 0.5. *Algebraic Multigrid* (AMG) was always implemented and it is known to be beneficial for unstructured meshes, as no coarse grids are generated or stored, thus reducing computing time. However, the FAS multigrid method performs better when non-linearities are present, because the non-linearities are maintained on each grid level through re-discretization. The FAS multigrid method was found to greatly improve convergence rates when compared to simulations with

no FAS multigrid method implemented.

6.5 Simulation Methodology

To facilitate the solution of potentially difficult problems, such as the TTM simulation, FLUENT 6.3.26 contains the *Full MultiGrid* (FMG) initialization procedure, which was used presently in order to provide a starting solution. After setting all simulation parameters and boundary conditions and initializing the solution, the simulation must be set to steady state, regardless of whether it is steady or unsteady. Following FMG initialization, if the simulation is unsteady, it can then be run in the unsteady mode. Such initialization can reduce the computation time by weeks and results in a more stable simulation.

Once the FMG initialization was completed, the CFL number was set to 0.0001 and the time step was set to 2.355×10^{-6} s (100 time steps per stator passage) to decrease the required computational time. Temporal and spatial discretization were set to second order. Following the findings in the T106 study, the RNG $k - \varepsilon$ turbulence model was used during the initial stages of the simulation and was later switched to the SST model. For the TTM study, the simulation is unsteady, and therefore it was run until a periodic trend was observed in the power produced by each rotor blade. This typically took 2 to 5 days of running, and it was found that the solution did not always converge.

After the solution was observed to approach a periodic state, the time step was reduced in a pattern as shown in Table 6.5. After each time step reduction, the simulation was run until no significant reduction in the residuals was noticeable between consecutive time steps (this took typically 1 to 2 days). The CFL number was kept constant until a

Table 6.5: Simulation parameters

Interval	Time step (μs)	CFL	Solver
0	2.355	0.0001	Explicit
1	0.785	0.0001	Explicit
2	0.3925	0.0001	Explicit
3	0.19625	0.0001	Explicit
4	0.19625	0.0005	Explicit
5	0.19625	0.001	Explicit
6	0.19625	0.005	Explicit
7	0.19625	0.01	Explicit
8	0.19625	0.05	Explicit
9	0.19625	0.1	Explicit or Implicit
10	0.19625	0.5	Implicit only
11	0.19625	≤ 1	Implicit only

time step of 1.9625×10^{-7} s (1200 time steps per stator passage) was obtained. The final time step of 1.9625×10^{-7} s was chosen, as it was found through trial and error that this time step yielded a stable simulation and predicted the anticipated coherent structures.

After the final time step was obtained, the CFL number was increased until the solution converged. The CFL number was never increased by more than 0.1 when the explicit solver was used to maintain solution stability. A overview of the simulation methodology is shown in Table 6.5. The initial simulation conditions are given at interval 0 and the temporal and spatial discretization schemes were of second-order for all intervals.

6.6 Simulation Uncertainty

Uncertainty and error in CFD are strongly dependent on the code, computers, and models used to run the simulations. For clarification, the *American Institute of Aeronautics and Astronautics* (AIAA) defines uncertainty and error as

Uncertainty A potential deficiency in any phase or activity of the modeling process that is due to a lack of knowledge (AIAA (1998))

Error A recognizable deficiency in any phase or activity of modeling and simulation that is not due to a lack of knowledge (AIAA (1998))

Simulation uncertainty and error can be divided into two basic groups; *acknowledged* and *unacknowledged* uncertainty and error. Acknowledged uncertainties and errors have methods that can be used to identify them and possibly eliminate them. Unacknowledged uncertainties and errors can not be found or eliminated. A further breakdown of simulation uncertainties is (Faragher (2004))

- Acknowledged uncertainties and errors
 - *Physical approximation uncertainty*
 - *Computer round-off error*
 - *Iterative convergence error*
 - *Discretization errors*
 - * *Spatial discretization error*
 - * *Temporal discretization errors*
 - * *Truncation errors*
- Unacknowledged uncertainties and errors
 - *Computer programming errors*
 - *Usage errors*

Physical approximation uncertainty deals with uncertainties due to assumptions that are made during the generation of the model. Simplification to the geometry used for the simulations, turbulence modeling, simplifications to boundary conditions and neglecting heat transfer are all examples that introduce physical approximation uncertainty. An estimate of physical approximation uncertainty is possible through validation of the simulation using experimental results (Faragher (2004)). However, quantitatively validating all local flow parameters in an unsteady simulation, such as in turbomachinery, is very difficult due to the large volume of data. It is more appropriate to quantitatively validate global flow parameters, such as efficiency or power. Some reasons for physical approximation uncertainty are (Mehta (1996))

1. Simulation flow is not fully understood.
2. Model parameters are known only to some degree of certainty.
3. Models are simplified.
4. Validation of the simulation is not possible due to lack of experimental data.

Computer round-off error develops as a result of a computer's inability to represent floating point numbers to an infinite number of decimal places. In the case of a computer that stores floating point numbers with 32 bits and single precision, which is typical of the computers used for the simulations covered in this study, numbers can only be stored with an accuracy of 1×10^{-7} . Round-off error is typically neglected because it is usually insignificant when compared to other uncertainties.

Iterative convergence error exists because the iterative process can never obtain a perfect solution. A criterion for stopping the simulation must be always adopted. In FLUENT 6.3.26, the stopping criterion is

$$\frac{\Phi(g)}{\Phi_{\max}(g)} < 1 \times 10^{-3} \quad (6.11)$$

$$\Phi(g) = \sqrt{\frac{\sum_{cells} \left(\frac{\partial g}{\partial t}\right)^2}{m_{cells}}} \quad (6.12)$$

where $\Phi(g)$ is the *residual* for a given flow parameter g . For the density based solver in FLUENT 6.3.26, the residual is a RMS average, over all cells, of the rate of change of any flow parameter during one internal iteration (the iterative process during one time step). The stopping criterion is determined by scaling the residual by the maximum residual in the first 5 iterations $\Phi_{\max}(g)$. The FLUENT 6.3.26 default scaled residual of 1×10^{-3} was chosen because further reduction in the scaled residuals was not possible due to time constraints and 1×10^{-3} is the maximum scaled residual required for a converged solution (Menter (2002)).

Discretization errors occur as a result of solving the governing equations on discrete points in space and time and are considered to be the most important errors in CFD simulations. Spatial discretization error exists because the simulation domain is meshed with a finite number of grid points or cells, whereas temporal discretization error exists due to the application of a time step which dictates the time between instantaneous solutions in unsteady simulations. Discretization errors can be reduced by increasing the number of grid points in spatial discretization or by reducing the time step size for temporal discretization.

As the spatial and temporal discretizations are refined, the solution becomes less sensitive to spatial and temporal discretizations. Once the sensitivity reaches 0, the spatial and temporal discretizations are said to be converged. If the spatial and temporal discretizations were refined such that they were infinitely small, then the simulation should, theoretically, approach the continuum solution and have no discretization error. However, this is not possible in practice due to the complexity of the governing equations and limitations in computational power. Therefore, an important aspect of any simulation is an appropriate grid convergence study to determine the discretization errors.

Another aspect of the discretization errors is the truncation error. Truncation error is the difference between the non-discretized governing equations, which a computer is incapable of solving, and the finite or discretized governing equations, which a computer can solve. This error is affected by the grid quality, localized flow features (such as shocks or boundary layers) and the order of the method used to discretize the governing equations. If, during discretization of the governing equations, the lowest order term that is truncated is of first order (namely, contains Δx as in Equation 6.13) then the method is “*first-order accurate*”, whereas, if the lowest order term that is truncated is of second order (contains Δx^2 as in Equation 6.14) then the method is “*second-order accurate*” (Faragher (2004)).

$$\frac{dg}{dx} = \frac{g_{i+1} - g_i}{\Delta x} + \underbrace{\frac{d^2g}{dx^2} \frac{\Delta x}{2} - \frac{d^3g}{dx^3} \frac{\Delta x^2}{6}}_{\text{truncated terms}} + \text{higher order terms} \quad (6.13)$$

$$\frac{dg}{dx} = \frac{g_{i+1} - g_{i-1}}{2\Delta x} - \underbrace{\frac{d^3g}{dx^3} \frac{\Delta x^2}{6}}_{\text{truncated terms}} + \text{higher order terms} \quad (6.14)$$

In first-order accurate methods, the leading term of the truncation error is an even order derivative and as a result the solution will show effects of *numerical dissipation* or *artificial viscosity* (Faragher (2004)). Numerical dissipation tends to increase the actual viscosity of the fluid and smoothen out gradients. While numerical dissipation does lead to inaccuracies in the result, it also tends to stabilize the solution so that, sometimes it is a positive attribute (Faragher (2004)). Second-order accurate methods have a truncation error with an odd-order derivative, which means that the solution will show effects of *numerical dispersion* that generates oscillations in the solution (Faragher (2004)). The combination of numerical dispersion and numerical dissipation is sometimes referred to as *numerical diffusion* (Faragher (2004)). The simulations conducted in this study were performed using second-order methods in both the spatial and temporal domains to achieve high accuracy. However, first-order methods were used in the initial stages of the simulation to take advantage of their stabilizing characteristics.

Computer programming errors are errors that are the result of mistakes made while writing the code for a CFD program. These errors can only be eliminated by the programmer. In a long-established commercial code, such as FLUENT 6.3.26, these errors are assumed to be negligible.

Usage errors occur due to improper use of a CFD code. This may occur during geometry or grid generation or during the simulation itself. Some examples of usage error include simulating turbulent flow with a laminar model or simulating compressible flow while assuming that the density is constant. The probability of the occurrence of these errors increases as the number of options increases in a CFD code. However, it is the author's

belief that, because of his careful study of modeling assumptions, the probability that such errors are present in these simulations would be negligible. This is further supported by the validation of the simulation results with experimental results.

Chapter 7

Preliminary Simulations

Preliminary simulations aimed at selecting the most suitable conditions and options for the computational models were carried out using the 2-D T106 low-pressure blade cascade, described by Stieger et al. (2003). The cross-section of a single blade is shown in Figure 7.1, which also shows the inlet velocity U_{in} , inlet and exit flow angles, forces acting on the blade and the blade axial chord c_a .

The experimentally found coefficient of tangential force for the T106 turbine blade was 3.29, and the experimentally determined locations of separation and reattachment of the separation bubble were $x/c_a = 0.75$ and $x/c_a = 0.88$, respectively. No experimental values are available for the coefficient of axial force.

During simulations with the various turbulence models, it was found that the SST and RSM (with the ω equation) models were extremely sensitive to the initial conditions of the flow. During early tests, the simulations using the SST and RSM models predicted much lower turbulent kinetic energy throughout the flow domain than those using the S-A,

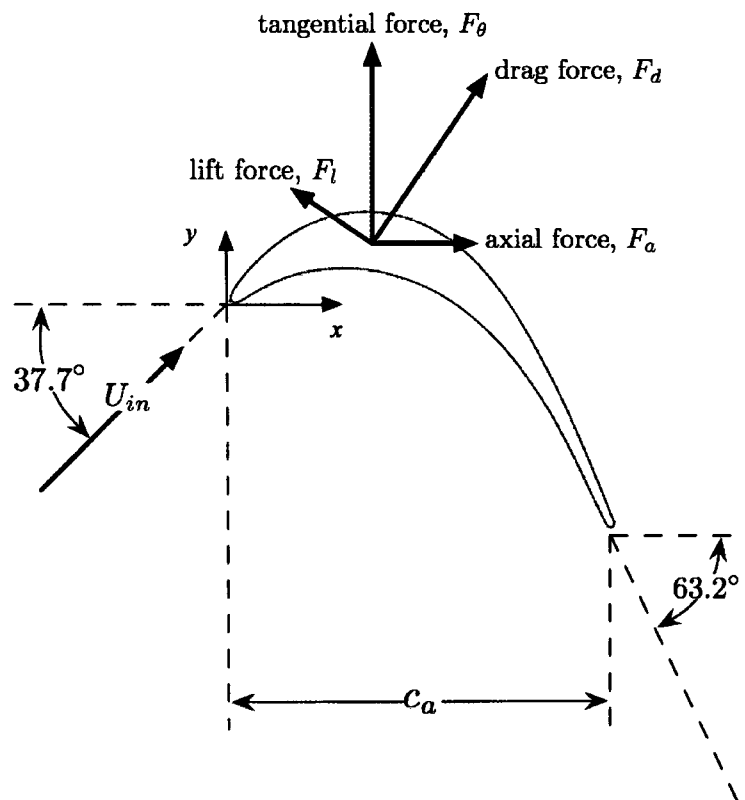


Figure 7.1: Sketch of the T106 turbine blade with applicable forces.

realizable $k-\varepsilon$ and RNG $k-\varepsilon$ turbulence models. This large discrepancy seemed unrealistic and prompted further study. Simulations were started with the RNG $k-\varepsilon$ turbulence model before switching to the SST or RSM models. This procedure was adopted in recent simulations. Having repeated several simulations following this approach, it was found that the turbulent kinetic energy predicted by the SST and RSM models were at acceptable levels, unlike those found in the earlier studies.

The terms *verification* and *validation* are used to identify two essential steps of any numerical simulation process. Verification deals with “solving the equations right” and determining whether the discretization is adequate, i.e., whether the mesh is sufficiently fine not to introduce unacceptably large errors (Roache (1998)). Validation involves the comparison between the numerical solution and experimental results to ensure that one is “solving the right equations” (Roache (1998)). A numerical code can not be validated as a whole, and each numerical simulation of a specific problem, or at least representative ones, need to be validated by comparison to results obtained by experiments or proved theoretically.

7.1 Simulation Verification

To examine the dependence of the solution on the computational grid density, simulations were performed using three different meshes (shown in Figure 6.3) using the SST turbulence model. The numbers of nodes m of the three grids increased by a factor of approximately $\sqrt{2}$ from one grid to the next. The number of cells m_{cells} , and the average cell size d is also shown. The variables used to compare the results using different grids

Table 7.1: Predicted time-averaged parameters for the different grids using the SST turbulence model

	m	m_{cells}	d (mm)	C_θ	C_a	L_{se}/c_a	L_{re}/c_a
Grid 3	11713	15958	1.86	3.10	1.93	0.84	0.93
Grid 2	16387	25306	1.48	3.10	1.94	0.82	0.91
Grid 1	26163	44858	1.11	3.11	1.95	0.80	0.90
experiment	-	-	-	3.29	-	0.75	0.88

were the coefficient of tangential force $C_\theta = 2F_\theta/\rho_{in}U_{in}^2c_aS$, the coefficient of axial force $C_a = 2F_a/\rho_{in}U_{in}^2c_aS$ and the relative locations of separation L_{se}/c_a and reattachment L_{re}/c_a of the separation bubble on the suction side of the blade. The grid sizes, as well as some simulation results, are shown in Table 7.1.

The computed values of the tangential and axial force coefficients for all three grids were nearly identical. However, as the grid density increased, the predicted locations of separation and reattachment shifted upstream, namely closer to the experimentally observed separation and reattachment points. Figure 7.2 also shows that, as the grid density increased, the wall shear stress distribution became smoother and the observed fluctuations became less unrealistic.

The error analysis for the T106 study, shown in Table 7.2, was conducted using the AES method, previously discussed in section 3.2. The analysis shows that the estimated error bands for grid 1 and grid 2 were lower than approximately 6% and 9%, respectively. In addition, the separation location tends towards the experimental value as the number of nodes increases. Therefore, to keep the error band as small as possible, grid 1 was chosen for the turbulence model study. Although grid 1 has roughly 60% more nodes than grid 2, this was not of considerable concern, as the difference in computational times between simulations with grids 1 and 2 was insignificant.

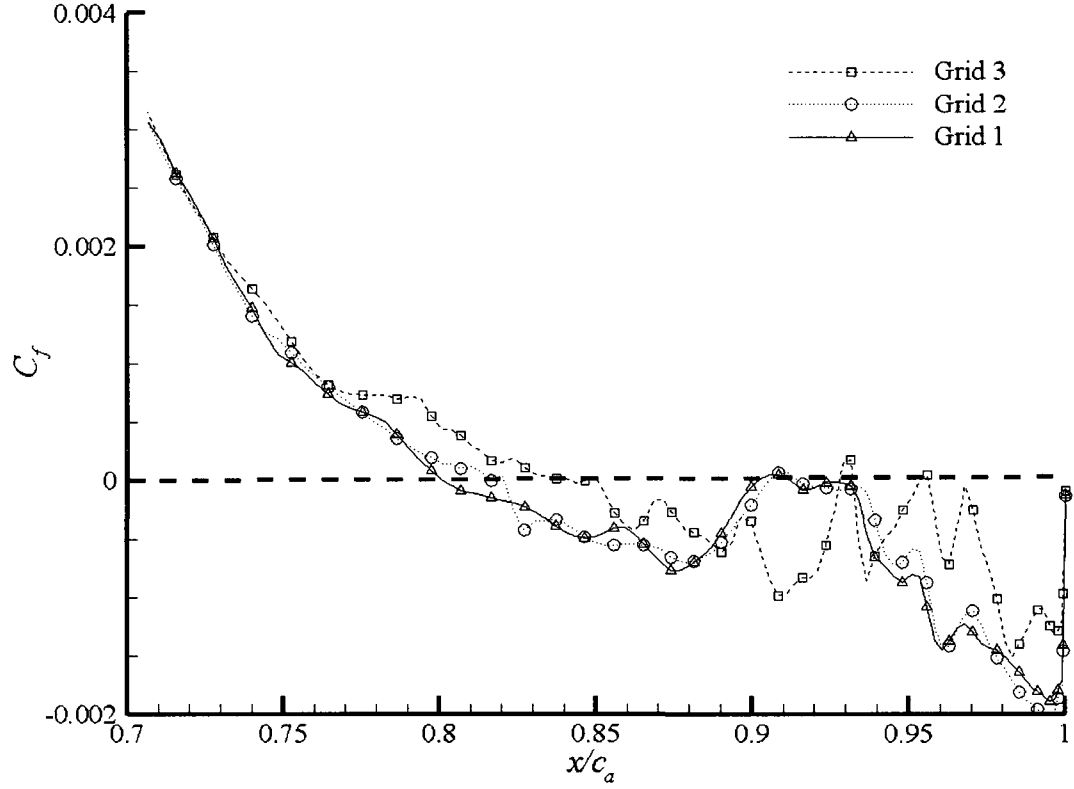


Figure 7.2: x -component of the coefficient of wall shear stress $C_f = 2\tau_{w,x}/\rho_{in}U_{in}^2$ on the suction side of the T106 blade for various grid densities ($\tau_{w,x}$ is the x -component of the wall shear stress).

Table 7.2: Error estimates for the T106 study

	Grid 3	Grid 2	Grid 1	GCI ¹	GCI ²
C_θ	3.10	3.10	3.11	1.69%	2.10%
C_a	1.93	1.94	1.95	1.83%	2.47%
L_{se}/c_a	0.84	0.82	0.80	6.07%	8.73%
L_{re}/c_a	0.93	0.91	0.90	3.19%	2.91%

Table 7.3: Comparison of tangential and axial force coefficients predicted using different turbulence models

	C_θ	C_a	Difference in C_θ
experiment	3.29	-	-
S-A	3.13	2.01	4.86%
realizable $k - \varepsilon$	3.13	2.03	4.86%
RNG $k - \varepsilon$	3.13	2.04	4.86%
SST	3.11	1.95	5.47%
RSM	3.09	1.91	6.08%

7.2 Turbulence Model Validation

When results using the various turbulence models were compared, all models predicted values of the coefficient of tangential force that were approximately 5% lower than the experimental value (Table 7.3). It was also found that the S-A, the realizable $k - \varepsilon$ and the RNG $k - \varepsilon$ turbulence models were not able to predict the separation bubble, whereas the SST and RSM turbulence models were able to do so, as shown in Figure 7.3. This is most likely due to stronger turbulence generation and mixing in the separation region, which occurred when the S-A or one of the two $k - \varepsilon$ turbulence models were used.

Figure 7.4 shows contour plots of the turbulent kinetic energy, predicted with the use of various turbulence models. The realizable $k - \varepsilon$ and RNG $k - \varepsilon$ turbulence models predicted higher turbulence kinetic energy, compared to those predicted by the SST and RSM turbulence models, along the suction surface of the blade and near the trailing edge. This is a common result when $k - \varepsilon$ turbulence models are used. However, directly behind the trailing edge the magnitudes of turbulent kinetic energy were comparable for all models. Results for the S-A turbulence model are not shown, because turbulence kinetic energy is not used in this model.

Figure 7.5 shows the measured and predicted variations of the coefficient of surface

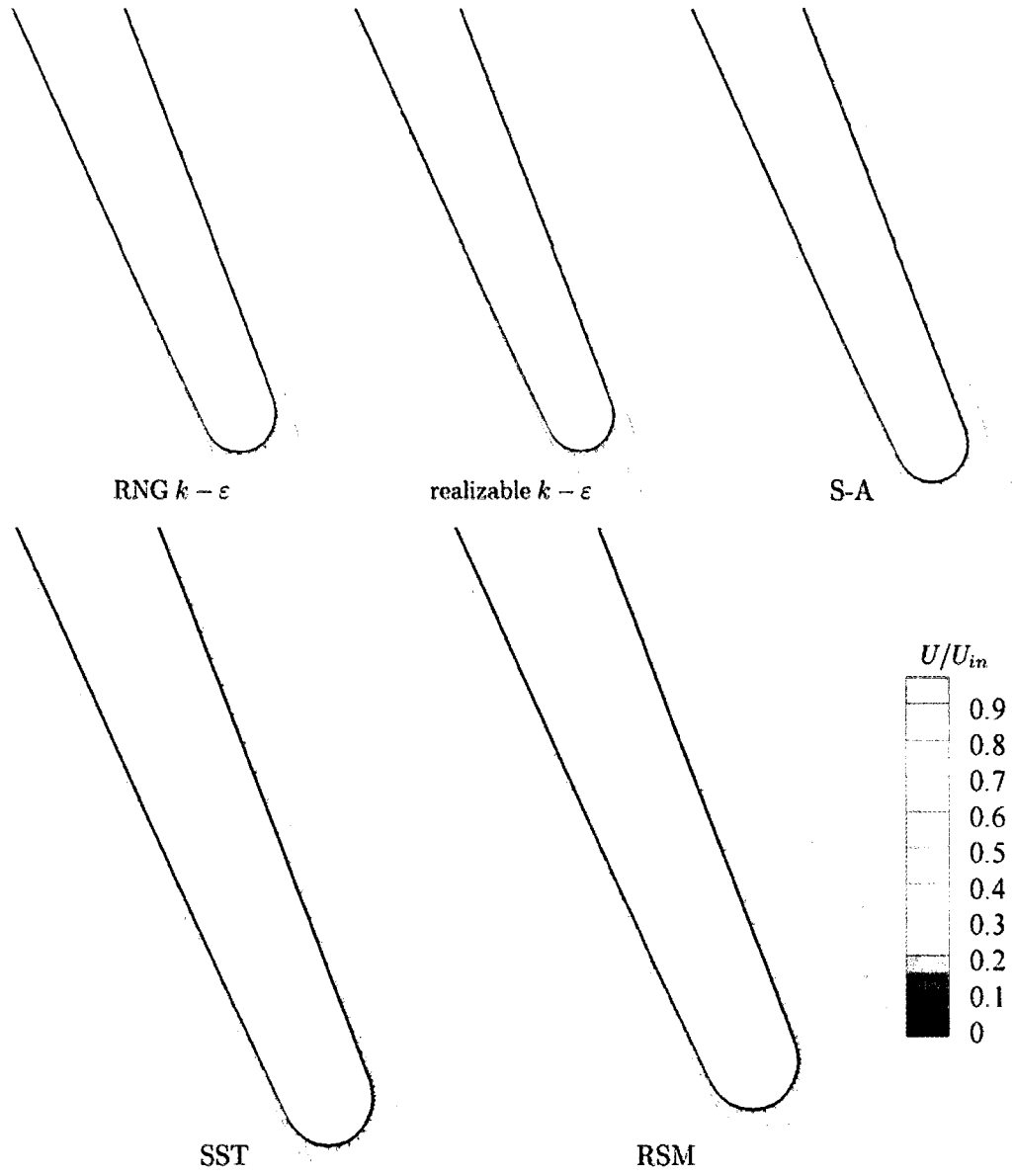


Figure 7.3: Vector plots, focused on the trailing edge of the T106 turbine blade. Vector colours correspond to the values of the non-dimensionalized velocity magnitude U/U_{∞} .

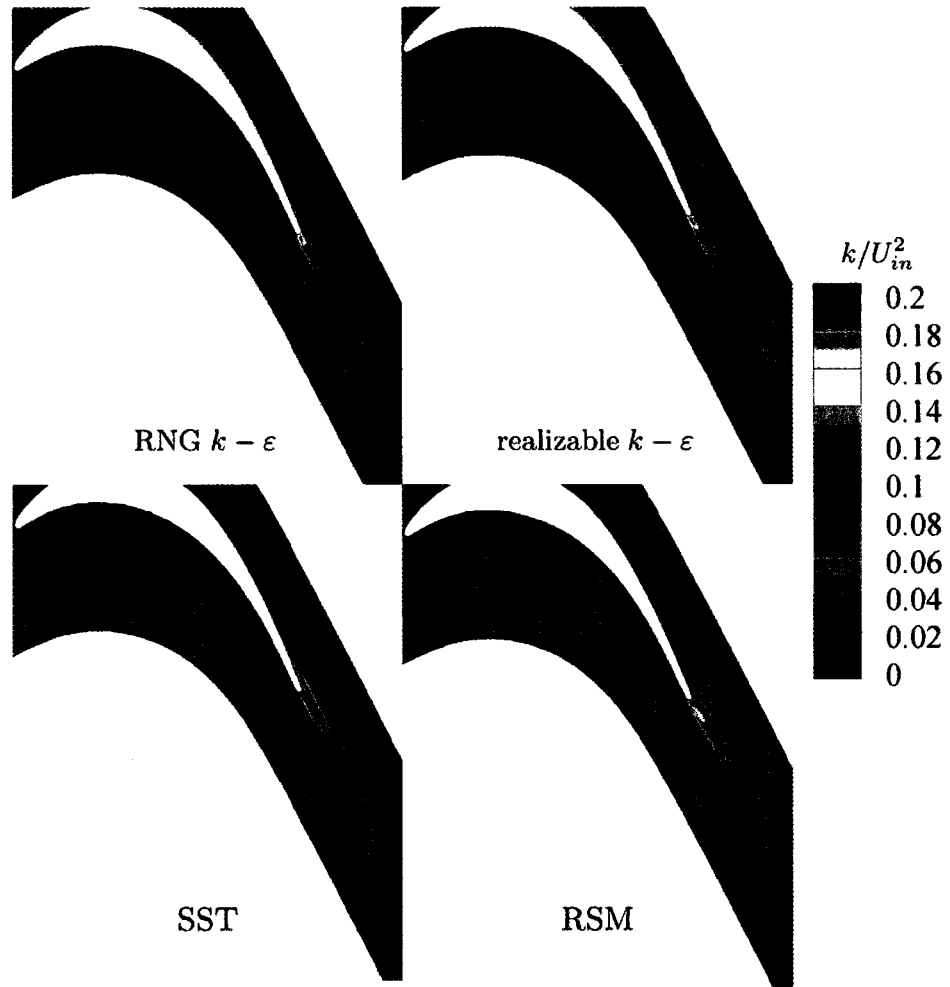


Figure 7.4: Non-dimensionalized time averaged turbulent kinetic energy k/U_{in}^2 near the blade trailing edge.

pressure $C_p = (P_0 - P) / (P_0 - P_{2s})$ along the T106 turbine blade. An important region to consider is the separation bubble region, which, in the experiments, starts at $x/c_a = 0.75$ and reattaches at approximately $x/c_a = 0.88$. Figure 7.6 shows a detailed view of the pressure variation near the trailing edge of the T106 turbine blade.

On the pressure side, the pressure variation in the range $0 < x/c_a < 0.80$ is captured adequately by all turbulence models tested. Near the trailing edge ($0.80 < x/c_a < 1$), predictions of all models deviated somewhat from the experimental values, however, the contributions of these differences to the error in the prediction of force coefficients were not as significant as contributions from the suction side.

On the suction side of the blade, there is a noticeable increase in the pressure from $x/c_a = 0$ to $x/c_a = 0.60$. In this range, all models over-predict the surface pressure and this contributes significantly to the error in the tangential and axial forces on the blade. This difference could be attributed to the inability of turbulence models to describe strongly accelerating flows, however, further studies are required to determine the source of this error. The S-A and $k - \varepsilon$ turbulence models were unable to predict the separation bubble near the trailing edge region, as shown in Figure 7.7. It is also clear that the values of the x -component of the wall shear stress on the suction side of the blade predicted by the S-A and $k - \varepsilon$ models were higher than those predicted by the SST and RSM models. Higher wall shear stress and turbulence level predictions by the S-A and $k - \varepsilon$ turbulence models are perhaps the reason that these models did not predict separation. The SST and RSM models predict a separation region in the range $x/c_a \simeq 0.8$ to $x/c_a \simeq 0.9$, which is closer to the trailing edge than the experiments indicated.

All models give comparable pressure distributions over the turbine blade except near the trailing edge region, as indicated clearly in Figure 7.6. It should be noted that these simulations were for steady flow over a turbine blade, which is different from the case of main interest, namely unsteady flow with a rotating rotor. In addition, while the T106 cascade was used to test turbulence models, the flow over this turbine blade is laminar over 70% of the chord. Therefore, the T106 turbine is not particularly suitable for this study and, in retrospect, it is acknowledged that a more suitable geometry should have been chosen. The reason for choosing this cascade in the early stages of this study was that detailed experimental data of the pressure distribution over suitable high pressure turbine blades were difficult to find. Note also that Stieger et al. (2003) found that unsteadiness tends to eliminate the separation bubble region and, in addition, the low-pressure local minimum at $x/c_a = 0.03$ is reduced in unsteady flow. Therefore, it may be speculated that in unsteady simulations the differences between the numerical and experimental results would be reduced.

As a result of this study, it was determined that the SST and RSM models give comparable results for a two-dimensional flow. However, it was found that, in the 2-D case, the RSM model requires 1.7 times more computational time, than the SST model, when implemented on a 2.6 GHz AMD Athlon 64 X2 Dual Core Processor 5000+ with 4 GB of RAM. To keep the simulation time as short as possible, the SST model was used for the primary studies.

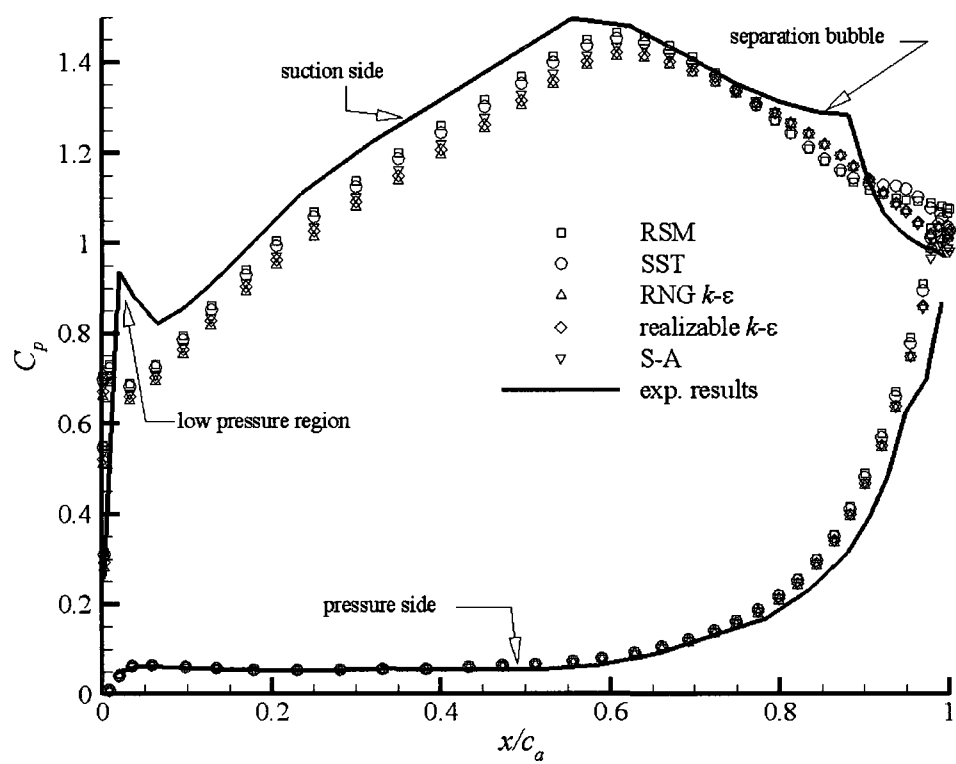


Figure 7.5: Pressure distribution along the T106 blade for various turbulence models.

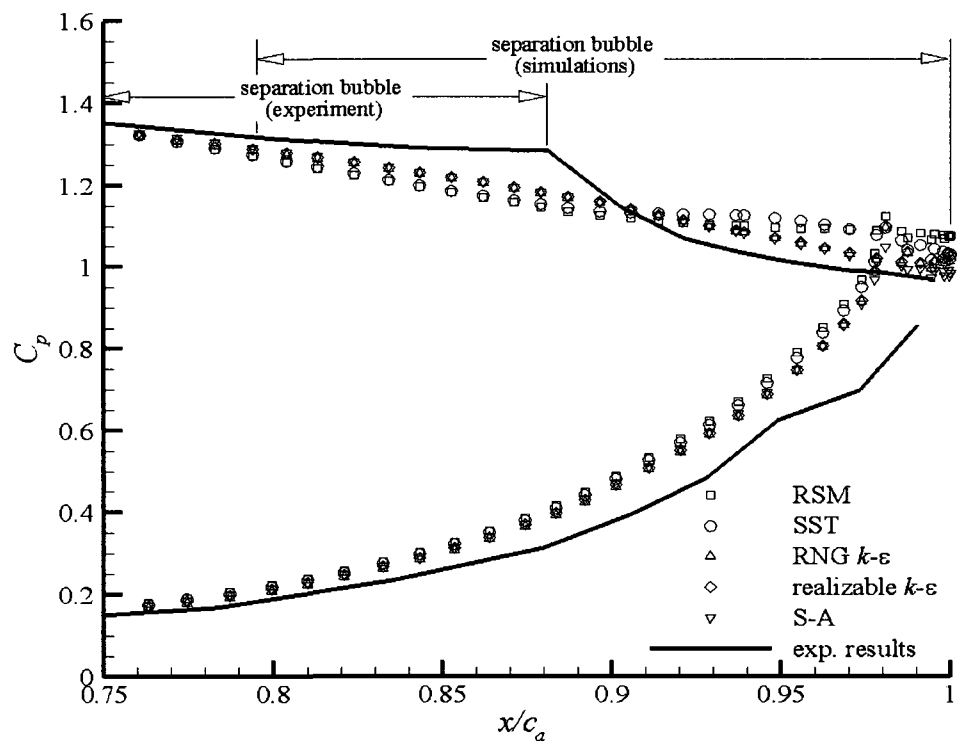


Figure 7.6: Pressure distribution along the T106 blade for various turbulence models.

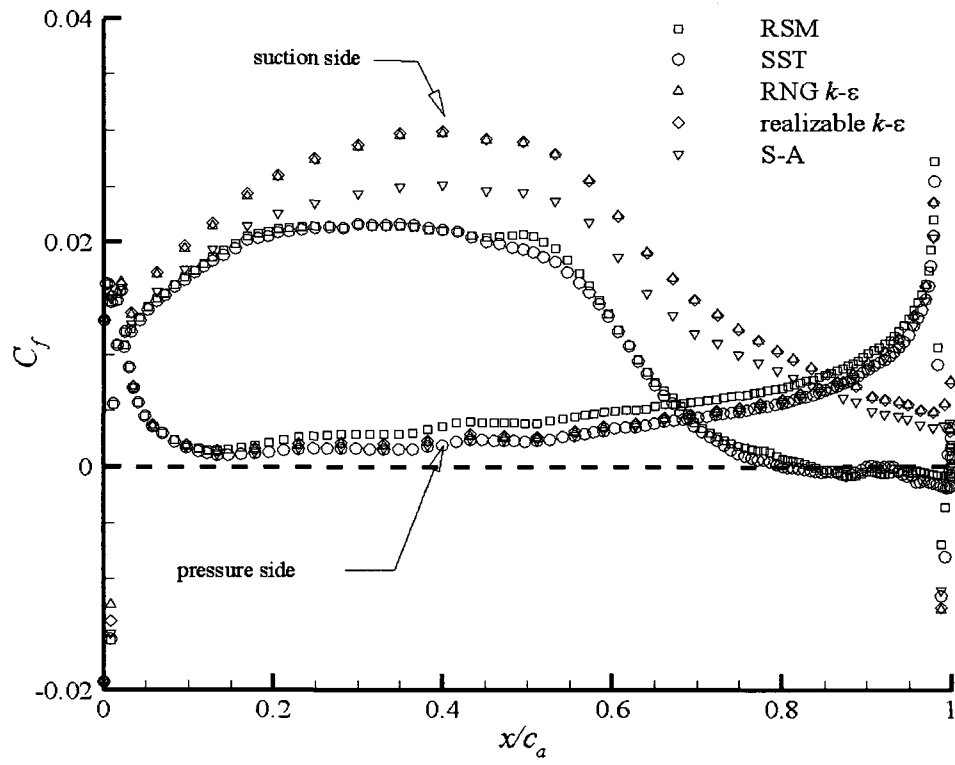


Figure 7.7: x -component of the coefficient of wall shear stress $C_f = 2\tau_{w,x}/\rho_{in}U_{in}^2$ over T106 blade for various turbulence models.

Chapter 8

Reference Simulations and Comparison with Experimental Results

The turbomachine used to study the effect of rotor blade scaling on CFD simulation results was obtained from the Institute for Thermal Turbomachinery and Machine Dynamics (TTM) at Graz University of Technology in Austria (Göttlich et al. (2004)). Experimental LDV and pneumatic flow field data are available for this turbine, which allows a comparison between the simulation results and the experimental results. Because no data were obtained for the pressure distribution over the vanes or rotor blades from the experiments, only comparisons of the velocity fields and the turbulence kinetic energy on planes B1 and C1 can be used to determine the accuracy of the simulation.

8.1 Simulation Verification

Three meshes were used to determine grid independence for the simulation of the TTM turbine. Due to the large amount of time required to run the simulation using the sliding mesh method, the grid independence study was conducted using the mixing plane model, which significantly reduced the simulation time. The three meshes, starting from the smallest and ending with the largest, are shown in Figures 6.5, 6.6 and 6.7, respectively. In addition, the node counts of the various grids and a summary of the simulation results are shown in Table 8.1. Results are presented using propagation of uncertainty based on the uncertainty (GCI value) for basic flow parameters (Tavoularis (2005)). It was assumed that the uncertainties in the inlet flow parameters and gas properties were negligible.

When a grid independence study was conducted, many flow parameters were found to exhibit oscillatory convergence. This was indicated by orders of convergence values that were less than 1. Because of this problem, the AES method was used to estimate the error band (GCI), as this method is still applicable when convergence is oscillatory, while the GCI method of Roache (1998) is only valid when the convergence pattern is monotonic.

States 1, 2 and 3 are defined as the inlet, a plane 80 mm downstream of the leading edge of the stator and a plane 158 mm downstream of the leading edge of the stator respectively. The results shown previously were determined by mass averaging of flow parameters over the corresponding plane. Parameters shown in Table 8.1 include the number of nodes m , the number of cells m_{cells} , the average length of one side of a cell d , the turbine expansion ratio $\bar{P}_{01}/P_{03}$, the non-dimensional stagnation temperature change $(\bar{T}_{01} - T_{03})/\bar{T}_{01}$, the non-dimensional mass flow rate $\dot{m}\sqrt{\mathbf{R}\bar{T}_{01}}/D_{hub}^2\bar{P}_{01}$ and the non-

Table 8.1: Comparison of simulation results using different grids for the TTM turbine

	Grid 3	Grid 2	Grid 1	GCI ¹	GCI ²
m	388209	539488	1143273	-	-
m_{cells}	1086984	1528798	3259851	-	-
d (mm)	2.12	1.89	1.47	-	-
$\tilde{P}_{01}/P_{03}$	3.4	3.4	3.4	6.2%	7.0%
$(\tilde{T}_{01} - T_{03})/\tilde{T}_{01}$	2.4×10^{-1}	2.4×10^{-1}	2.4×10^{-1}	4.7%	5.5%
$(\dot{m}\sqrt{\mathbf{R}\tilde{T}_{01}})/(D_{hub}^2\tilde{P}_{01})$	9.8×10^{-3}	9.8×10^{-3}	9.8×10^{-3}	3.0%	3.0%
$W/(\tilde{P}_{01}D_{hub}^2\sqrt{\mathbf{R}\tilde{T}_{01}})$	1.01×10^{-1}	1.01×10^{-1}	1.01×10^{-1}	2.5%	3.1%
η_{tt}	82%	82%	83%	4.4%	5.0%
ζ	1.0	1.0	1.0	34.8%	38.9%

dimensional power output $W/\tilde{P}_{01}D_{hub}^2\sqrt{\mathbf{R}\tilde{T}_{01}}$, where D_{hub} is the diameter of the hub in the rotor section. Furthermore, the stage total-to-total efficiency η_{tt} is defined using Equation 3.11, and the entropy loss coefficient ζ_s is defined using Equation 3.18. The parameters GCI¹ and GCI² indicate error brackets of the results using grids 1 and 2, respectively, at a 90% confidence level. Table 8.1 shows that the maximum estimated error bracket is lower than 7% for both grids 1 and 2 with the exception of the entropy loss coefficient. The apparently large error in the entropy loss coefficient need not be very disconcerting. The reasoning for this is presented in the following paragraph.

The uncertainty in the entropy loss coefficient is primarily due to the uncertainty in the change in entropy $E_{\Delta s}$, which is determined from the uncertainties in the total exit pressure $E_{P_{02}}$ and temperature $E_{T_{02}}$ (uncertainties in P_{01} , T_{01} , $\mathbf{C}_p$ and $\mathbf{R}$ are assumed to be negligible) as

$$E_{\Delta s} = \sqrt{\left(\frac{\partial \Delta s}{\partial T_{02}} E_{T_{02}}\right)^2 + \left(\frac{\partial \Delta s}{\partial P_{02}} E_{P_{02}}\right)^2} \quad (8.1)$$

where

$$\Delta s = \mathbf{C_p} \ln \left(\frac{T_{02}}{T_{01}} \right) - \mathbf{R} \ln \left(\frac{P_{02}}{P_{01}} \right) \quad (8.2)$$

Computing the derivatives and substituting into equation 8.1 gives the relative uncertainty in the change in entropy as

$$\frac{E_{\Delta s}}{\Delta s} = \sqrt{\left(\frac{\mathbf{C_p}}{\Delta s} \right)^2 \left(\frac{E_{T_{02}}}{T_{02}} \right)^2 + \left(\frac{\mathbf{R}}{\Delta s} \right)^2 \left(\frac{E_{P_{02}}}{P_{02}} \right)^2} \quad (8.3)$$

This shows that the the relative uncertainties in the exit total pressure and temperature amplify the relative uncertainty in the change in entropy by the respective sensitivity. The sensitivity of the relative uncertainty in the change in entropy to the relative uncertainty in the exit total temperature is given by $\mathbf{C_p}/\Delta s$, whereas the sensitivity of the relative uncertainty in the change in entropy to the relative uncertainty in the exit total pressure is $\mathbf{R}/\Delta s$. For example, using given values of specific heat ($\mathbf{C_p} = 1006.43 \text{ J/kg K}$) and gas constant ($\mathbf{R} = 287 \text{ J/kg K}$) and considering a representative case, for grid 1, in which $E_{T_{02}}/T_{02} = 0.91\%$, $E_{P_{02}}/P_{02} = 6.21\%$ and $\Delta s = 64.6 \text{ J/kg K}$, the relative uncertainty in the change in entropy can be estimated as

$$\frac{E_{\Delta s}}{\Delta s} = \sqrt{\left(\frac{1006.43 \text{ J/kg K}}{64.6 \text{ J/kg K}} \right)^2 (0.0091)^2 + \left(\frac{287 \text{ J/kg K}}{64.6 \text{ J/kg K}} \right)^2 (0.0621)^2} \quad (8.4)$$

$$= \sqrt{(15.6)^2 (0.009)^2 + (4.4)^2 (0.062)^2} \quad (8.5)$$

$$\approx \sqrt{0.02 + 0.075} \approx 0.31 \quad (8.6)$$

Therefore, even for low relative uncertainties in the exit total pressure (6.2%) and tem-

perature (0.9%), the relative uncertainty in the change in entropy turns out to be high (31%) because of the high sensitivities. Reducing the uncertainty in the change in entropy to approximately 5% would be an unrealistic goal, if one considers the complexity of the simulation.

The difference between the experimental results for the turbine expansion ratio ($P_{01}/P_{03} \approx 3.5$) and the non-dimensional power ($W/P_{01}D_{hub}^2\sqrt{RT_{01}} = 1.00 \times 10^{-1}$) and the corresponding numerical results using grid 1 are only 2.0% and 0.7%, respectively. This shows that these flow parameters are predicted more accurately than the AES method estimates. This may be the case for other flow parameters as well.

8.2 Simulation Validation

Experimental LDV and pneumatic flow field data are available for this turbine, which allows a comparison between the simulation results and the experimental results. Experimental results were obtained for this stage along planes B1 and C1 shown in Figure 5.5 in chapter 5. Plane B1 is located 64 mm behind the leading edge of the stator vanes and plane C1 is located 140 mm behind the leading edge of the stator vanes. In order to determine whether the simulation solution is accurate, the numerical results are compared to available experimental results, which have been provided by the Institute for Thermal Turbomachinery and Machine Dynamics in Austria. Time has been non-dimensionalized using the rotor passing period $\tau_r = 1.57 \times 10^{-4}$ s, namely the time it takes for one rotor blade to rotate by an angle equivalent to one rotor passage.

Figure 8.3 and 8.5 present the numerical solutions on the left and the experimental

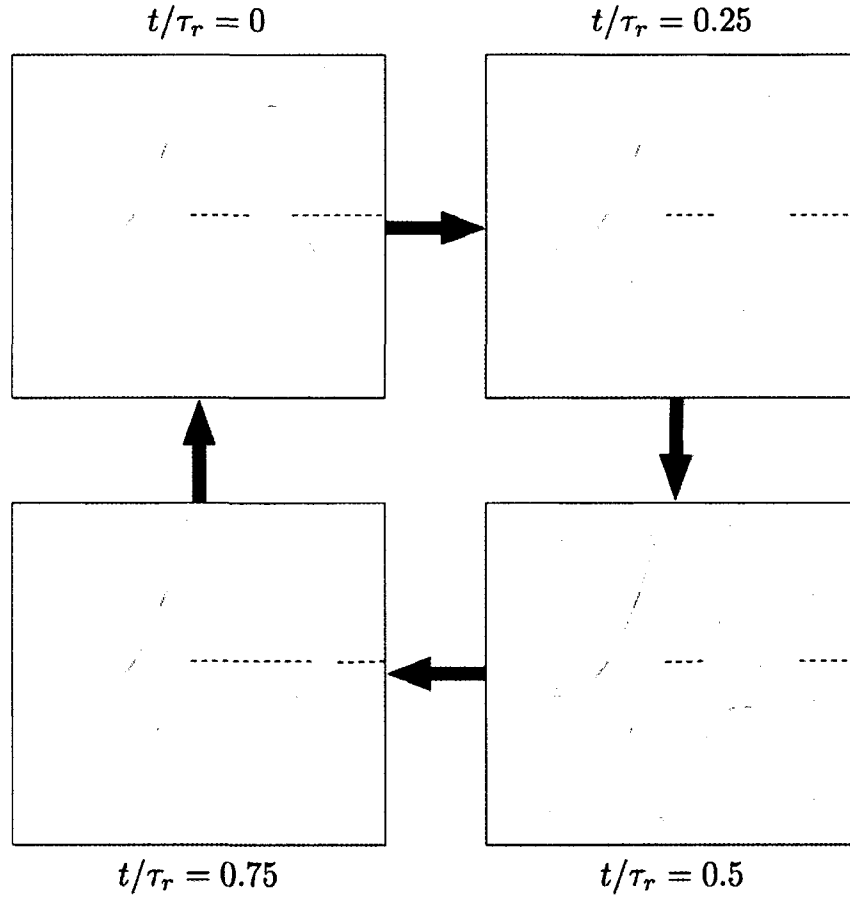


Figure 8.1: Relative positions of rotor blades at different times during the cycle; each plot is viewed in a direction normal to a plane tangent to the hub at a location indicated by the dashed line; the entire spans of vanes and blades are shown.

(LDV) results on the right. The plots are shown as if the viewer were downstream of the plane in question, looking upstream along the axis of the turbine. The time indices are identical for both the numerical and experimental solutions. However, the numerical solutions present data over the entire domain (from 0% span to 100% span), whereas the experimental results display data only from 32% span to 89% span for plane B1, and from 25% span to 80% span for plane C1.

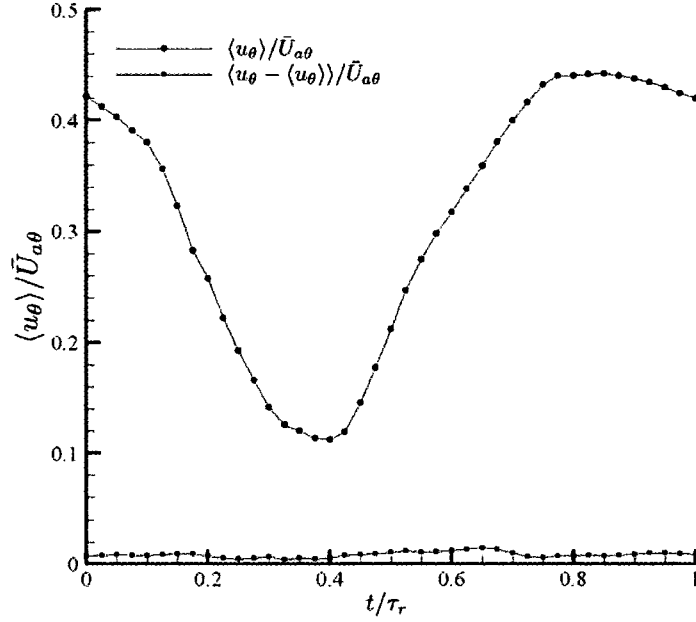


Figure 8.2: Comparison between ensemble average of tangential velocity and cycle-to-cycle fluctuations of the tangential velocity from the simulation.

Because the simulation was completed using URANS the cycle-to-cycle variation in flow parameters is negligible. For example, Figure 8.2 shows a comparison between the ensemble average, over 5 rotor blade passing cycles, of the tangential velocity $\langle u_\theta \rangle$, from the simulation, and the cycle-to-cycle variation of the tangential velocity $\langle u_\theta - \langle u_\theta \rangle \rangle$. The cycle-to-cycle variation is approximately 2.6% of the time-averaged tangential velocity $\overline{u_\theta}$. Therefore, ensemble averaging was not performed for additional data that is presented. Only the last period obtained from the simulations is presented.

Figure 8.3 shows a comparison of the predicted and measured phase-average velocity magnitude $U_{a\theta} = \sqrt{u_\theta^2 + u_a^2}$ (the subscript $a\theta$ indicates that the velocity magnitude is only composed of the tangential u_θ and axial u_a velocity components), non-dimensionalized by the space-time area weighted average velocity $\overline{\overline{U}}_{a\theta}$ on plane B1

$$\overline{\overline{U}} = \frac{1}{A} \frac{1}{\tau_p} \int_0^{\tau_p} \int_A U dA dt \quad (8.7)$$

The experimental results were phase-averaged over approximately 2000 rotor blade passing cycles. For the experimental results along plane B1, $\overline{\overline{U}}_{a\theta} = 377.52$ m/s. The numerical results along plane B1 were normalized by $\overline{\overline{U}}_{a\theta} = 361.88$ m/s, which is the area weighted average over the same area as the one used to compute the experimental area weighted average. The difference between the area weighted average velocities in the experiment and the simulations (both computed over the experimental area) is only 1.7%, which is acceptable, if one considers the experimental and numerical uncertainties.

In the experimental results, regions of low velocity clearly indicate the stator wakes, but also the presence of low speed cores of passage vortices, which contain boundary layer fluid upstream of the stator. According to Göttlich et al. (2004), the presented velocity contours identify the location of the passage vortex formed on the suction side near the casing, but are not sufficiently close to the hub to identify the hub passage vortex. The velocity deficit in the wake increases as the radial distance increases. The experimental results also show a shock on plane B1, shown in Fig. 8.3. Göttlich et al. (2004) also demonstrate that the rotor position affects the flow on plane B1, in particular the fluid angle.

In general, the numerical results on plane B1 show similar patterns as the experimental results, although they also have certain differences. Like the experimental ones, predicted stator wakes are clearly visible and their velocity deficits increase radially from the hub towards the casing. In contrast to the experiments, no evidence of low-speed fluid

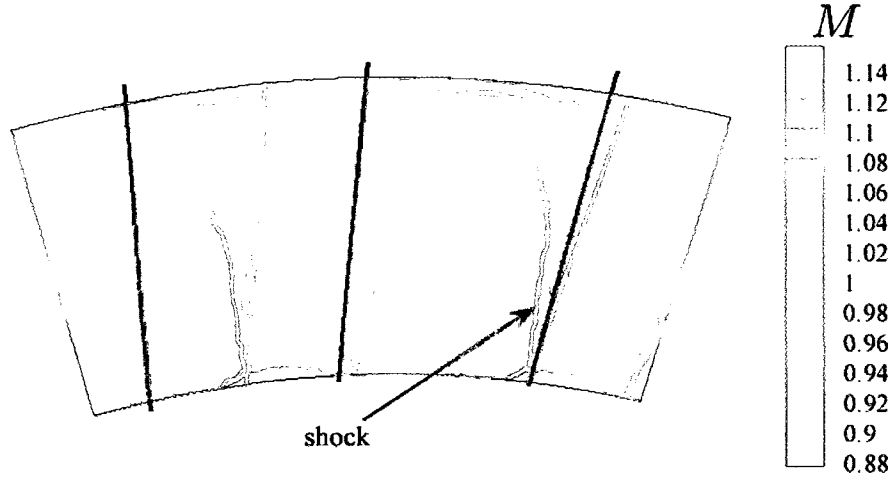


Figure 8.4: Detail of shock after stator vanes.

transported by passage vortices is contained by the simulation results. The circumferential positions of the stator wakes are shifted more towards the suction sides of the vanes in the simulations than in the experiments. Figure 8.4 shows a contour plot of the Mach number in plane B1. The numerical results also capture the formation of shock waves at locations close to those observed experimentally (Figure 8.3 and 8.4). Mesh refinement would likely result in more distinctive shocks. In addition, the effect of the rotor leading edge on the high velocity fluid in the stator passage is also captured as seen by a distortion in the high-velocity fluid region.

Figure 8.5 shows measured and predicted isocontours of $U_{a\theta}/\overline{\overline{U}}_{a\theta}$ on plane C1. $\overline{\overline{U}}_{a\theta} = 196.74$ m/s for the experimental results and $\overline{\overline{U}}_{a\theta} = 178.12$ m/s for the numerical results, area weighted averaged over an area equal to the area in the experimental results. This is a difference of approximately 10.0%.

The experimental results clearly show the rotor wakes, indicated by regions of low velocity fluid. The velocity deficit in the wakes does not depend appreciably on radial location. Regions of low velocity and high velocity fluid are observed towards the blade suction side (i.e., the side towards the direction of rotation) of the rotor wake, with the low velocity regions occurring more radially outward than the high velocity regions. Göttlich et al. (2004) attribute these low and high velocity regions to secondary flow phenomena caused by the rotor blades.

The experiments also demonstrate that the relative positions of the rotor blades and the stator vanes play an important role in the flow field in plane C1. An understanding of these complex phenomena can be assisted by Figure 8.6, showing entropy contours in the present simulations, which clearly identify the location of the stator wake. When the trailing edge of sv1 is approximately in line with the leading edge of rb1, the wake of sv1 passes through the passage between rb2 and rb3 (region B in figure 8.5). At this time ($t/\tau_r = 0$), a region of high velocity fluid is observed between rb2 and rb3, as shown in Figure 8.5. At the same time, the wake of sv2 is impinging on the leading edge of rb1. Along plane C1, a region of low velocity is observed experimentally, as well as in the simulations, between rb1 and rb2 (region A in figure 8.5). Therefore, when a stator wake passes directly through the rotor passage, high velocity fluid is observed. When the stator wake impinges directly onto the leading edge of a rotor blade, low velocity fluid is observed in the rotor passages on either side of the rotor blade in question. To further clarify this process, two sets of circles, A and B, have been marked in Figure 8.5 to identify the same region between two rotor blades at different times of the cycle. The proper sequence of events is such that one follows

region A as time increases from $t/\tau_r = 0$ to $t/\tau_r = 0.75$ and then switches to region B, corresponding to the time range from $t/\tau_r = 1.0$ to $t/\tau_r = 1.75$. As the stator wake passes between two rotor blades, the speed in region A increases from $t/\tau_r = 0$ to $t/\tau_r = 1.0$. In addition, the experiments show that the high- and low-speed regions associated with secondary flows become more distinct. For $t/\tau_r = 1.0$ to $t/\tau_r = 1.5$, the velocity in region B decreases with time. This interval corresponds to the rotation of a rotor blade by 15° , which is equal to one stator pitch. Experiments show that the secondary flow strength decreases during that time.

It is clear that the numerical simulation captures the rotor wakes, indicated by regions of low velocity fluid, and the flow patterns associated with the passage of stator wakes through the rotor. However, the numerical results show little evidence of the secondary flow phenomenon observed experimentally. A careful observation of the simulation results showed that they reproduced an incoming boundary layer and a spanwise pressure gradient which are responsible for the generation of the horseshoe vortex which develops into the passage vortex. Despite this, no clear horseshoe vortex was detected in front of the leading edge of the rotor blade near the endwall. However, possible evidence of the prediction of the horseshoe vortex and the passage vortex is contained in Figure 8.7, which shows isocontours of the entropy loss coefficient downstream of the rotor blade trailing edges. Previous investigators who were also unable to identify clearly the passage vortex attributed this to the effect of artificial dissipation (Constantinescu et al. (2004)) or to a low mesh density (Chernobrovkin and Lakshminarayana (1999)) in the formation region of the horseshoe vortex.

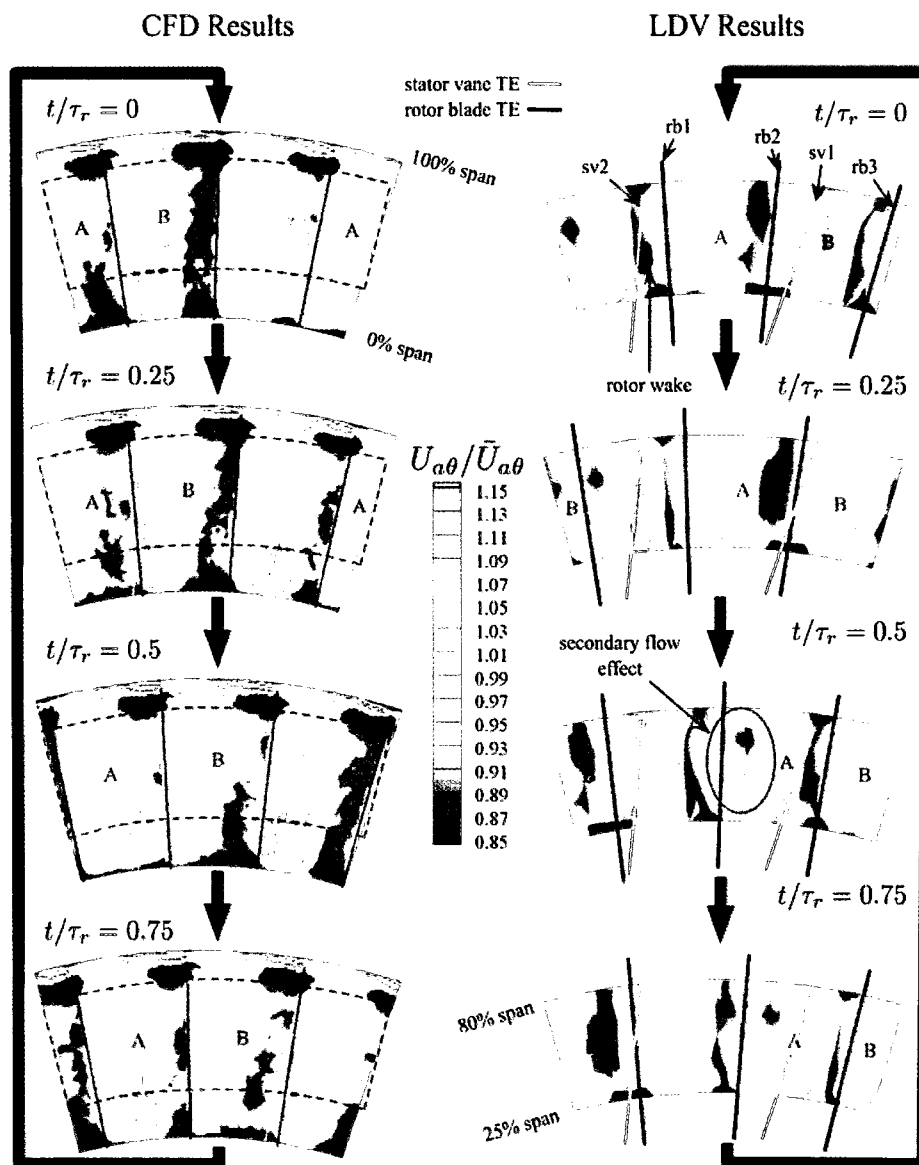


Figure 8.5: Comparison of predicted and measured (Göttlich et al. (2004)) non-dimensionalized velocities $U_{a\theta}/\bar{U}_{a\theta}$ on plane C1.

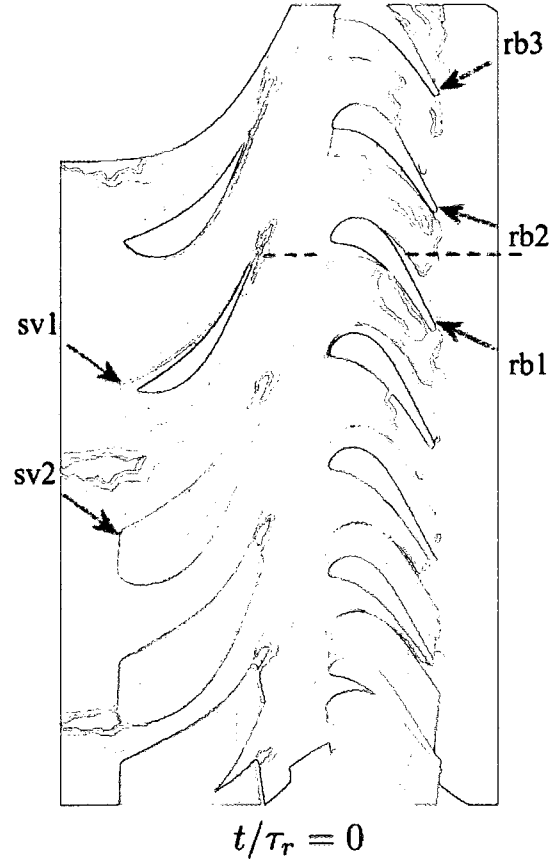


Figure 8.6: Predicted isocontours of entropy, showing the influence of rotor blade position on stator wakes at $t/\tau_r = 0$ and 50% span. Stator vanes are indicated as sv1 and sv2, whereas rotor blades are indicated as rb1, rb2 and rb3; entropy increases from blue towards red.

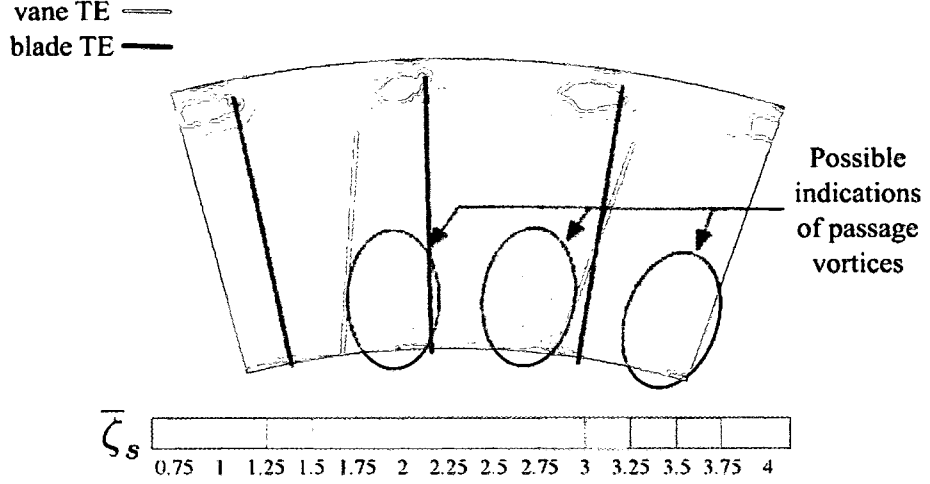


Figure 8.7: Predicted isocontours of time-averaged entropy loss coefficient $\bar{\zeta}_s$ on plane C1; high entropy loss coefficient near the root of the rotor blade gives possible indication to existence of passage vortices.

Statistical estimates of turbulent kinetic energy and other velocity fluctuations were non-dimensionalized using the values $\bar{U}_{a\theta} = 196.74$ m/s for the experimental results and $\bar{U}_{a\theta} = 178.12$ m/s for the numerical results. For the numerical results, $\bar{U}_{a\theta}$ was determined by an area-weighted average over an area equal to the area for which experimental results are available (Göttlich et al. (2004)). The kinetic energy of velocity fluctuations consists of different contributions, which differ for the experiments and the simulations (Chang and Tavoularis (2005)). For the experimental case, the kinetic energy was split into 2 parts:

- The kinetic energy of in-cycle variations

$$- \langle k_{cv} \rangle = \frac{3}{4} \left(\langle u_{a,cv}^2 \rangle + \langle u_{\theta,cv}^2 \rangle \right) \text{ where } \langle u_{cv} \rangle = \langle u \rangle - \bar{u}$$

- The kinetic energy of cycle-to-cycle fluctuations

$$- \langle k_{cc} \rangle = \frac{3}{4} \left(\langle u_a'^2 \rangle + \langle u_\theta'^2 \rangle \right) \text{ where } \langle u'^2 \rangle = \langle (u - \langle u \rangle)^2 \rangle$$

For the simulations the kinetic energy was initially divided into 3 parts:

- The kinetic energy of in-cycle variations

$$- \langle k_{cv} \rangle = \frac{3}{4} \left(\langle u_{a,cv} \rangle^2 + \langle u_{\theta,cv} \rangle^2 \right) \text{ where } \langle u_{cv} \rangle = \langle u \rangle - \bar{u}$$

- The kinetic energy of cycle-to-cycle fluctuations (resolved part)

$$- \langle k_{cc,res} \rangle = \frac{3}{4} \left(\langle u_{a,res}'^2 \rangle + \langle u_{\theta,res}'^2 \rangle \right) \text{ where } \langle u_{res}' \rangle = \langle u - \langle u \rangle \rangle$$

- The kinetic energy of cycle-to-cycle fluctuations (unresolved part)

$$- \langle k_{cc,ures} \rangle = \langle k \rangle \text{ where } k \text{ is the turbulence kinetic energy obtained directly from the turbulence model in the simulation}$$

Because no measurements of the radial velocity u_r' have been reported, $\frac{3}{4} (\langle u_{\theta}'^2 \rangle + \langle u_a'^2 \rangle)$ was introduced instead of the usual definition $\frac{1}{2} (\langle u_{\theta}'^2 \rangle + \langle u_a'^2 \rangle + \langle u_r'^2 \rangle)$ for the kinetic energy per unit mass, under the assumption that $\langle u_r'^2 \rangle \simeq \frac{1}{2} (\langle u_{\theta}'^2 \rangle + \langle u_a'^2 \rangle)$. This was done for both the simulation and experimental results in order to facilitate a direct comparison between the two cases.

As seen in Figure 8.8, the resolved component of the kinetic energy of cycle-to-cycle fluctuations is negligible with respect to the kinetic energy of in-cycle variations. Therefore, for simplicity, the resolved component of the kinetic energy of cycle-to-cycle fluctuations was ignored. In addition, as stated previously, ensemble averaging was not required because the cycle-to-cycle fluctuations of velocities are small. This is reflected by the fact that the resolved component of the kinetic energy of cycle-to-cycle fluctuations is small. As a result,

the division of kinetic energy of velocity fluctuations in the simulations was adjusted to reflect these simplifications:

- the kinetic energy of in-cycle variations $k_{cv} = \frac{3}{4} (u_{a,cv}^2 + u_{\theta,cv}^2)$ where $u_{cv} = u - \bar{u}$
- the kinetic energy of cycle-to-cycle fluctuations $k_{cc} = k$ where k is the turbulence kinetic energy obtained directly from the turbulence model in the simulation

For the experimental results the total kinetic energy was defined as

$$\langle k_{tot} \rangle = \langle k_{cv} \rangle + \langle k_{cc} \rangle = \frac{3}{4} (\langle u_{a,cv}^2 \rangle + \langle u_{\theta,cv}^2 \rangle) + \frac{3}{4} (\langle u_a'^2 \rangle + \langle u_\theta'^2 \rangle) \quad (8.8)$$

Whereas, for the simulation results the total kinetic energy was defined

$$k_{tot} = k_{cv} + k_{cc} = \frac{3}{4} (u_{a,cv}^2 + u_{\theta,cv}^2) + k \quad (8.9)$$

By dividing the kinetic energy of velocity fluctuation in this way the most prominent velocity fluctuations can be compared. For example, if only the cycle-to-cycle fluctuations are used for comparison, the simulation shows no rotor wake in Figure 8.8, while the experimental results indicates the rotor wake passing by a peak in the kinetic energy of the cycle-to-cycle fluctuations as shown in Figure 8.9. However, if the in-cycle variations are used in conjunction with the cycle-to-cycle fluctuations then the rotor wake is clearly indicated in both Figures 8.8 and 8.9. In addition, a region of high kinetic energy, that occurs after the rotor wake passing, is shown in both figures. The largest peak in the kinetic energy, shown in both the experimental and simulation results, corresponded to a large decrease in the tangential velocity (flow direction is primarily axial) as shown in Figure

8.10. The second, smaller peak in the kinetic energy corresponded to an increase in the tangential velocity (flow obtained a large tangential component in a direction opposite to the rotor rotation direction) as shown in Figure 8.10. The minimum and maximum tangential velocities were not as low and as high in the simulations as was observed in the experiments. Therefore, the simulations showed smaller peaks in the kinetic energy when compared to the experiments. In addition, the time between the minimum value of the tangential velocity and the maximum value was greater in the simulations than in the experiments. This caused the peaks in the kinetic energy from the simulations to be spread farther apart than was observed in the experiments. Other higher frequency fluctuations can be seen in the tangential velocity in the experimental results that are not seen in the simulation results. However, these high frequency fluctuations do not contribute significantly to the kinetic energy when compared to the largest fluctuations explained previously.

Figure 8.11 shows measured and predicted contours of the non-dimensionalized turbulent kinetic energy $k_{a\theta}/\overline{U}_{a\theta}^2$ were

$$k_{a\theta} = \langle k_{tot} \rangle \text{ for experimental results} \quad (8.10)$$

$$k_{a\theta} = k_{tot} \text{ for simulation results} \quad (8.11)$$

The simulation results show wakes with lower levels of kinetic energy than those observed in the experimental results, as shown in Figure 8.11. In addition, as stated previously, the simulation results do not show the secondary flow and the passage vortex that are indicated by the regions of high kinetic energy in region A at $t/\tau_r = 0.5$ in the experimental results. The simulation results do show the tip leakage vortex, which the experimental contours miss because experimental results were only collected from 24.9% span to 80.2%

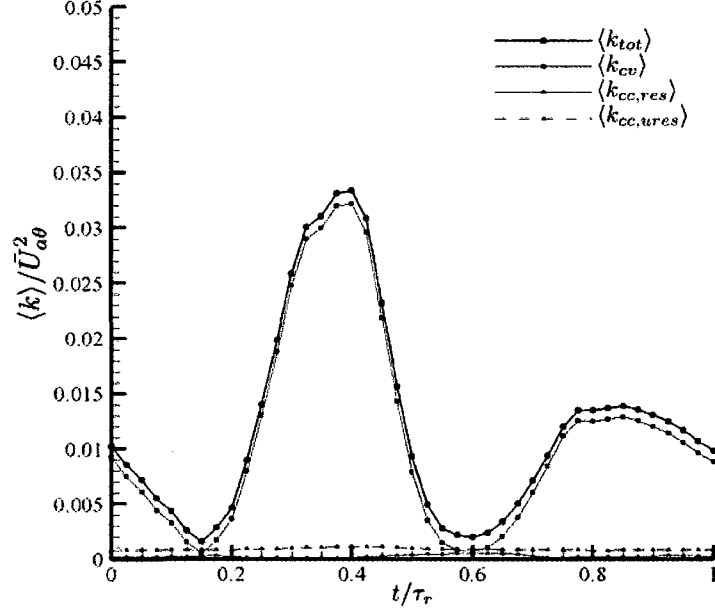


Figure 8.8: Kinetic energies of various velocity fluctuations and variations for the simulation results at $r = 238$ mm on plane C1.

span whereas the simulations show that the tip leakage vortex was located in the region beyond the 80.2% span. The experimental results show that the flow kinetic energy in the rotor wake is higher than those in the passage vortex and the secondary flow. This would lead to a conclusion that most of the loss is concentrated in the rotor wake rather than in the secondary flow and passage vortex. In contrast, the simulation results show that the flow kinetic energy is largest in the tip leakage vortex.

In general, the simulation underestimated the kinetic energy of the flow, especially near the hub, as shown in Figure 8.11. Kinetic energy near the hub was underestimated because the simulation did not capture passage vortices that were observed in the experiments. Kinetic energy in the rotor wakes was, most likely, underestimated because of excessive artificial dissipation of velocity as shown in Figure 8.10.

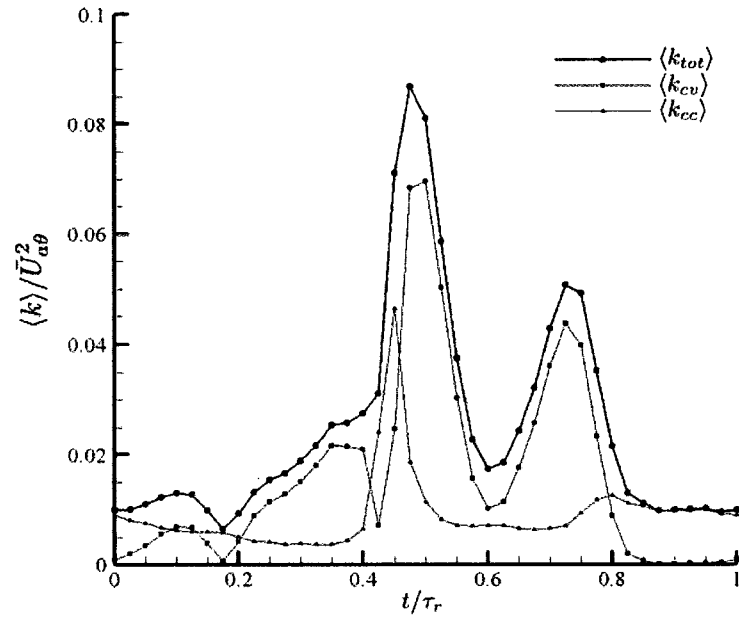


Figure 8.9: Kinetic energies of various velocity fluctuations and variations for the experimental results at $r = 238$ mm on plane C1.

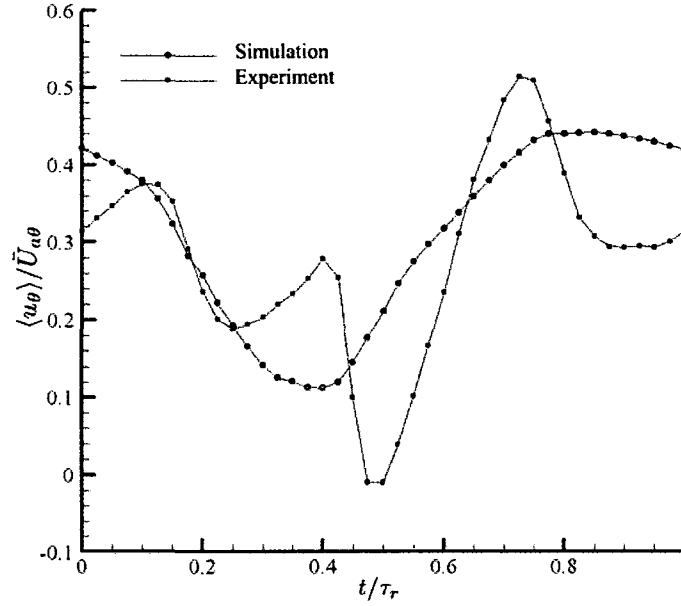


Figure 8.10: Ensemble average of tangential velocity at $r = 238$ mm on plane C1.

Figure 8.12 shows a comparison of the predicted and measured turbine power outputs. The numerical result is approximately 4.1% higher than the experimental result. Considering the various approximations and uncertainties in both experiments and simulations, this level of agreement for the turbine power output is deemed to be acceptable. In addition, the time history of the predicted power output shows very small cycle-to-cycle variations, which confirms that the solution has approached a steady state.

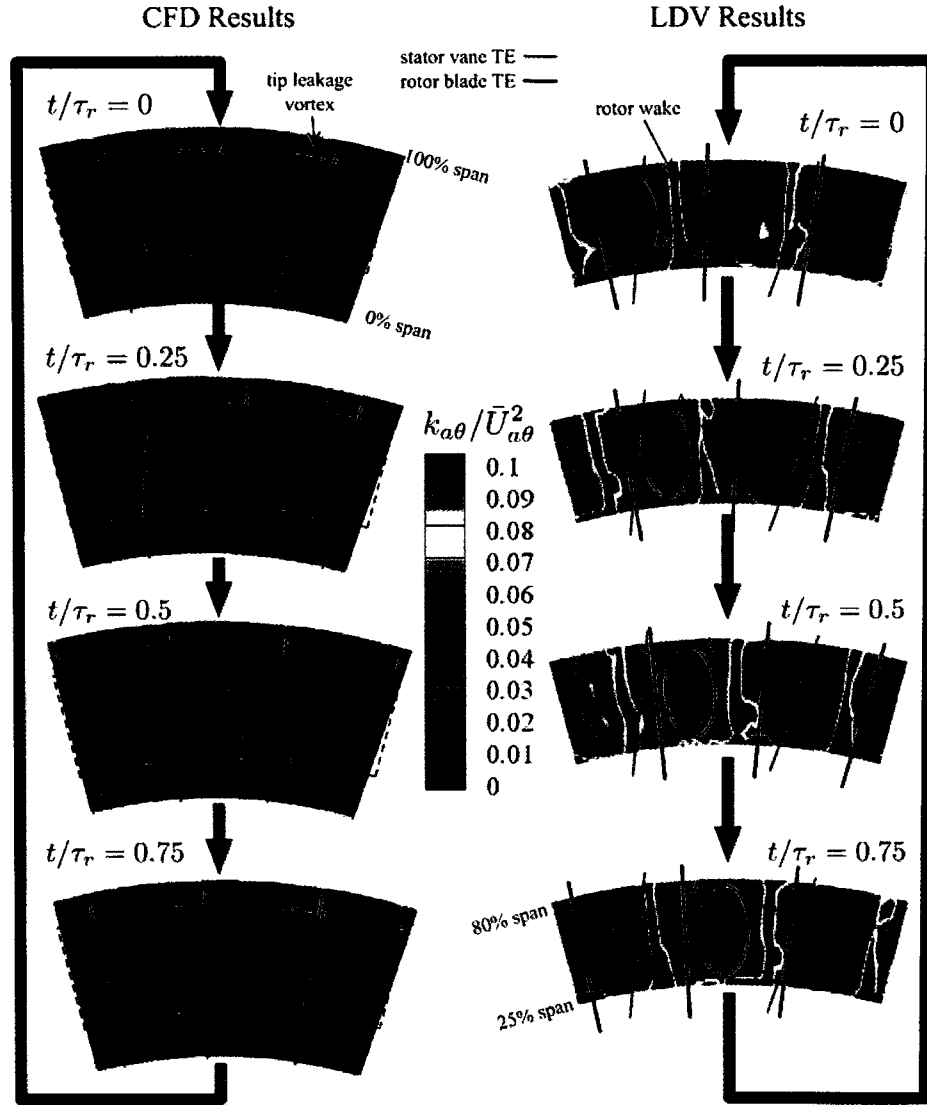


Figure 8.11: Comparison of predicted and measured (Göttlich et al. (2004)) non-dimensionalized turbulent kinetic energies $k_{a\theta} / \bar{U}_{a\theta}^2$ on plane C1.

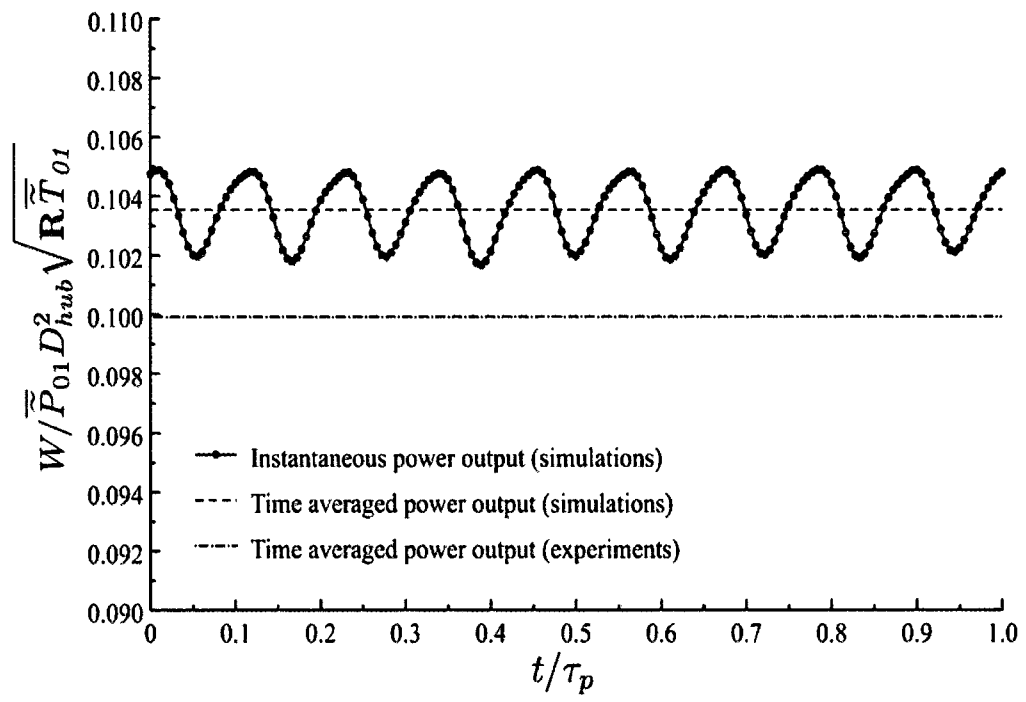


Figure 8.12: Predicted and measured non-dimensionalized turbine power output.

Chapter 9

Comparative Assessment of Scaling Effects

This chapter gives detailed comparisons of the simulations of the three geometries that have been considered. In addition to unsteady results, time averages $\bar{g}$ and standard deviations g_{SD} of the various parameters over one simulation rotational period have been calculated on several surfaces of interest.

The relative positions of the rotor blades for all three cases are shown in Figure 9.1. In addition, the stator wakes are also visible, marked by high entropy values, shown and the surface of a reference rotor blade (rb1) has been marked as darker than the surfaces of the other blades. The reference rotor blade is the one that, at time 0, has its leading edge at the hub in-line with the trailing edge of a stator vane at the hub.

In this study, there are three distinct time periods which can be used as scales for normalizing time t :

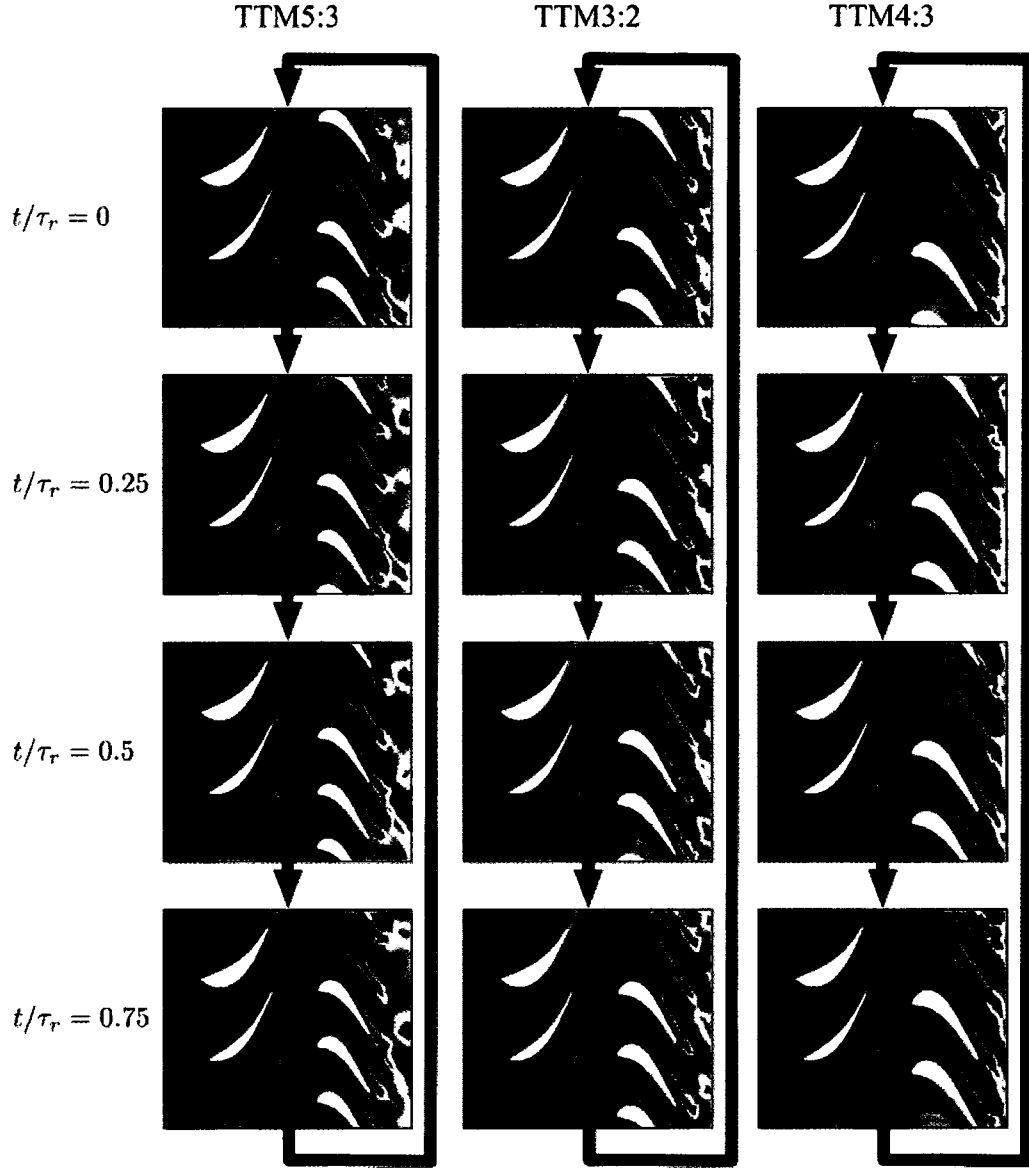


Figure 9.1: Relative positions of rotor blades for the three simulation cases at 4 time steps and at 50% span; one blade that serves as reference in the discussion (rb1) is indicated as darker than the others; stator wakes are shown as areas of large entropy (entropy increases from blue to red).

- The simulation rotational period τ_p , which is the time it takes the rotor to rotate by an angle equal to the angle between the periodic boundaries; this was 30° for TTM3:2 and 45° for TTM5:3 and TTM4:3.
- The stator vane passing period τ_s , which is the time it takes the rotor to rotate by an angle equal to the angle between two consecutive stator vanes; this was 15° for all cases.
- The rotor blade passing period τ_r , which is the time it takes the rotor to rotate by an angle equal to the angle between two consecutive rotor blades; this was 10° for TTM3:2, 9° for TTM5:3, and 11.25° for TTM4:3.

Additional scales used for the normalization of other flow parameters have been summarized in Table 9.1. These parameters include the time-space mass weighted averages $\widetilde{U}_{B1}$ and $\widetilde{U}_{C1}$ of the velocity magnitude along planes B1 and C1, respectively, the time-space mass weighted averages $\bar{\rho}_{B1}$ and $\bar{\rho}_{C1}$ of the density along planes B1 and C1, the time-space mass weighted averages of the total pressure $\bar{P}_{01}$ and the total temperature $\bar{T}_{01}$, the midspan chord length c of a rotor blade, and representative averages of the bulk velocity $\widetilde{U}_{Bulk}$ and the bulk density $\bar{\rho}_{Bulk}$, defined as

$$\widetilde{U}_{Bulk} = \frac{\widetilde{U}_{C1} + \widetilde{U}_{B1}}{2} \quad (9.1)$$

$$\bar{\rho}_{Bulk} = \frac{\bar{\rho}_{C1} + \bar{\rho}_{B1}}{2} \quad (9.2)$$

Figure 9.2 shows the different planes used for visualization of isocontours of the various properties. There are 5 axial planes (B1, B2, B3, B4 and C1) and 3 spanwise planes. As stated previously, plane B1 is located between the stator and the rotor (1.14 times the

Table 9.1: Flow properties used for normalization

	TTM3:2	TTM5:3	TTM4:3
$\bar{U}_{B1}$ (m/s)	351.8	363.4	375.3
$\bar{U}_{C1}$ (m/s)	176.7	182.6	174.7
$\bar{\rho}_{B1}$ (kg/m ³)	1.70	1.71	1.64
$\bar{\rho}_{C1}$ (kg/m ³)	1.00	1.03	1.02
$\bar{P}_{01}$ (kPa)	333.2	333.2	333.2
$\bar{T}_{01}$ (K)	402.7	402.7	402.7
$\bar{c}$ (mm)	55.9	50.3	62.9
$\bar{U}_{Bulk}$ (m/s)	264.3	273.0	275.0
$\bar{\rho}_{Bulk}$ (m/s)	1.35	1.37	1.33

midspan stator vane axial chord $c_{a,st}$ downstream of the leading edge of the stator vane or 64 mm behind the leading edge of the stator vane) for all cases, whereas plane C1 is located just behind the trailing edge of the rotor blades (1.21 times the midspan rotor blade axial chord $c_{a,rt}$ downstream of the leading edge of the rotor blade). Planes B2, B3, and B4 are located at 10%, 50% and 90% of the rotor axial hub chord, respectively. The 3 spanwise planes are located at 10%, 50% and 90% span.

9.1 Computational Gain by Scaling

Unsteady simulations using a time step of 1.9625×10^{-7} s were used to resolve unsteady effects of interest in this work. The three configurations for the simulations included the unscaled case (TTM3:2; $\Lambda = 1.5$), a case downscaled by 10% (TTM5:3; $\Lambda_{sc} = 1.67$, $q = 0.90$), and a case upscaled by 12.5% (TTM4:3; $\Lambda_{sc} = 1.33$, $q = 1.125$). The numbers of nodes in the mesh for each of these cases were approximately 0.5×10^6 , 0.7×10^6 and 0.7×10^6 , respectively. The grids for the scaled cases were generated in an identical manner

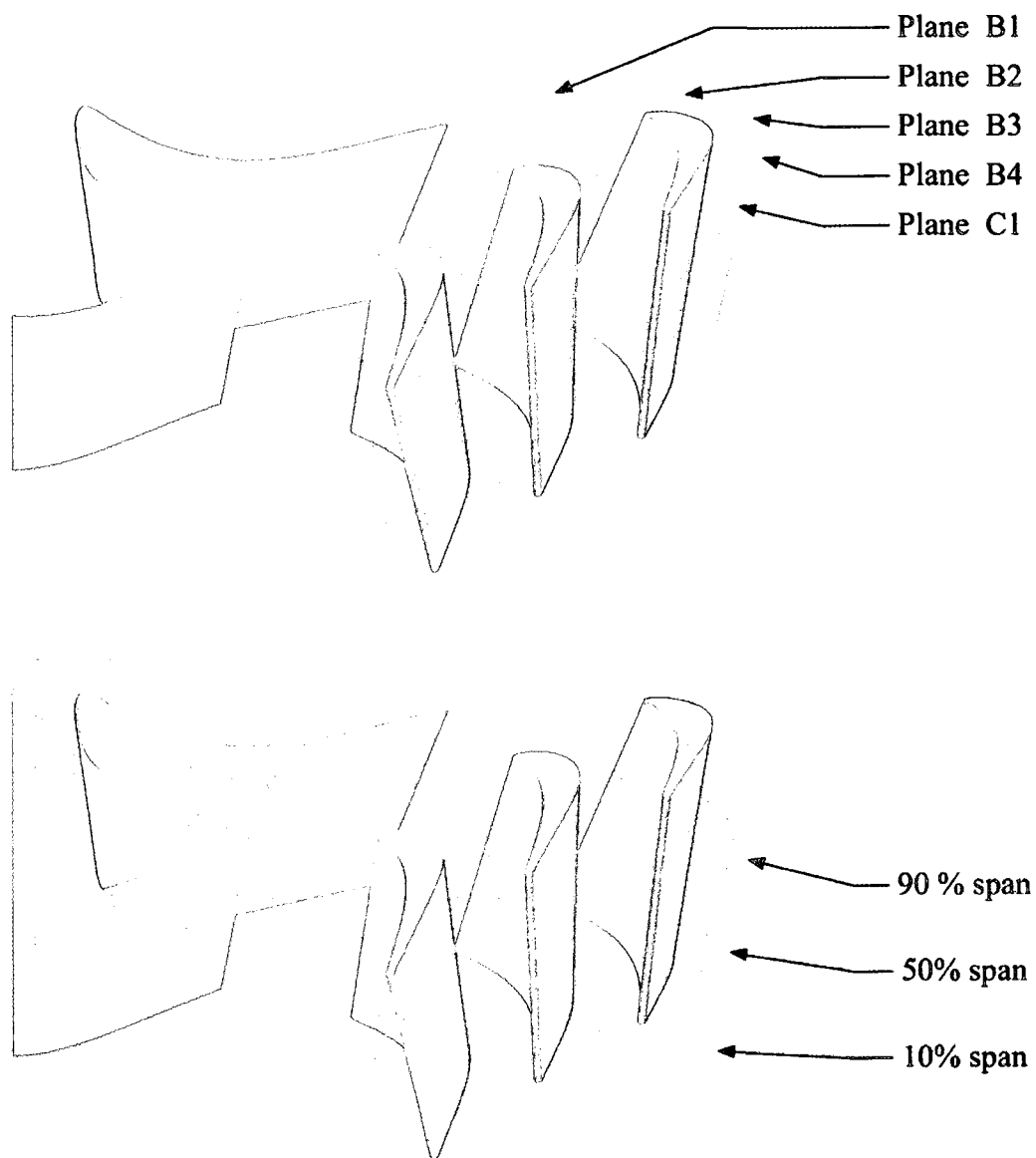


Figure 9.2: Locations of planes used for visualization of isocontours.

to the grid chosen for the unscaled case. Parallel simulations were run in clusters of 20 processors. Average computing times for the TTM3:2, TTM5:3 and TTM4:3 cases were, respectively, 0.9, 3.1 and 2.6 min. per time-step, confirming that TTM3:2 simulations are the most efficient and that scaling can have a significant impact on the simulation time.

9.2 Turbine Performance Parameters

Turbomachinery performance was determined by analyzing the time-averaged $\overline{W}/\overline{P}_{01}D_{hub}^2\sqrt{\mathbf{R}\overline{T}_{01}}$ and time-varying non-dimensionalized power $W/\overline{P}_{01}D_{hub}^2\sqrt{\mathbf{R}\overline{T}_{01}}$, the isentropic efficiency η_{tt} , the non-dimensional total temperature drop $\overline{T}_{01} - \overline{T}_{03}/\overline{T}_{01}$ across the turbine and the total pressure ratio $\overline{P}_{01}/\overline{P}_{03}$. Time-varying performance parameters were obtained by spatial mass-averaging of the corresponding local values (P_0 , T_0 , etc.) over the inlet and outlet planes. The spatially mass-averaged parameters were then plotted against time. Time-averaged performance parameters were obtained by time averaging the time varying performance parameters over at least three rotor blade passing periods $3\tau_r$.

Table 9.2 shows that scaling has a negligible effect on the time-averaged turbine performance. Both downscaling and upscaling decreased the predicted turbine power output $\overline{W}$ (shown normalized by the inlet total pressure $\overline{P}_{01}$, the hub diameter at the rotor D_{hub} , the gas constant of air $\mathbf{R}$ and the total inlet temperature $\overline{T}_{01}$), the isentropic efficiency η_{tt} and left the total temperature drop $\overline{T}_{01} - \overline{T}_{03}$ across the turbine, and the total pressure ratio $\overline{P}_{01}/\overline{P}_{03}$ essentially unchanged.

The time-varying power output is altered by scaling as seen in Figure 9.3. Both downscaling and upscaling reduced the fluctuations of the unsteady power output with

Table 9.2: Time averaged turbine performance parameters for scaled and unscaled cases

Case	$\frac{\bar{W}}{\bar{P}_{01} D_{hub}^2 \sqrt{R \bar{T}_{01}}}$	η_{tt}	$\frac{\bar{P}_{01}}{\bar{P}_{03}}$	$\frac{(\bar{T}_{01} - \bar{T}_{03})}{\bar{T}_{01}}$
TTM3:2	0.104	0.89	3.2	0.25
TTM5:3	0.103 (-1.0%)	0.88 (-1.1%)	3.2 (0.0%)	0.25 (0.0%)
TTM4:3	0.103 (-1.0%)	0.88 (-1.1%)	3.2 (0.0%)	0.25 (0.0%)

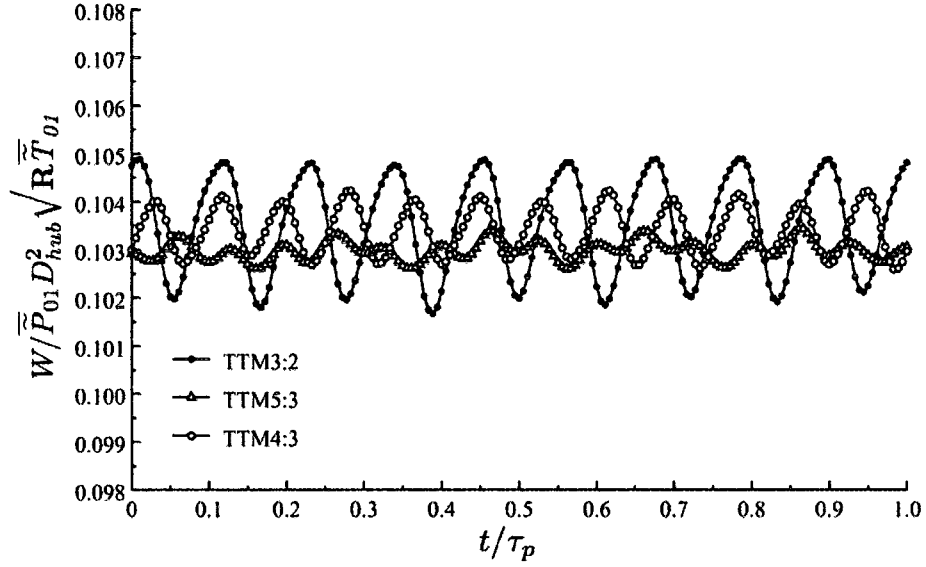


Figure 9.3: Comparison of non-dimensionalized unsteady turbine power output between cases.

respect to the unscaled case. The drop in the time-averaged power output was 1.0%, which is slightly lower than estimates under the constant total power assumption. In summary, both downscaling and upscaling resulted in slightly lower time-averaged power output, however, the observed change is less than the uncertainty of the simulations.

9.3 Rotor Blade Surface Pressure

The rotor blade surface static pressure was normalized by a dynamic pressure $\frac{1}{2}\bar{\rho}_{Bulk}\bar{U}_{Bulk}^2$, defined using the bulk density and the bulk velocity.

9.3.1 Time-Averaged Pressure

Pressure data over the rotor blade surfaces and at 10%, 50% and 90% spans were collected at 80 instances per rotor blade passing period τ_r . Averages were computed over at least 3 rotor blade passing periods. Figure 9.4 shows the time-averaged surface pressure coefficient $\bar{P}/\frac{1}{2}\bar{\rho}_{Bulk}\bar{U}_{Bulk}^2$ over the entire rotor blade surface for the three cases. There are very small differences in the rotor blade surface pressure distributions. However, near the root of the blades and towards the leading edge, on the suction surface of the blade, the TTM4:3 case has lower pressure than the TTM3:2 and TTM5:3 cases. The differences among the three cases in the region of high pressure (regions A in Figure 9.4), near the tip of the blades and towards the leading edge on the pressure side of the blades, are relatively small and likely within the uncertainty of the simulations. In the TTM3:2 case, there seems to be a larger pressure gradient between the high and low pressure regions on the pressure side of the blade.

Figures 9.5 to 9.7 show more clearly the differences among the three cases at 10%, 50% and 90% spans. At all spanwise locations, the pressure distribution along the pressure side of the rotor blade was higher in the TTM3:2 case along the entire length of the blade when compared to the TTM5:3 and TTM4:3 cases. Over the suction side, slightly lower pressures were observed in the scaled cases along over approximately 80% of the

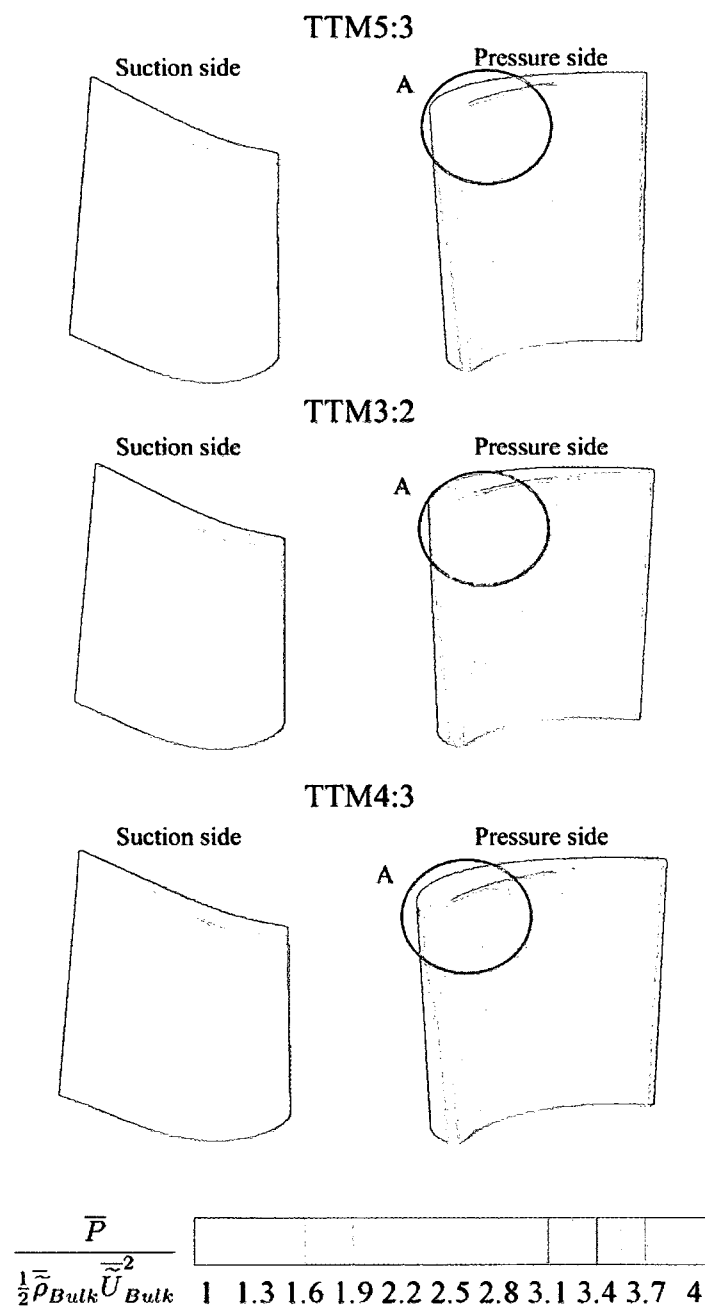


Figure 9.4: Time averaged surface pressure distributions over a rotor blade for the three cases considered.

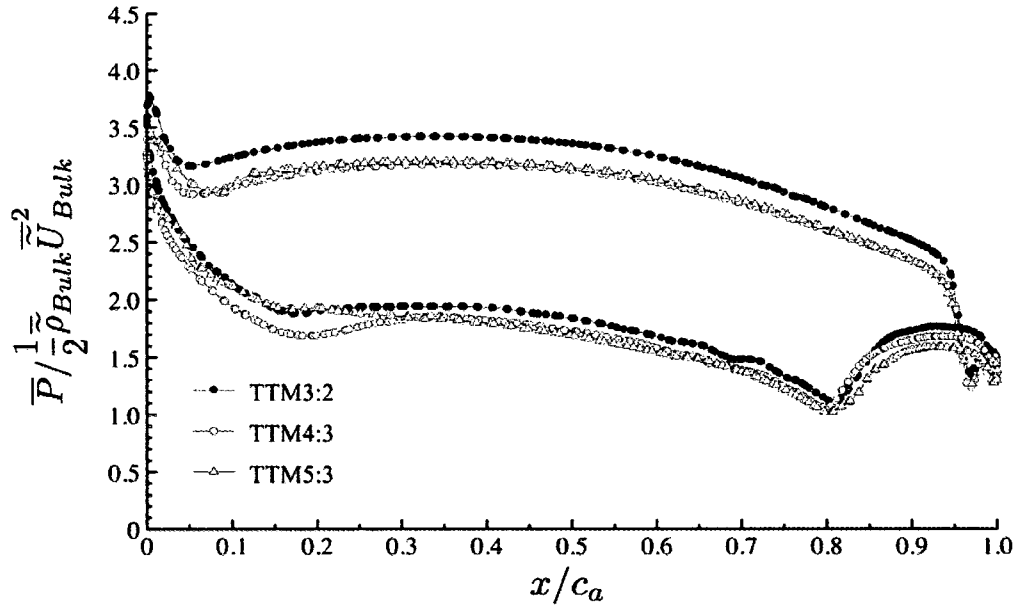


Figure 9.5: Time averaged pressure distribution over a rotor blade at 10% span.

blade surface, starting from the rotor blade leading edge. In contrast, from approximately 80% to 100% along the suction side of the rotor blades, the local pressure was higher in some locations in the scaled cases than that in the unscaled case. At 10% span, a large bump in the pressure distribution (significantly lower local pressure) on the suction side, at approximately 20% axial chord, develops following upscaling. The differences in the time averaged rotor blade pressure decrease with increasing span and the bump seen at 10% span essentially vanishes at 50% span and beyond.

9.3.2 Pressure Fluctuation Amplitude

The variations of the amplitude of pressure fluctuations over the rotor blade surface for the three cases considered are shown in Figure 9.8. The TTM5:3 case shows much lower

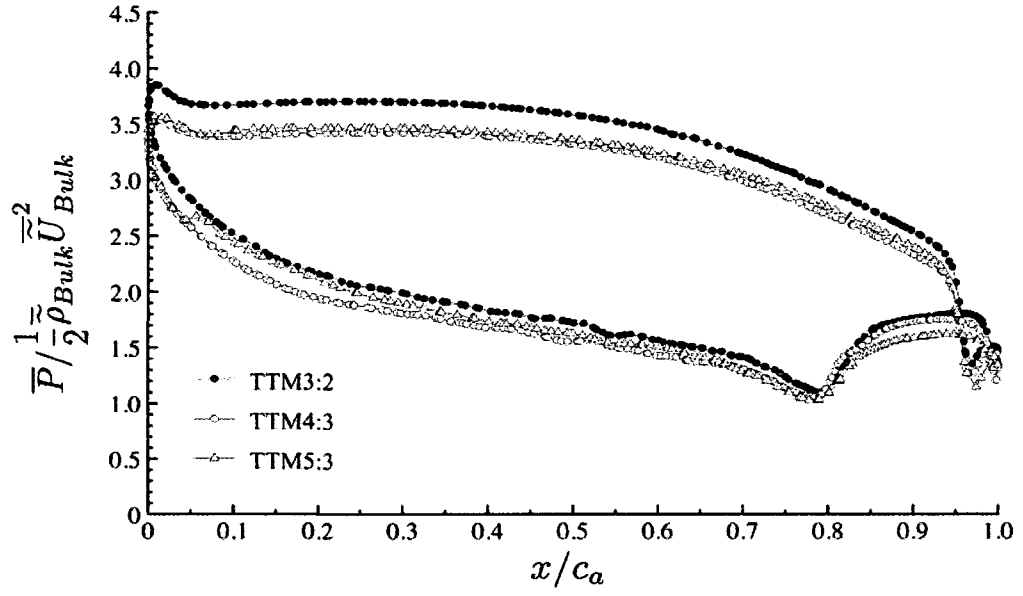


Figure 9.6: Time averaged pressure distribution over a rotor blade at 50% span.

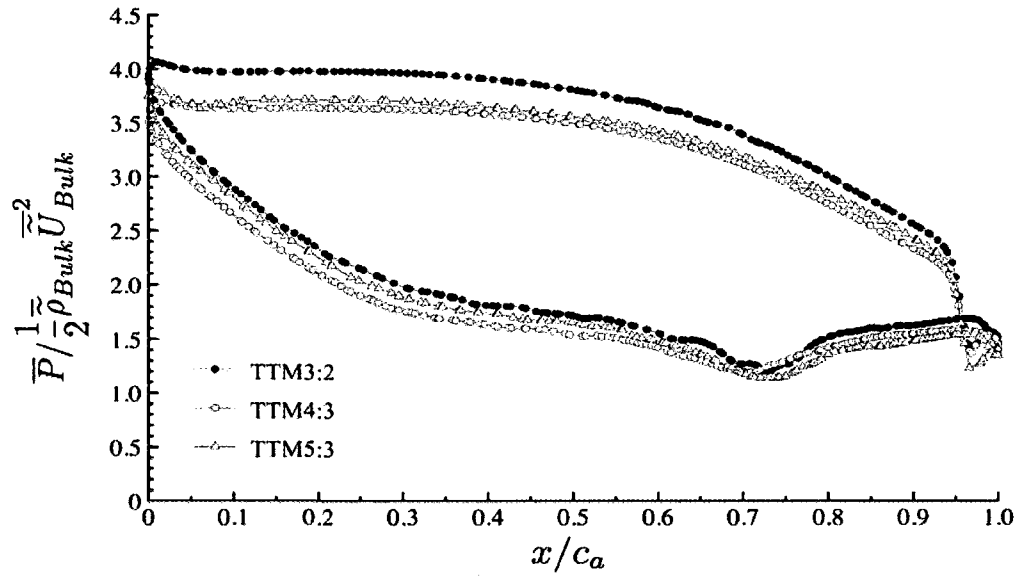


Figure 9.7: Time averaged pressure distribution over a rotor blade at 90% span.

amplitudes of pressure fluctuations near the root of the blade and towards the leading edge (regions A in Figure 9.8), while the TTM4:3 case shows higher amplitudes of pressure fluctuations. In addition, the amplitudes of pressure fluctuations on the pressure side of the rotor blade (regions B in Figure 9.8) are lower in the two scaled cases than in the TTM3:2 case. The regions of high amplitude pressure fluctuations are at approximately the same location for all cases. There is a line of high fluctuation amplitude fluid towards the trailing edge and on the suction side of the rotor blade (regions C in Figure 9.8). Pressure fluctuations in this region may be attributed to a shock wave that originates on the adjacent blade, in the direction of rotation, and is reflected off the rotor blade seen in Figure 9.8.

9.3.3 Unsteady Pressure

Figures 9.9 and 9.10 show plots of the time varying pressure $P/\frac{1}{2}\bar{\rho}_{Bulk}\bar{U}_{Bulk}^2$ over the suction and pressure surfaces of a rotor blade at 10%, 50% and 90% spans for the three cases examined. In all cases, time has been normalized by the stator vane passing period τ_s . These figures show clearly that, in all cases, the amplitudes of pressure fluctuations on the suction and pressure surfaces of the rotor blade decrease from the root to the tip. When comparing the three cases against each other, all cases seem to exhibit comparable amplitudes of pressure fluctuations and similar distributions of pressure. Further inspection shows that the amplitude of pressure fluctuations in the upscaled case (TTM4:3) is slightly larger than that the unscaled case (TTM3:2) and that the amplitude of pressure fluctuations in the downscaled case (TTM5:3) is slightly lower than that in the unscaled case (TTM3:2). To quantify the amplitude of pressure fluctuations, the power spectral density (PSD) of the pressure fluctuations was calculated and is shown in Figure 9.11.

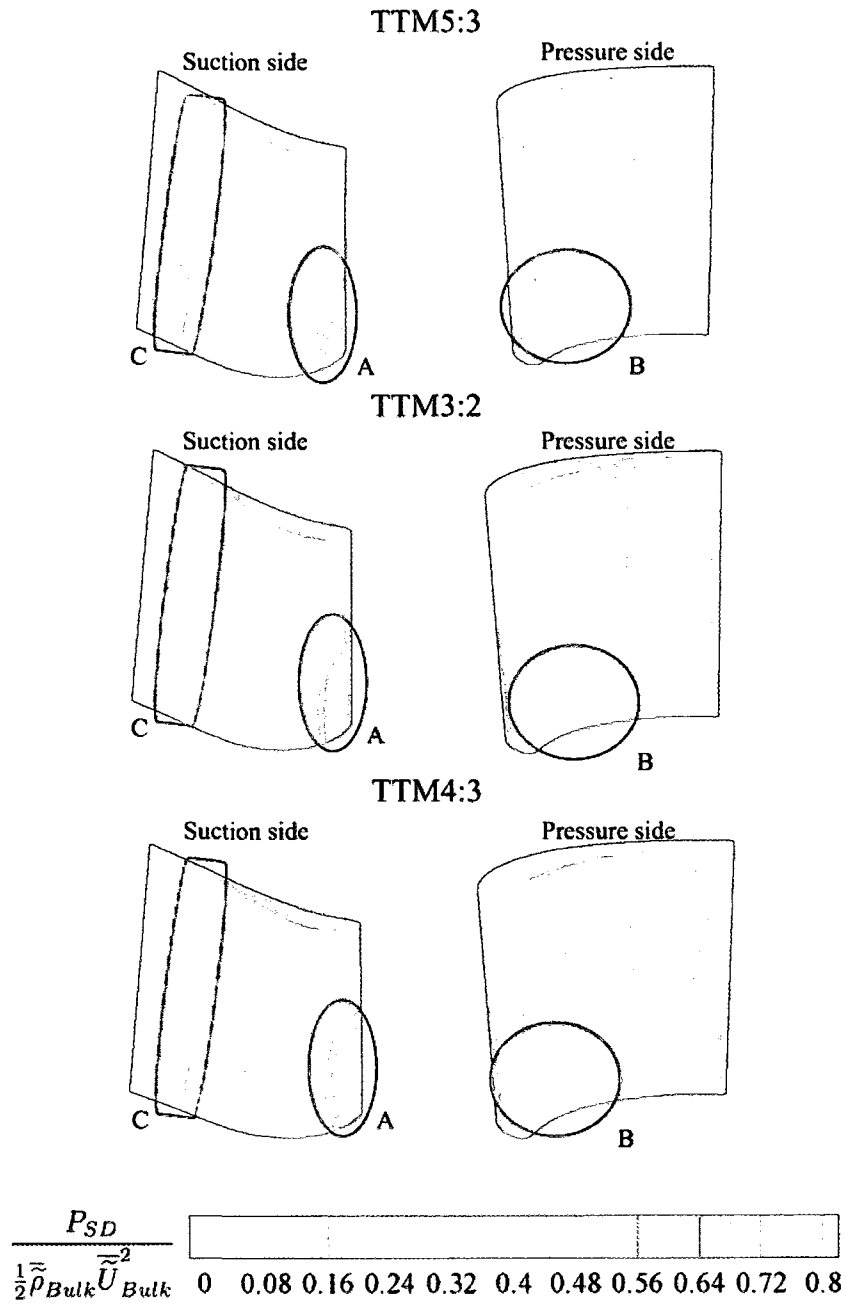


Figure 9.8: Pressure fluctuation amplitude over the surface of a rotor blade for the 3 cases.

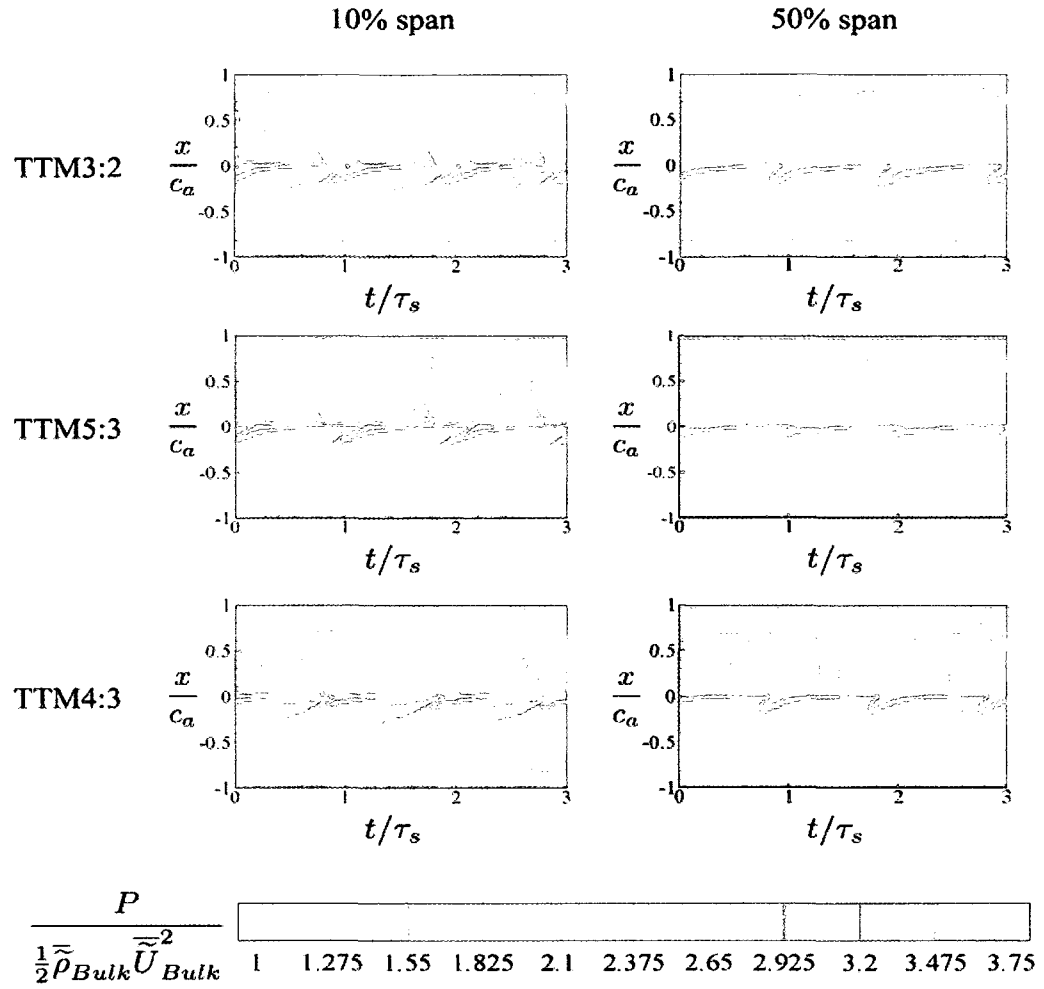


Figure 9.9: Isocontours of the time-varying surface pressure coefficient $P/\frac{1}{2}\bar{\rho}_{Bulk}\bar{U}_{Bulk}^2$ on the rotor blade at 10% and 50% spans; positive values of x/c_a indicate the pressure side of the rotor blade while negative values of x/c_a indicate the suction side of the rotor blade.

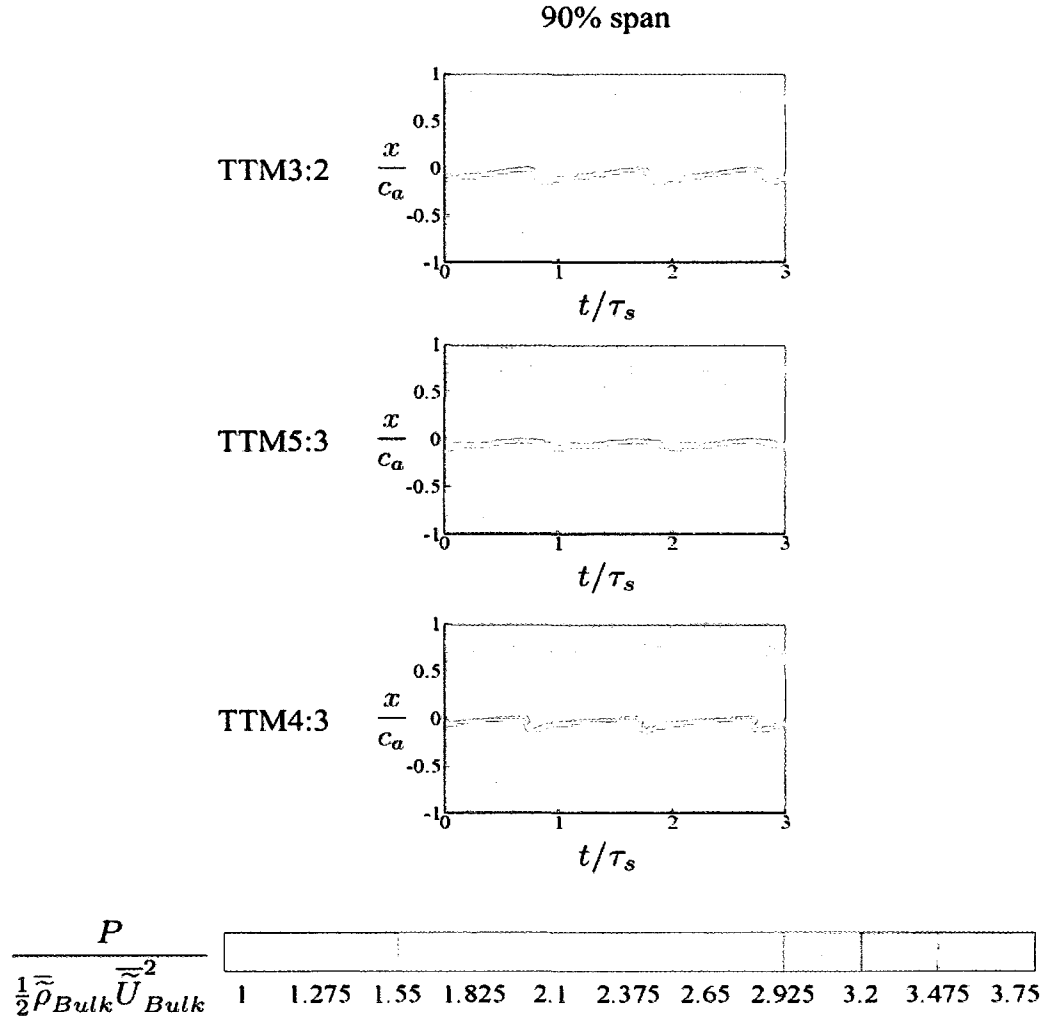


Figure 9.10: Isocontours of the time-varying surface pressure coefficient $P/\frac{1}{2}\bar{\rho}_{Bulk}\bar{U}_{Bulk}^2$ on the rotor blade at 90% span; positive values of x/c_a indicate the pressure side of the rotor blade while negative values of x/c_a indicate the suction side of the rotor blade.

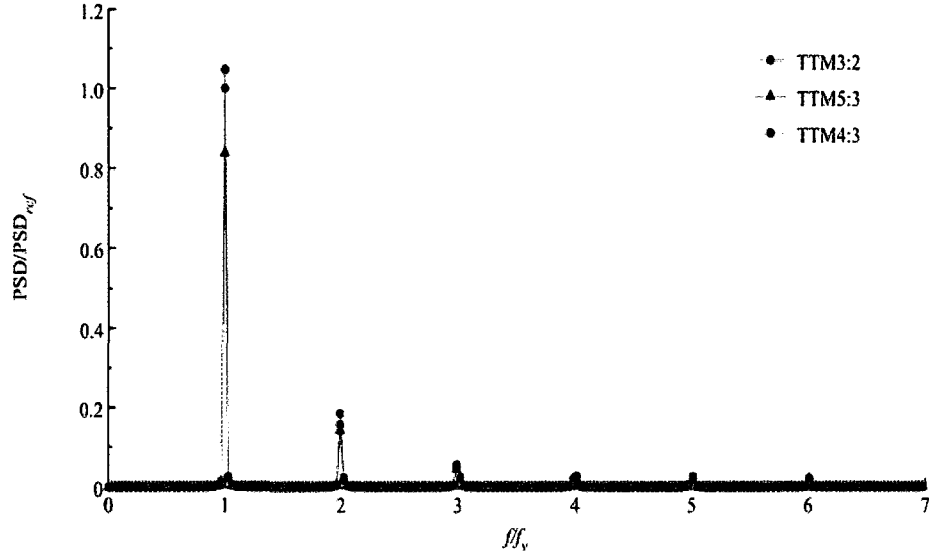


Figure 9.11: Power spectral density (PSD) of pressure fluctuations on the suction side of a rotor blade at 10% span and 10% axial chord.

Figures 9.11 and 9.12 show that, by comparison to the unscaled case (TTM3:2), both downsizing the rotor blade by 10% (TTM5:3) and upsizing the rotor blade by 12.5% (TTM4:3) left unchanged the fundamental vane passing frequency f_v and its harmonics, indicated by peaks of pressure fluctuations on the rotor blade. This is consistent with expectations, because rotor blade scaling did not affect the stator vane geometry. Nevertheless, upsizing increased slightly the magnitude of the pressure fluctuations on the rotor blade, while downsizing decreased the magnitude of pressure fluctuations or power spectral density (PSD) peak magnitude on the rotor blade by approximately 20% at the fundamental frequency. Therefore as the blade size decreased, the magnitude of the pressure fluctuations also decreased.

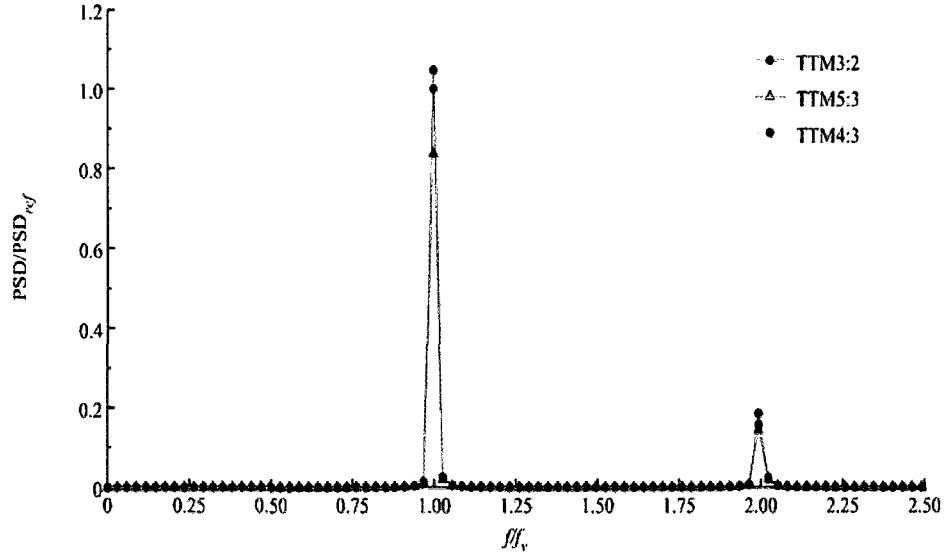


Figure 9.12: Power spectral density (PSD) of pressure fluctuations on the suction side of a rotor blade at 10% span and 10% axial chord. Detail of fundamental frequency and first harmonic.

9.4 Stator Vane Surface Pressure

In the same manner as the rotor blade surface static pressure, the stator vane surface static pressure was also normalized with the dynamic pressure $\frac{1}{2}\bar{\rho}_{Bulk}\bar{U}_{Bulk}^2$ defined using the bulk density and bulk velocity.

9.4.1 Time-Averaged Pressure

Pressure data on the stator vane were collected at 40 instances per rotor blade passing period. The time-averaged pressure distribution over the stator vanes, shown in Figure 9.13, is not significantly affected by scaling. The region of low pressure near the root and towards the trailing edge on the suction side of the stator vane (regions A in Figure 9.13) is slightly smaller in the TTM3:2 case than in the other two cases. Near the leading

edge of the vane on the pressure side of the stator vane, higher pressures appear to exist in the TTM3:2 simulations than in the TTM5:3 and TTM4:3 simulations.

9.4.2 Pressure Fluctuation Amplitude

The amplitudes of pressure fluctuations on the stator vane for the three cases considered are shown in Figure 9.14. The TTM3:2 case shows the highest amplitude of pressure fluctuations on the suction side of the vane, near the casing and along the trailing edge (regions A in Figure 9.14). The entire trailing edge region, on the suction side of the stator vane, experiences significantly lower fluctuation amplitudes in the TTM5:3 case than in either the TTM3:2 or TTM4:3 cases. The TTM4:3 and TTM3:2 cases have similar distributions of the pressure fluctuation amplitude with an exception near the hub (regions B in Figure 9.14), where the TTM4:3 case shows higher pressure fluctuation amplitudes than the TTM3:2 case.

9.4.3 Unsteady Pressure

Figures 9.15 and 9.16 show plots of the time varying pressure $P/\frac{1}{2}\bar{\rho}_{Bulk}\bar{U}_{Bulk}^2$ along 60% of the suction surface of a stator vane, near the trailing edge, at 10%, 50% and 90% spans. Time has been normalized by the rotor vane passing period τ_r .

The effect of scaling is clearly seen at all spanwise locations. The TTM4:3 simulation results show the lowest space-time average pressure at all spanwise locations, while the highest space-time average pressure distribution over the stator vane trailing edge is found in the TTM5:3 case. In addition, the results show that the largest gradient in the pressure change over time at all spanwise locations is seen in the TTM4:3 case. There is also a

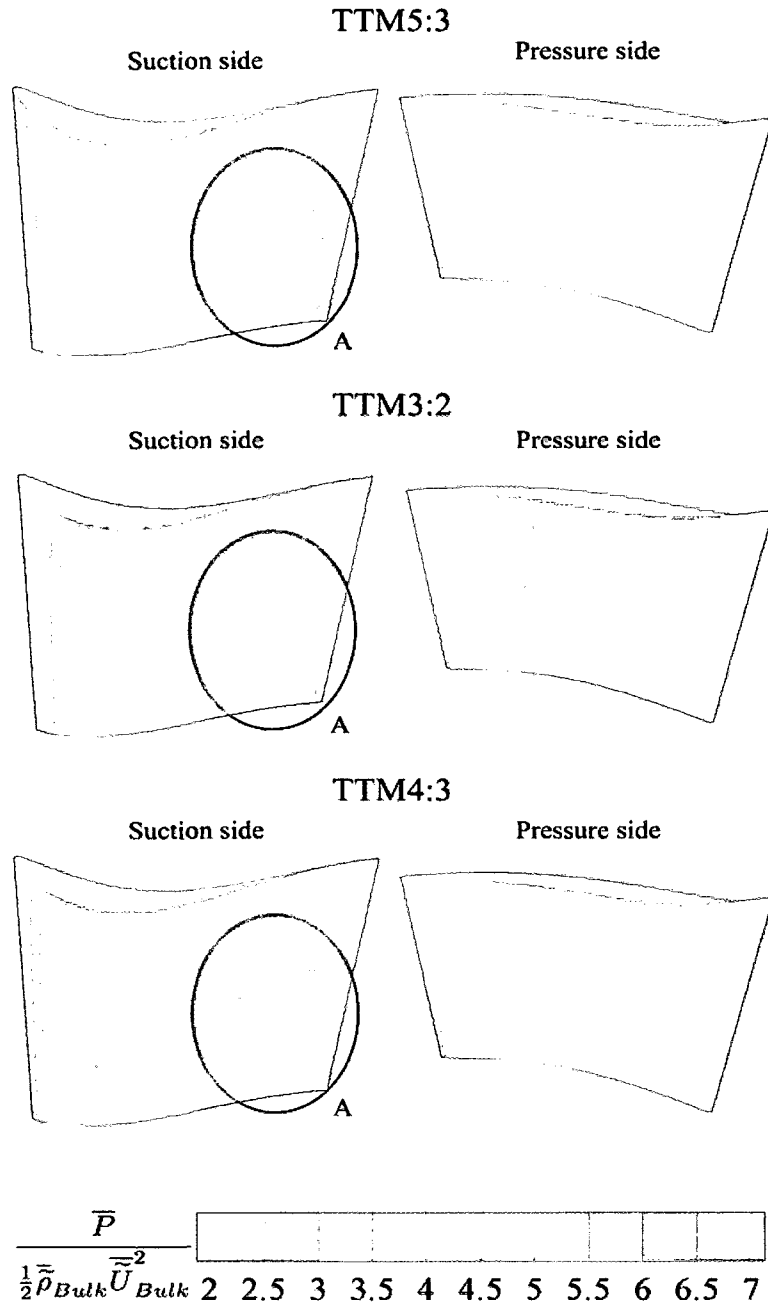


Figure 9.13: Time averaged surface pressure $\bar{P}/\frac{1}{2}\bar{\rho}_{Bulk}\bar{U}_{Bulk}^2$ distribution on a stator vane for the three cases considered.

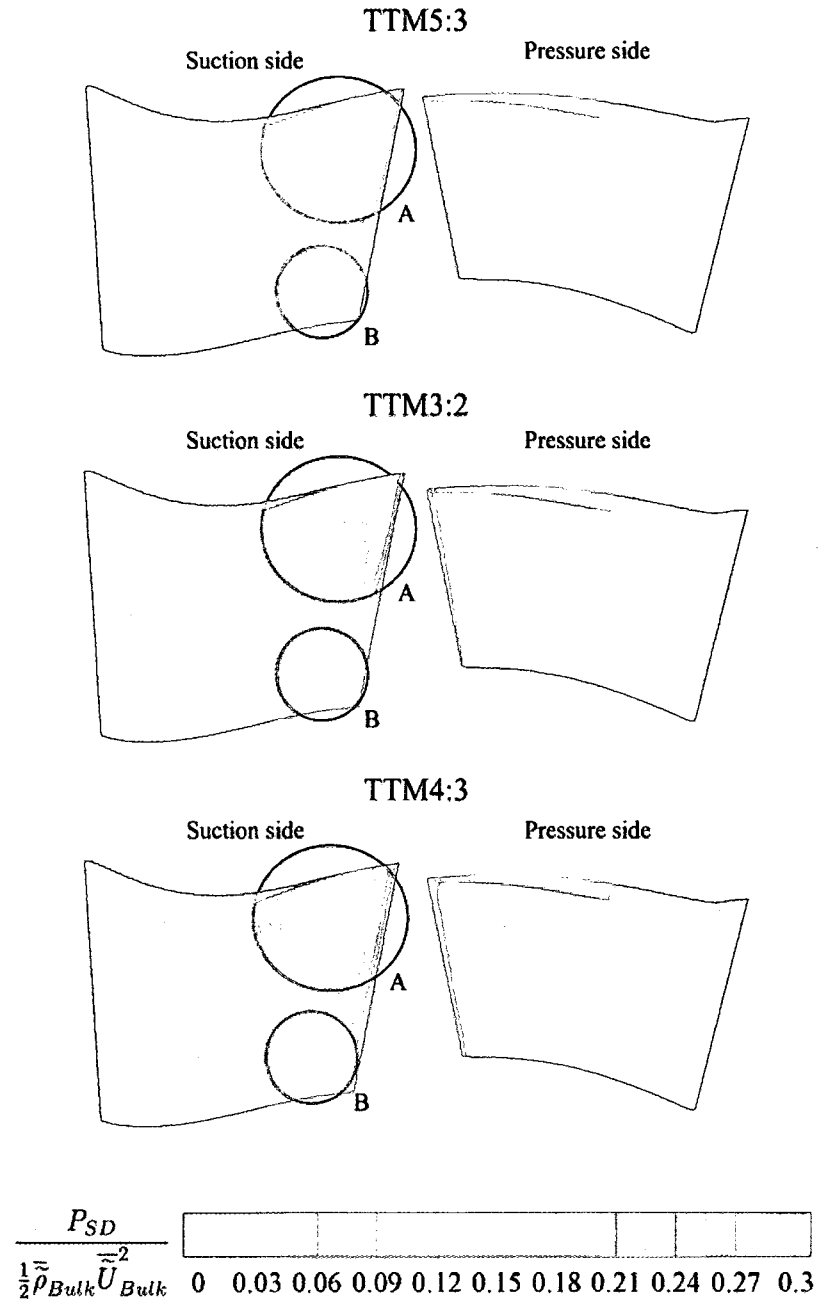


Figure 9.14: Pressure fluctuation amplitude $P_{SD}/\frac{1}{2}\bar{\rho}_{Bulk}\bar{U}_{Bulk}^2$ over the surface of a stator vane for the 3 cases.

distinct concentration of high pressure, near the trailing edge at 90% span, in all the cases. The sharp rise in pressure over time, seen in all the cases at 10% span, may be a result of the stator vane trailing edge shock. The sharp rise is only seen at $t/\tau_r = 0, 1, 2, 3$, because this is when the rotor blade leading edge is in-line with the stator vane trailing edge. At this point in time, the shock is strongest and produces a sharp rise in the pressure. This pressure rise is more gradual in the TTM3:2 and TTM5:3 cases as seen in Figure 9.15. As a result, increasing the blade size appears to dramatically strengthen the shock.

Figures 9.17 and 9.18 show that, compared to the unscaled case, the peak frequencies of pressure fluctuations on the stator vanes increased when the rotor blade was downsized and decreased when it was upsized. Again, this is consistent with expectations based on the frequencies of blades passing past the trailing edge of each vane for the three cases considered. The magnitudes of pressure fluctuations on the vane surface decreased slightly by upsizing and dramatically by downsizing of the rotor blade. The pressure fluctuations on the surface of the stator vane are caused by a periodic pressure wave that forms just downstream of the rotor blade leading edge (on the suction side of the blade) and just after passing through the stator vane trailing edge shock. This pressure wave moves upstream from the leading edge of the rotor blades and impacts the suction side of the stator vane near the trailing edge.

9.5 In-Flow Pressure

Pressure contours on selected surfaces inside the flow (i.e., not on the surfaces of a blade or a vane), were non-dimensionalized using the space-time mass weighted total inlet

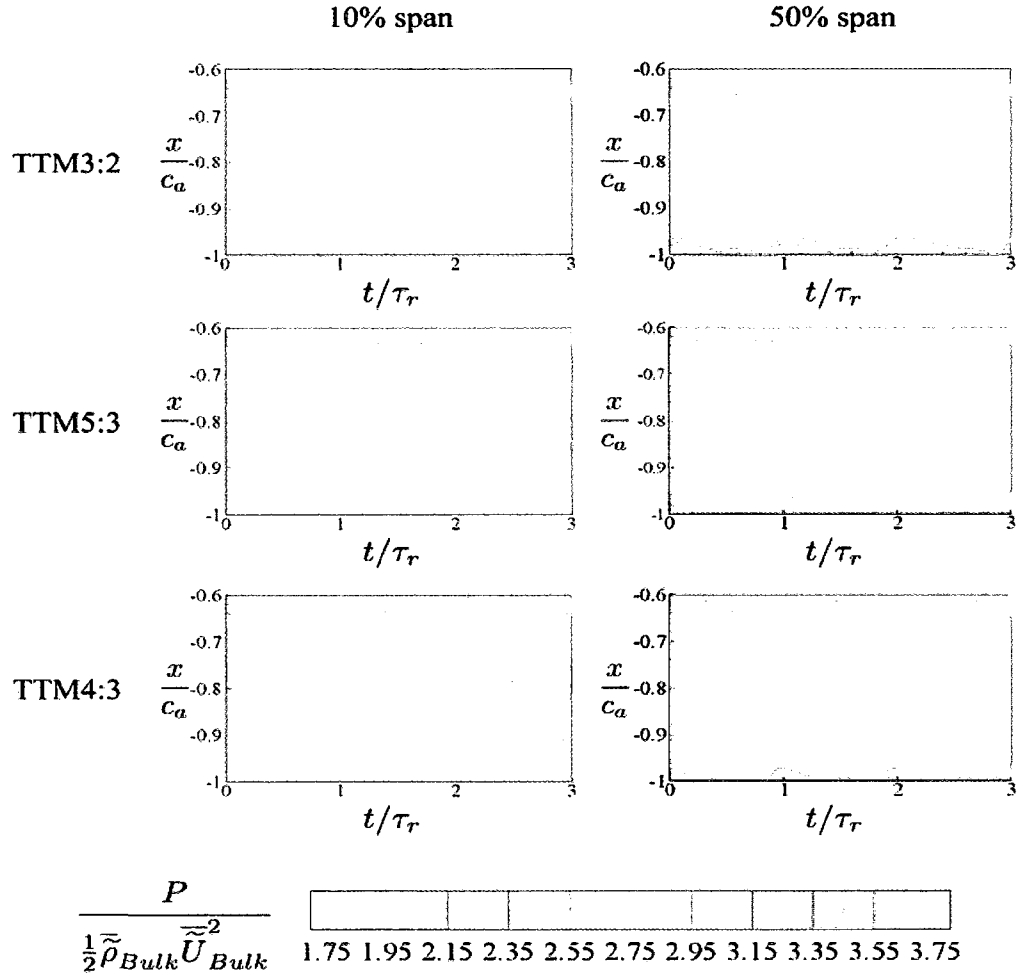


Figure 9.15: Isocontours of the time-varying surface pressure coefficient $P/\frac{1}{2}\bar{\rho}_{Bulk}\bar{U}_{Bulk}^2$ on the stator vane at 10% and 50% spans and from 60% to 100% of the stator vane axial chord on the suction side of the stator vane.

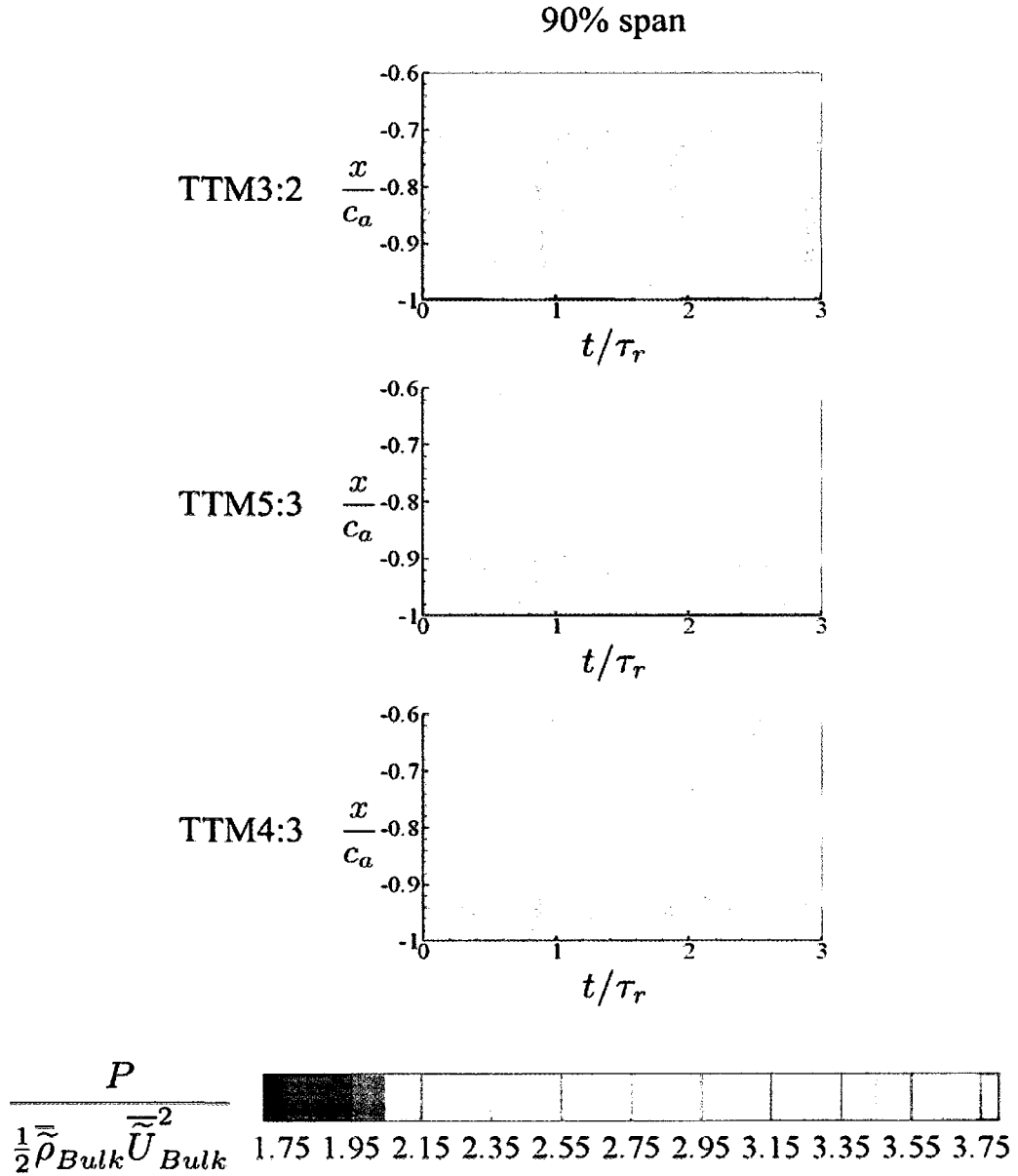


Figure 9.16: Isocontours of the time-varying surface pressure coefficient $P/\frac{1}{2}\bar{\rho}_{Bulk}\bar{U}_{Bulk}^2$ on the stator vane at 90% span and from 60% to 100% of the stator vane axial chord on the suction side of the stator vane.

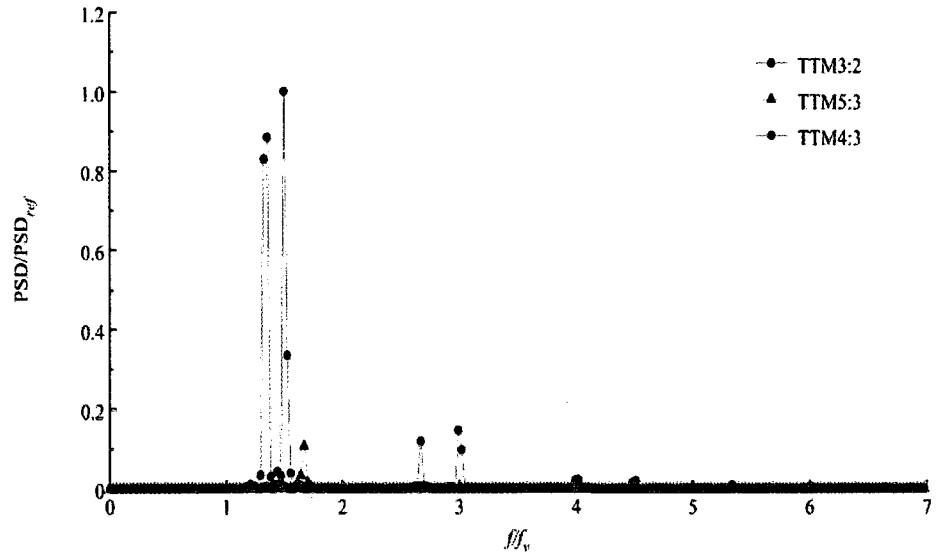


Figure 9.17: Power spectral density (PSD) of pressure fluctuations on the suction side of a stator vane at 10% span and 85% axial chord.

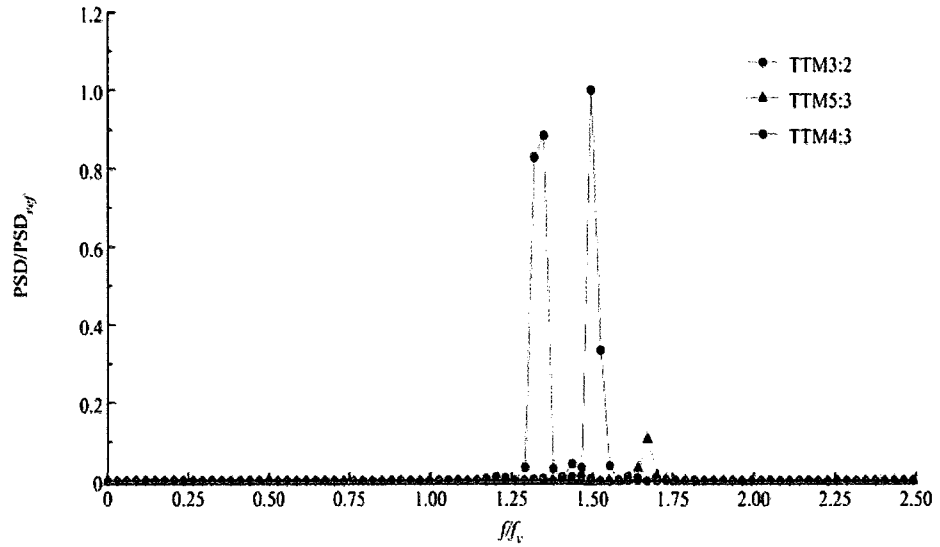


Figure 9.18: Power spectral density (PSD) of pressure fluctuations on the suction side of a stator vane at 10% span and 85% axial chord. Detail of fundamental frequency.

pressure $\overline{P}_{01}$.

9.5.1 Pressure Fluctuation Amplitude

Isocontours of pressure fluctuation amplitude are plotted on three axial planes in Figures 9.19 to 9.21. Results from the TTM5:3 simulations show much lower amplitudes of pressure fluctuations on plane B1 than in the other two cases. All three cases show a high-fluctuation region near the tip and just towards the suction side of the stator vane trailing edge. On the remaining axial planes, the average amplitudes of pressure fluctuations are comparable. On plane B3 the regions of high pressure fluctuations are similar in all cases. However, the TTM4:3 case shows regions of low pressure fluctuations near the rotor tip on the pressure side of the rotor blade and near the root on the suction side of the blade where the other two cases show regions of high pressure fluctuations. On plane C1, the amplitude of pressure fluctuations increases in the tip gap regions as the blade size increases. The additional regions of fluid with a high amplitude of fluctuations may be due to the increased axial length of the tip gap, which permits larger amounts of fluid to flow through the tip gap. In addition, the amplitude of pressure fluctuations drops as air flow through the rotor is essentially isolated near the hub and tip gap regions. All the cases exhibit comparable amplitudes of pressure fluctuations along plane C1.

9.6 Turbine Losses

Losses were quantified using entropy generation as defined by Equation 3.18 referred to as the entropy loss coefficient.

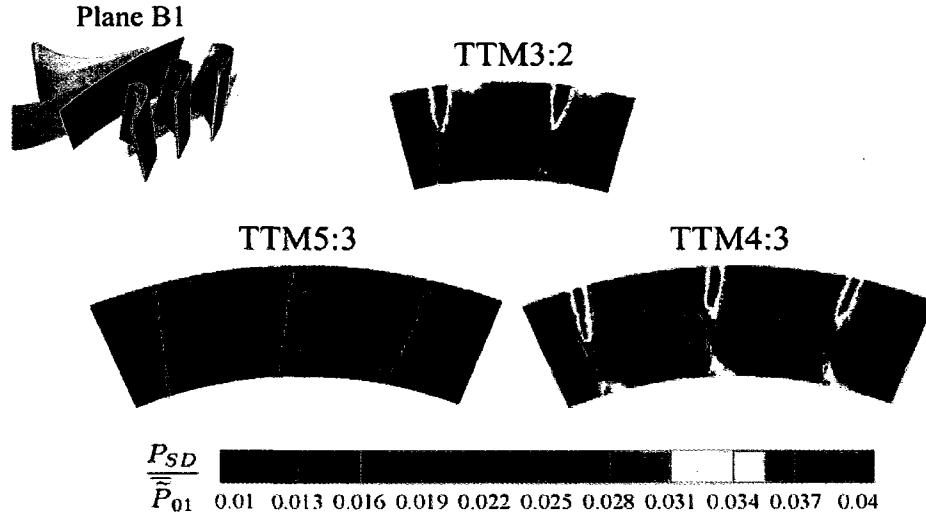


Figure 9.19: Predicted isocontours of the amplitude of pressure fluctuations $P_{SD}/\bar{P}_{01}$ on plane B1.

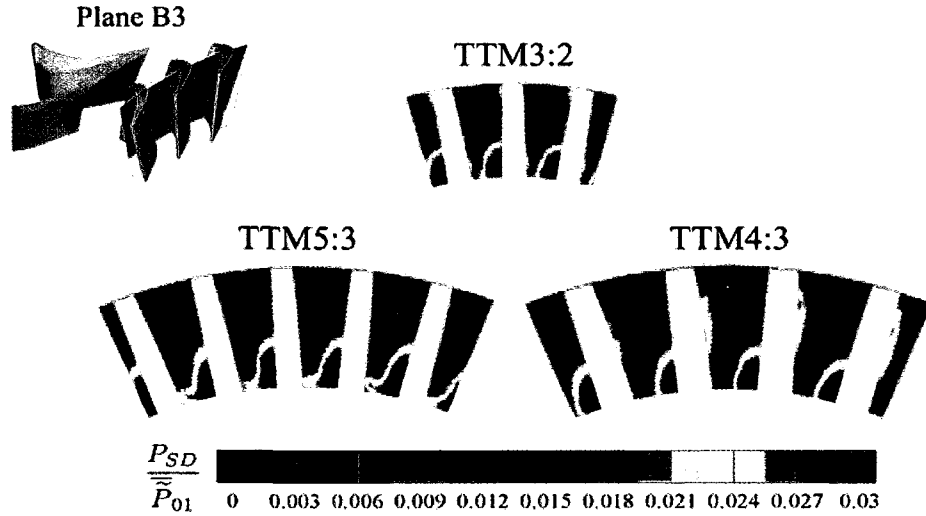


Figure 9.20: Predicted isocontours of the amplitude of pressure fluctuations $P_{SD}/\bar{P}_{01}$ on plane B3.

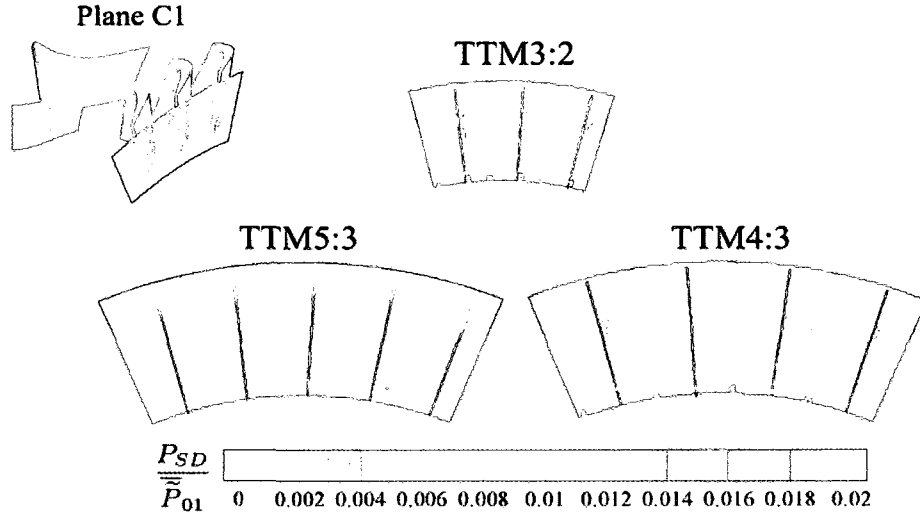


Figure 9.21: Predicted isocontours of the amplitude of pressure fluctuations $P_{SD}/\bar{P}_{01}$ on plane C1.

9.6.1 Time-Averaged Losses

Isocontours of the entropy loss coefficient along two axial planes are presented in Figures 9.22 and 9.23. It is clear that TTM4:3 simulations exhibited higher losses than the other two cases. While losses in the tip leakage vortex were the same in all cases, losses in the wake and near the hub increase as the blade size increases from the TTM5:3 case to the TTM4:3 case. This may be attributed to the increase in the thickness of the blade, which results in a larger wake in the TTM4:3 case when compared to the other two cases. Because wake fluid experiences shear which leads to high losses, the thicker the blade is, the larger the wake would be and it would contain more high-loss fluid ($\bar{\zeta}_s \geq 4$). The blade size does not seem to affect the start of the formation of the tip leakage vortex. In all cases, there is no tip leakage vortex visible until plane B4 at approximately 90% span.

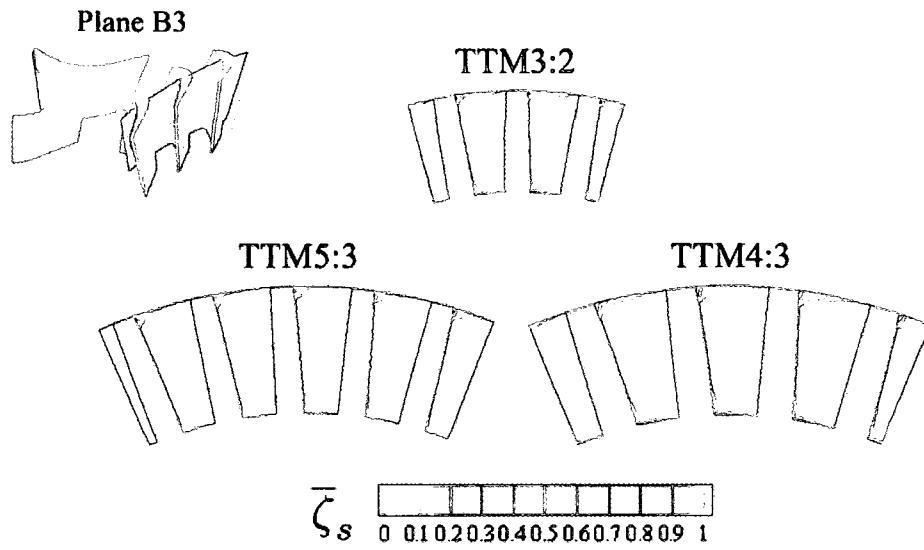


Figure 9.22: Predicted isocontours of entropy loss coefficient $\bar{\zeta}_s$ on plane B3.

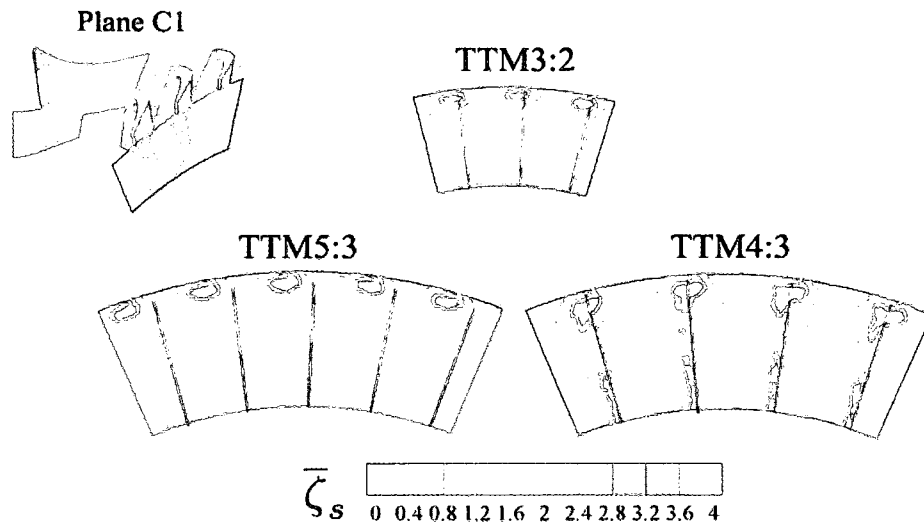


Figure 9.23: Predicted isocontours of entropy loss coefficient $\bar{\zeta}_s$ on plane C1.

9.7 Turbulent Kinetic Energy

The following plots include only the turbulent kinetic energy obtained from the turbulence model. In-cycle velocity fluctuations were ignored, as only small-scale velocity fluctuations were of interest. In addition, as discussed earlier (chapter 8, section 8.2), ensemble averaging showed that cycle-to-cycle velocity fluctuations were negligible. Therefore, cycle-to-cycle velocity fluctuations were ignored as well. Normalization of the turbulence kinetic energy was accomplished using the square of the bulk velocity $\overline{U}_{Bulk}^2$.

9.7.1 Time-Averaged Turbulent Kinetic Energy

Figure 9.24 shows predicted isocontours of the turbulent kinetic energy along plane C1. When comparing the three cases, the TTM4:3 case exhibits higher turbulence kinetic energy in the rotor blade wakes. As was discussed previously, the higher turbulent kinetic energy is due to the greater blade thickness in the TTM4:3 case and a shorter distance between plane C1 and the rotor blade trailing edge. Therefore, turbulent kinetic energy in the wake increases as the blade size increases. The blade size does not seem to affect the turbulent kinetic energy peak magnitude in the tip leakage vortices.

9.8 Vorticity

The vorticity magnitude was normalized by the ratio $\overline{U}_{Bulk}/c$ of the bulk velocity and the midspan rotor chord.

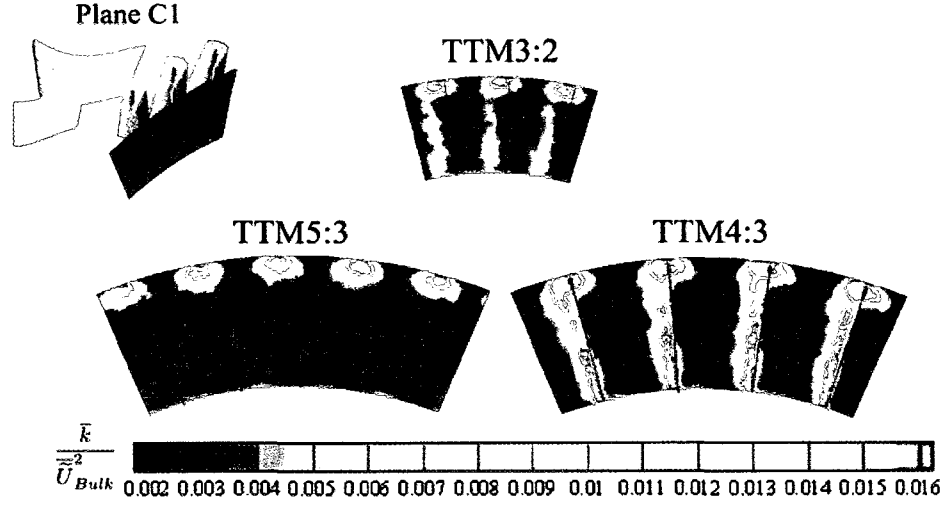


Figure 9.24: Predicted isocontours of time-averaged turbulent kinetic energy $\bar{k}/\bar{U}_{Bulk}^2$ on plane C1.

9.8.1 Time-Averaged Vorticity

Isocontours of the time-averaged vorticity magnitude on plane C1 are shown in Figure 9.25. Regions of relatively high vorticity ($\bar{\omega}_{mag}c/\bar{U}_{Bulk} \geq 2$) clearly define the wakes of the rotor blades and also correspond to regions of high turbulent kinetic energy and loss (note that the gradient transport model employed in the present simulation connects the turbulent kinetic energy to the mean velocity gradient, which also contributes to the mean vorticity). Near the root of the blades in the rotor blade wakes there are regions of high vorticity ($\bar{\omega}_{mag}c/\bar{U}_{Bulk} \geq 4$) not likely associated with the boundary layers and which could possibly be associated with passage vortices. The sizes of these regions are largest in the TTM4:3 case and much smaller in the TTM5:3 case; the latter observation is consistent with the very low turbulent kinetic energy on plane C1 in the TTM5:3 case. Concerning the observed differences between the cases, one may note that the larger blade

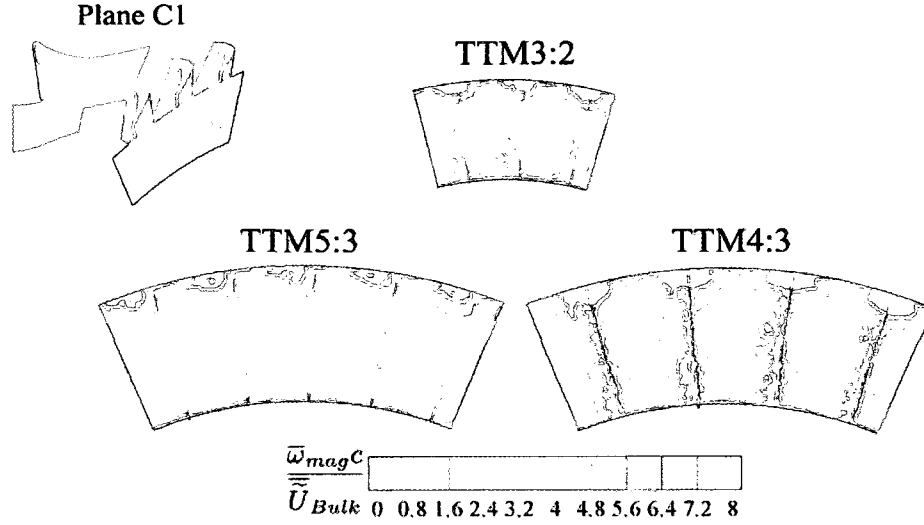


Figure 9.25: Predicted isocontours of time-averaged dimensionless vorticity magnitude $\bar{\omega}_{mag}c/\bar{U}_{Bulk}$ on plane C1.

in the TTM4:3 case may be the reason for the possible increase in the size of the passage vortices as discussed previously. The observed region of high vorticity ($\bar{\omega}_{mag}c/\bar{U}_{Bulk} \geq 5$) along the right side periodic boundary is attributed to an inaccuracy resulting from the calculation of vorticity along periodic boundaries in FLUENT 6.3.26.

Instantaneous isosurfaces with $Q_m = 1$ used to identify coherent structures are shown in Figure 9.26. Many small structures are noticeable near the leading edges of the rotor blades in all cases. In addition, structures are also visible near the hub at the leading edges of the rotor blades and in the rotor passage, which may be evidence of the existence of hub passage vortices. The sizes of the hub passage vortices seem to increase as the blade size increases. In all cases, the large tip leakage vortices are visible up to some downstream location (the same for all cases), at which they cease to be identified by the chosen value of Q_m . The location of formation of the tip leakage vortex appears to be insensitive to

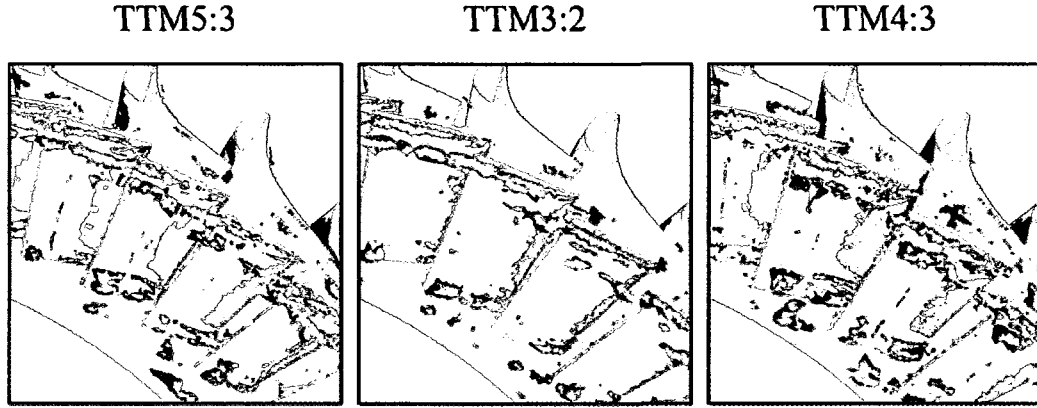


Figure 9.26: Instantaneous isosurfaces with $Q_m = 1$ used to identify coherent structures.

scaling, in conformity with observations based on the entropy loss coefficient. The lengths and diameters of the vortices appear to be the same in all cases as well, in support of observations based on time-averaged entropy loss coefficient plots. This indicates that the size of the tip leakage vortex is insensitive to scaling.

9.9 Unsteady In-Flow Properties

This section focuses on the unsteady effects that occur away from the walls (unsteady effects on the rotor blades and stator vanes have been ignored).

9.9.1 Pressure

The unsteady static pressure variations on the 10% span plane and on planes B1 and C1 are shown in Figures 9.27, 9.28 and 9.29, respectively.

On the 10% span plane, a stator vane trailing edge shock is visible in all cases. The

motion of the rotor blades, relative to the stator vanes, tends to create an instantaneous throat section (at the same region as the shock in all cases at times $t/\tau_r = 0$ and $t/\tau_r = 0.25$) when the rotor blade leading edge has just passed the stator vane trailing edge ($t/\tau_r = 0.25$). Because the flow is transonic just upstream of this instantaneous throat section on the suction side of the stator vane, the blade rb1 and the stator vane axially upstream create a supersonic diffuser. This causes the flow to decelerate and the pressure to increase. High local pressures are seen just after the shock when a rotor blade leading edge and a stator vane trailing edge are aligned such that the flow area between them is minimized. Once this area is increased after the rotor blade moves forward, the pressures after the shock are reduced. When comparing the regions of high pressure just after the shock, between rb1 and the upstream stator vane at $t/\tau_r = 0.25$, the TTM4:3 case shows higher pressures than the other two cases near the leading edge of rb1. As a result, upscaling leads to a slight increase in the pressure rise, whereas downscaling has little effect on the pressure rise through the throat.

Figures 9.27, 9.28 and 9.29 also show a pressure wave that forms near the leading edge of the rotor blade (on the suction side of the blade) and closely after passing through the stator vane trailing edge shock. This periodic pressure wave is thought to be responsible for the pressure fluctuations on the surface of the stator vane. This pressure wave moves upstream, as seen by a distortion in the shape of the low pressure region on the suction side of the stator vane (regions A in Figure 9.27), from the leading edge of the rotor blades and impacts the suction side of the stator vane near the trailing edge causing pressure fluctuations as seen in Figure 9.14.

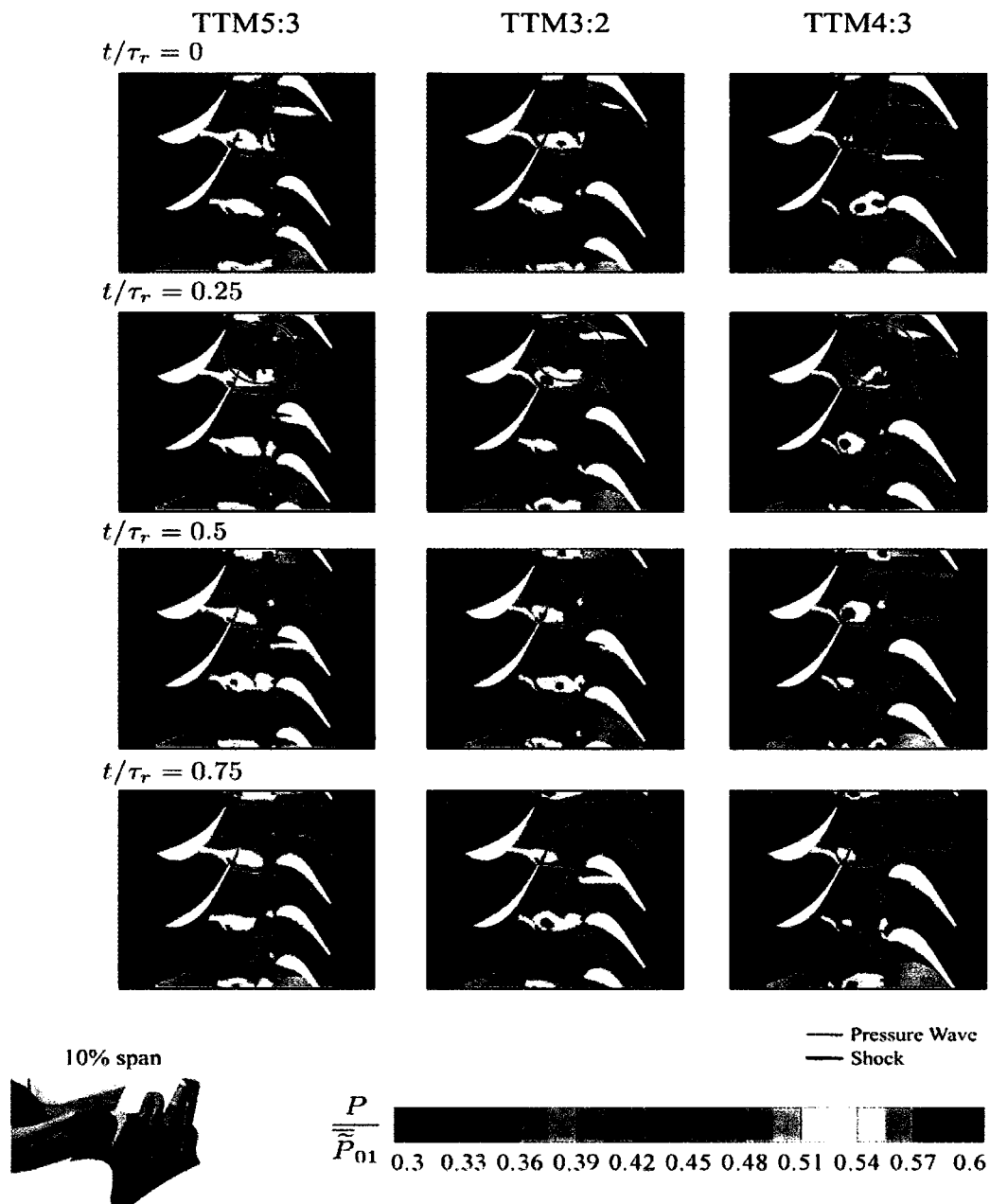


Figure 9.27: Predicted isocontours of pressure $P/\bar{P}_{01}$ at 10% span. Shock is graphically approximated by a black line that was manually added.

Figure 9.28 shows how the size of the low pressure region $P/\bar{P}_{01} < 0.36$, on the suction side of the stator vane near the trailing edge, fluctuates as the rotor blade passes. As $rb1$ approaches the next stator vane, the upstream moving pressure wave causes the pressure to drop, as seen in Figure 9.28. By comparison, the TTM5:3 cases show very little effect of the pressure wave. The TTM3:2 case shows a slight effect of the pressure wave, while the TTM4:3 case shows a noticeable effect. In all cases, the effect of the pressure wave is seen as a decrease in the local pressure in the regions of high pressure $P/\bar{P}_{01} > 0.59$ in Figure 9.28. The effect of the instantaneous throat is also seen in all cases in Figure 9.28. Just after a rotor blade leading edge passes by a stator vane trailing edge, the pressure increases near the respective stator vane trailing edge.

There is a noticeable increase in the pressure near the suction side of the rotor blade at various times. When comparing Figure 9.29 and Figure 9.1, which show the stator wakes in relation to the rotor blades, one can see that the increase in pressure near the suction side of a rotor blade occurs when a stator wake impinges directly onto the leading edge of the same blade. Comparison of the three cases showed that scaling has little effect on this phenomenon.

9.9.2 Losses

Instantaneous isocontour plots of the entropy loss coefficient ζ_s for plane C1 (Figure 9.2) are shown in Figure 9.30. Time was non-dimensionalized using the rotor passing period τ_r .

Referring to the distribution of losses downstream of the rotor (Figure 9.30), as was seen in the time-averaged plots, increasing the blade size increases the size and magnitude

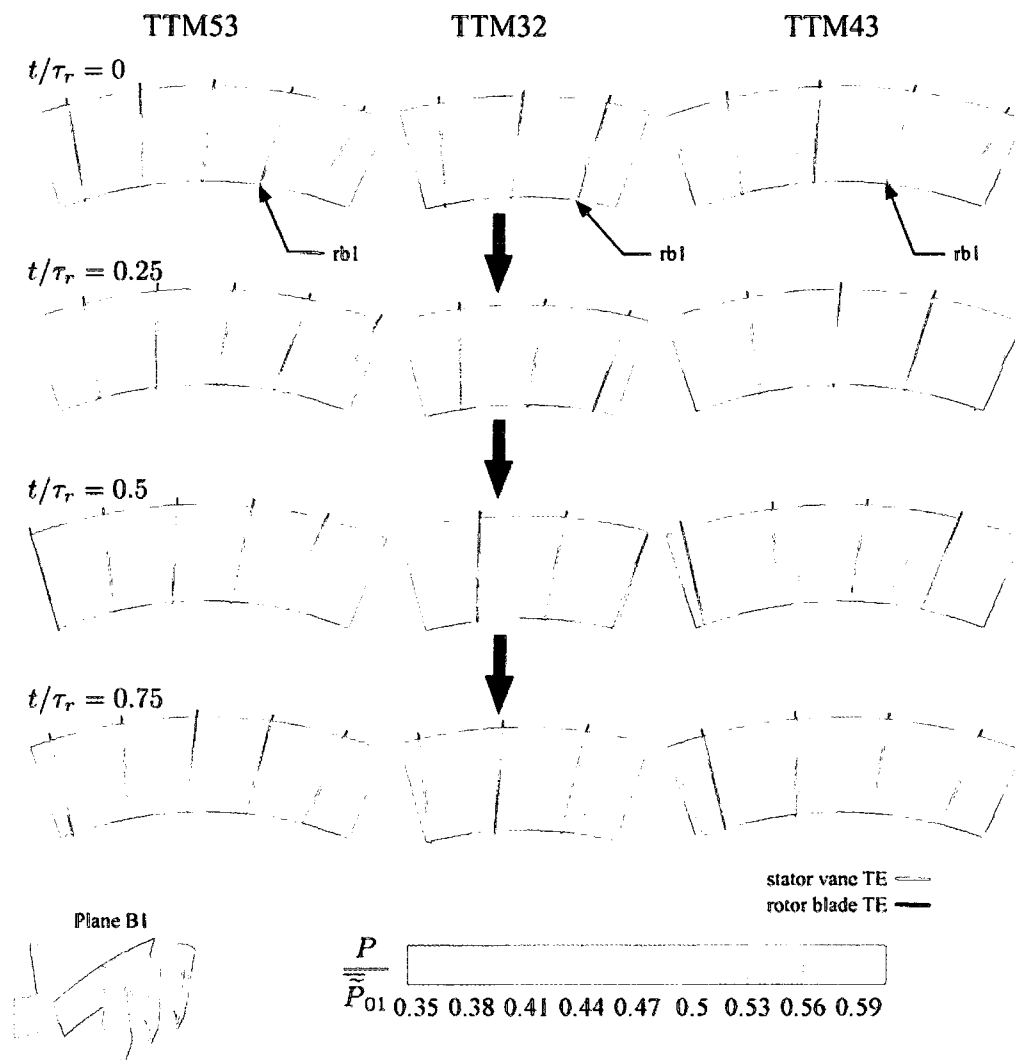


Figure 9.28: Predicted isocontours of pressure $P/\bar{P}_{01}$ on plane B1.

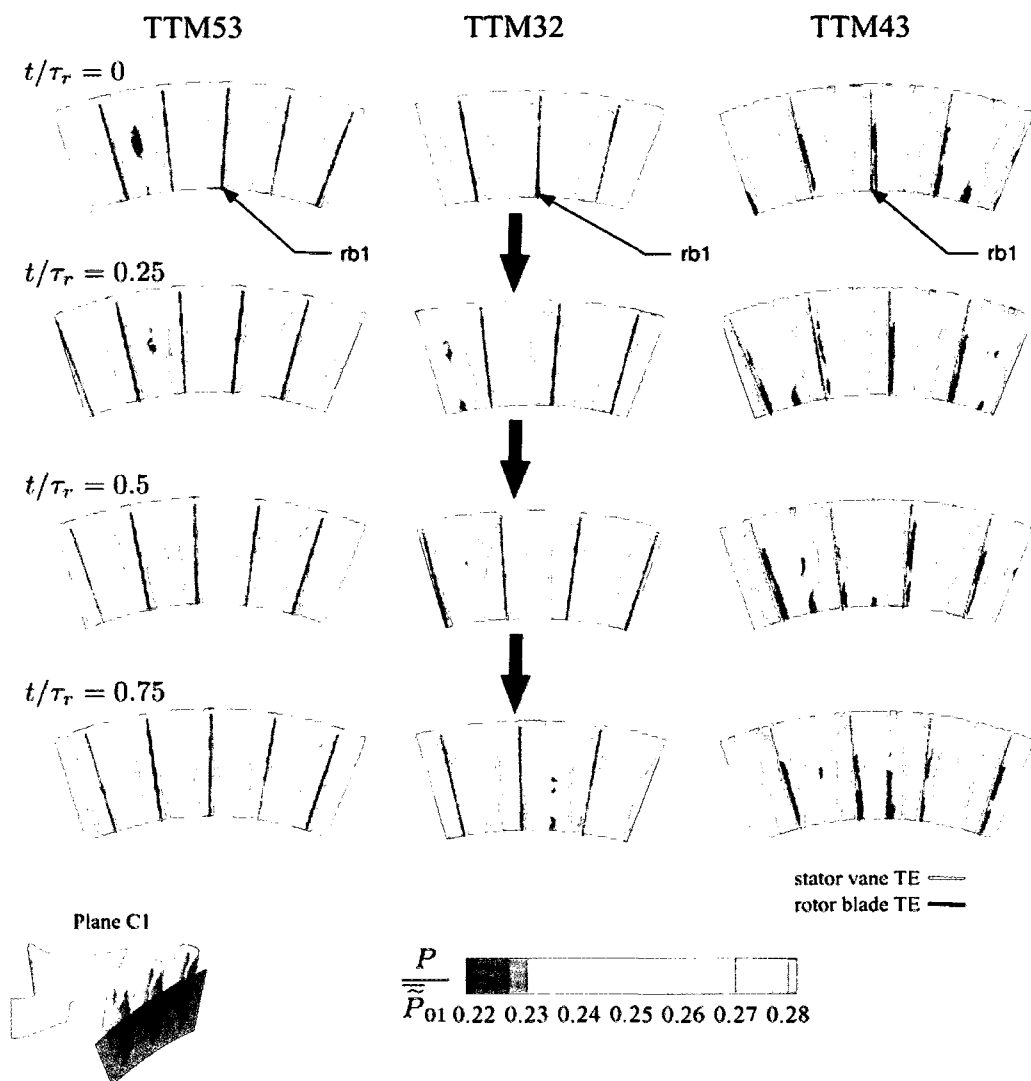


Figure 9.29: Predicted isocontours of pressure $P/\bar{P}_{01}$ on plane C1.

of losses in the wake regions (indicated by regions behind the rotor blade with $\zeta_s \geq 1.6$). In addition to changes in the rotor wakes, differences in the free-stream losses between the scaled and unscaled cases are also noticeable. With regard to the unsteadiness of the entropy loss coefficient, there is no affect of the blade position relative to the vane on the entropy loss coefficient distribution. The entropy loss coefficient distribution remains fixed relative to the rotor blade. However, increasing the blade size decreases the free stream entropy loss between the blades. Losses are reduced as the blades size increases because the angle between blades increases as the blade increases. Therefore, there is more space between the blades for fluid to flow and as a result losses decrease.

9.10 Concluding Remarks

In the present work, solely because of the availability of experimental results, the “unscaled” TTM3:2 geometry was used as the reference, although it requires a smaller computational domain than the “scaled” cases TTM5:3 and TTM4:3. In practice, the situation would have likely been reversed and one would perform simulations using TTM3:2 as a scaled surrogate for the other two geometries. To reflect reality, in this section alone, cases TTM5:3 and TTM4:3 will be referred to as unscaled cases and case TTM3:2 will be referred to as the scaled case.

Comparison of the computational requirements of the three simulations showed that the scaled case (TTM3:2) required approximately 1/3 of the simulation time per time step than the other two cases. As a result, scaling which reduced the computational domain volume by approximately 33%, reduced the computational time by twice that percent.

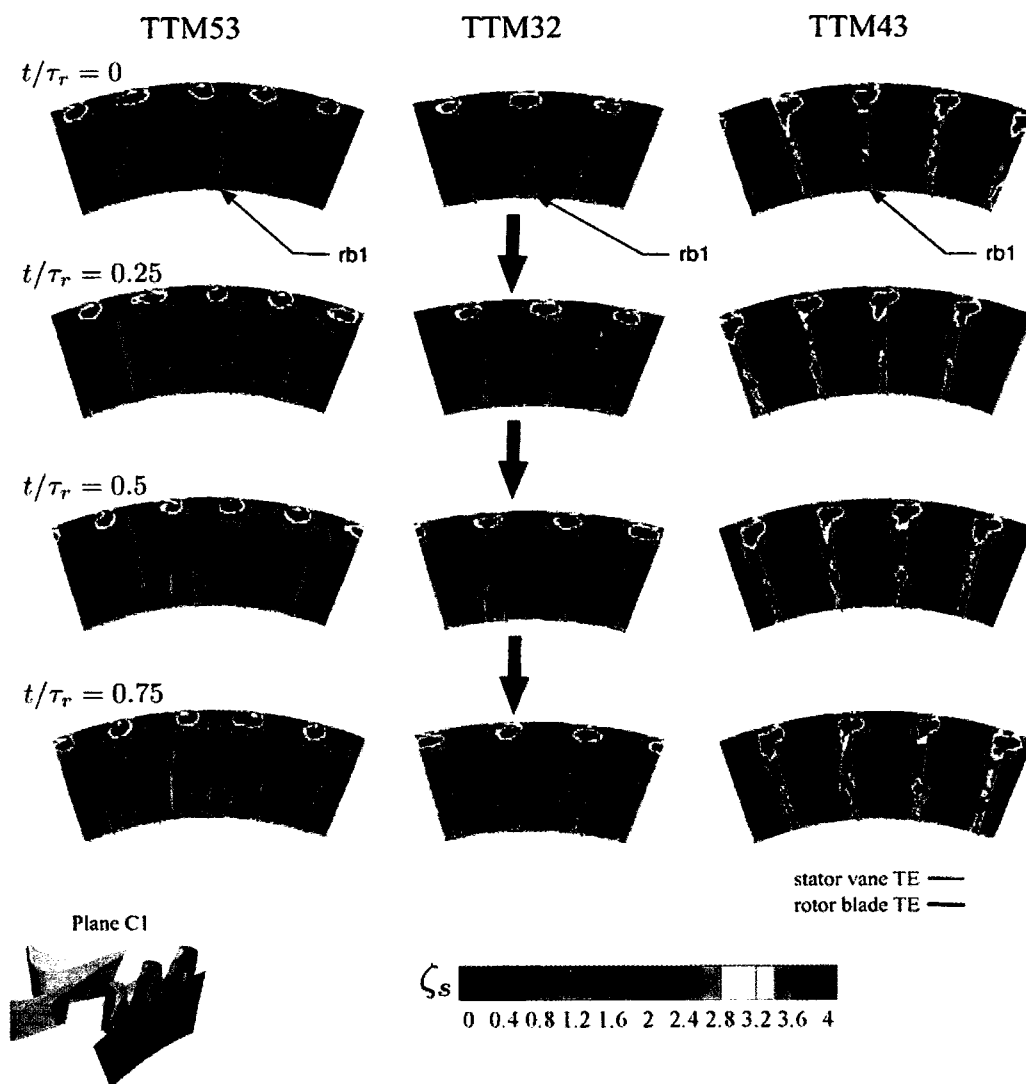


Figure 9.30: Predicted isocontours of the entropy loss coefficient ζ_s on plane C1.

Results showed that scaling left unchanged the fundamental vane passing frequency and its harmonics because rotor blade scaling did not affect the stator vane geometry. Nevertheless, scaling decreased the magnitude of the pressure fluctuations on the rotor blade by approximately 20% when downscaling and slightly increased the magnitude of pressure fluctuations when upscaling. Therefore, as the blade size increased, blade surface pressure fluctuations were intensified.

The peak frequencies of pressure fluctuations on the stator vanes varied depending on scaling. When the number of rotor blades was reduced during scaling, the peak frequencies of pressure fluctuations decreased and, when the number of rotor blades increased, the peak frequencies of pressure fluctuations increased accordingly. Again, this is consistent with expectations based on the frequencies of blades passing past the trailing edge of each vane for the three cases considered. The magnitudes of pressure fluctuations on the vane surface decreased slightly by upsizing and dramatically by downsizing of the rotor blade. In contrast to the rotor blade, as the factor Λ_{sc} more closely approached an integer number, pressure fluctuations on the stator vanes decreased.

While cases with small scaling ratios were not examined presently, these results are compatible with previous work (Arnone and Pacciani (1996)) suggesting that, in numerical simulations of gas turbine operation, scaling should not exceed 1% if one wants to avoid strong departures from the predicted pressure fluctuations of the unscaled geometry.

The present results demonstrate that blade scaling affects significantly the magnitude of unsteady pressure fluctuations, which influence the level of vibratory stresses on the blades. Scaling was found to lead to a significant underestimation or overestimation of

the pressure fluctuations on the rotor blades and stator vanes, possibly leading to underprediction or overprediction of blade flutter and vibration amplitudes. Moreover, scaling has been found to affect the size and magnitude of losses in the rotor blade wakes. Scaling did not significantly affect turbine power output or turbine efficiency.

During an initial phase of the present study, rotor blade scaling was enforced without any modification to the rotor blade tip profile. Because the blade height is not constant but it increases downstream, scaling created a non-uniform size of the tip gap, whose effect interfered with the effect of scaling. This problem was corrected by adjusting the rotor blade tip profile so that the tip gap size was uniform and equal for all cases. Although not part of the main study of scaling effects, it seems instructional to present results obtained with the variable tip gap geometries, as these show a profound effect of the tip gap size on the turbine performance and loss. Details of these results can be found in Appendix A.

While the turbomachine studied was designed for use in a lab and lacked several elements of actual jet engines, the present results are deemed to be of relevance to commercial turbomachines. The absence of a combustion chamber, and the associated relatively low turbine inlet temperature, are not of great concern, because the gas properties do not vary significantly with temperature. Therefore, the fluid properties in the present simulations would not be expected to differ significantly from those in actual turbines. If a second stage or other downstream devices were present in the geometry, as may be expected in actual high-pressure turbines, one may expect that the pressure fluctuations on the trailing edge of the rotor blades would have been higher. However, because the upstream stator had

little effect on the pressure fluctuations aft of the 50% axial chord location of a rotor blade, the existence of a downstream stator is not expected to affect the pressure fluctuations near the leading edge of the rotor blade. Another difference from commercial engines is that the present turbine operated at relatively low rotational speeds. Increasing the rotational speed should only increase the fundamental frequency of pressure fluctuations, and is not expected to influence significantly the observed trend concerning the effect of scaling on the pressure fluctuation amplitude.

Chapter 10

Conclusions and Recommendations for Future Work

10.1 Conclusions

As a result of the turbulence model study on the 2-D T106 turbine cascade, it was determined that all models predicted comparable pressure distributions over the turbine blade except near the trailing edge region. The SST and RSM models both captured flow separation near the trailing edge that was missed by the S-A and $k - \varepsilon$ based turbulence models. However, it was found that, in the 2-D case, the RSM model requires 1.7 times more computational time than the SST model, when implemented on a 2.6 GHz AMD Athlon 64 X2 Dual Core Processor 5000+ with 4 GB of RAM. To keep the simulation time as short as possible, the SST model was used for the main studies.

Three meshes (approximately 0.3, 0.5 and 1.1 million nodes) were used to deter-

mine grid independence for the simulation of the TTM turbine. Completion of the grid independence study showed that several flow parameters exhibited oscillatory convergence. Because of this problem, the AES method was used to estimate the error band (GCI), as this method is applicable even when convergence is oscillatory. The estimate of the error band (GCI) was used to determine the uncertainty of the simulations. Due to high sensitivities to the total pressure and total temperature, the relative uncertainty in the change in entropy was found to be approximately 6 times the relative uncertainty in pressure. Comparison of the relative uncertainty of various flow parameters showed that the grid with approximately 0.5 million nodes (0.17 million nodes per blade passage) was adequate to resolve the turbine flow features.

Simulation validation showed that the simulations captured the major flow features such as the wakes, the stator vane trailing edge shock and the tip leakage vortices. However, the identification of the passage vortex in the simulation results was difficult. Analysis of the simulations showed that they predicted the development of an incoming boundary layer and a spanwise pressure gradient, which are responsible for the generation of the passage vortex. Despite this, no clear passage vortex was visible downstream of the rotor. However, possible evidence of the horseshoe vortex and the passage vortex was detected in plots of the entropy loss coefficient downstream of the rotor blade trailing edges.

The numerical simulations were able to capture velocity fluctuations occurring at the major frequencies but failed to capture velocity fluctuations at frequencies higher than the rotor blade passing frequency. This may be a result of the lower levels of kinetic energy predicted by the simulations than those observed in the experimental results. While the

numerical simulations under-predicted the levels of kinetic energy, the simulation results were able to show the tip leakage vortex whereas the experimental results did not. A comparison of the predicted and measured turbine power outputs showed that the prediction was approximately 1.7% lower than the measurement. Considering the various approximations and uncertainties in both experiments and simulations, this level of agreement for the turbine power output was deemed to be acceptable.

Comparison of the computational requirements of the three simulations showed that the scaled case (TTM3:2) required approximately 30% of the simulation time per time step required by the other two cases. As a result, scaling that only reduced the computational domain volume by approximately 33%, reduced the computational requirements by twice that proportion. The present results demonstrate that blade scaling affects significantly the magnitude of unsteady pressure fluctuations, which influence the level of vibratory stresses on the blades. Scaling was found to lead to a significant underestimation of the pressure fluctuations on the rotor blades and stator vanes, possibly leading to under-prediction of blade flutter and vibration amplitudes. Moreover, scaling has been found to affect the size of the tip leakage vortices, and consequently turbine efficiency. In addition, scaling led to an underestimation of the turbine power output.

10.2 Recommendations for Future Work

Due to constraints in time and computational resources, only three cases were simulated. It would have been desirable to simulate additional cases, including cases with $\Lambda_{sc} = 1$ and $\Lambda_{sc} = 2$ and some other intermediate values. This would have provided more

data for comparison with existing results and cases with $\Lambda_{sc} = 1$ and $\Lambda_{sc} = 2$ would have allowed comparisons with the extreme cases of a small pitch size and a large rotor blade leading edge radius. The relationship between computational time required and simulation domain size would have been determined more clearly. This would help in determining the optimum amount of scaling. Therefore, it is recommended that such cases be examined if opportunity arises in the future.

The objective of this work was to provide guidance to designers concerning the effect of scaling on the prediction of vibratory stresses on the blades. Using the results from this study an analysis of the blade vibrational mode can be completed to determine the effect of scaling on the blade vibrational mode shapes. The location of large amplitude pressure fluctuations may affect the vibrational torque on the blade about the blade root and should be analyzed as well.

References

- Adamczyk, J. (1985). Model equation of simulating flows in multistage turbomachinery. *J. of Turbomach.* (85-GT-226).
- Adamczyk, J., M. Celestina, T. Beach, and M. Barnett (1990). Simulation of three-dimensional viscous flow within a multistage turbomachine. *J. of Turbomach.* 112(3), 370–376.
- AIAA (1998). Guide for the verification and validation of computational fluid dynamics simulations. Technical Report G-077-1998, American Institute of Aeronautics and Astronautics.
- Ansys (2006). *FLUENT 6.3 User's Guide*. Ansys Inc.
- Ansys (2008). *GAMBIT 2.3 Documentation*. Ansys Inc.
- Arnone, A. and R. Pacciani (1996). Rotor-stator interaction analysis using the navier-stokes equations and a multigrid method. *J. of Turbomach.* 118, 679–689.
- Bardina, J., P. Huang, and T. Coakley (1997). Turbulence modeling validation. In *28th Fluid Dynamics Conference*, Snowmass Village, CO.
- Barth, T. and D. Jespersen (1989). The design and application of upwind schemes on

- unstructured meshes. Technical Report AIAA-89-0366, AIAA 27th Aerospace Science Meeting, Reno, Nevada.
- Blazek, J. (2005). *Computational Fluid Dynamics: Principles and Applications* (2nd ed.). Elsevier Boston.
- Bradshaw, P. (1987). Turbulent secondary flows. *Ann. Rev. Fluid Mech.* 19, 53–74.
- Bradshaw, P. (1997). Understanding and prediction of turbulent flow - 1996. *Int. J. Heat Fluid Flow* 18, 45–54.
- Celik, I. and J. Li (2005). Assesment of numerical uncertainty for the calculations of turbulent flow over a backward-facing step. *Int. J. Numer. Meth. Fluids* 49, 1015–1031.
- Celik, I., J. Li, G. Hu, and C. Shaffer (2005). Limitations of richardson extrapolation and some possible remedies. *J. Fluids Eng.* 127, 795–805.
- Chang, D. and S. Tavoularis (2005). Unsteady numerical simulations of turbulence and coherent structures in axial flow near a narrow gap. *J. Fluids Eng.* 127, 458–466.
- Chang, D. and S. Tavoularis (2007). Numerical simulation of turbulent flow in a 37-rod bundle. *Nuclear Engineering and Design* 237, 575–590.
- Chernobrovkin, A. and B. Lakshminarayana (1999). Numerical simulation of complex turbomachinery flows. Technical Report NASA/CR-1999-209303, NASA Glenn Research Center, 21000 Brookpark Road Cleveland, OH 44135.
- Clark, J., G. Stetson, S. Magge, R. Ni, C. Haldeman Jr., and M. Dunn (2000). The effect of airfoil scaling on the predicted unsteady loading onthe blade of a 1 and 1/2 stage

- transonic turbine and a comparison with experimental results. In *Proceedings of the IGTI: ASME Turbo Expo 2000*, Number 2000-GT-0446, Munich, Germany. ASME.
- Constantinescu, G., R. Ettema, M. Koken, and M. Muste (2004). An eddy-resolving technique to predict the unsteady horseshoe vortex and wake of a circular pier. In *Second International Conference on Scour and Erosion, November 14-17, Singapore*, Iowa City, IA 52242-1585, USA.
- Dawes, N. (1992a). Towards improved throughflow capability: The use of 3d viscous flow solvers in a multistage environment. *J. of Turbomach.* 114, 8.
- Dawes, W. (1992b). Toward improved throughflow capability: The use of three-dimensional viscous flow solvers in a multistage environment. *J. of Turbomach.* 114, 8–17.
- Denton, J. (1993). Loss mechanisms in turbomachines. *J. of Turbomach.* 115, 621–656.
- Erdos, J., E. Alznerf, and W. McNally (1977). Numerical solution of periodic transonic flow through a fan stage. *AIAA Journal* 15(11), 1559–1568.
- Faragher, J. (2004). Probabilistic methods for the quantification of uncertainty and error in computational fluid dynamics simulations. Technical Report DSTO-TR-1633, Australian Government Department of Defence.
- Giles, M. (1988). Calculation of unsteady Wake/Rotor interaction. *J. Propul. Power* 4(4), 356–362.
- Giles, M. (1990). Stator/Rotor interaction in a transonic turbine. *J. Propul. Power* 6(5), 621–627.

- Goldstein, R. and R. Spores (1988). Turbulent transport on the endwall in the region between adjacent turbine blades. *J. Heat Transfer* 110, 862–869.
- Göttlich, E., F. Neumayer, J. Woisetschläger, W. Sanz, and F. Heitmeir (2004). Investigation of stator-rotor interaction in a transonic turbine stage using laser doppler velocimetry and pneumatic probes. *J. of Turbomach.* 126, 297–305.
- Hall, K., J. Thomas, and W. Clark (2002). Computation of unsteady nonlinear flows in cascades using a harmonic balance technique. *AIAA Journal* 40(5), 879–886.
- He, L. (1990). An euler solution for unsteady flows around oscillating blades. *J. of Turbomach.* 112, 714–722.
- He, L. and J. Denton (1994). Three-dimensional time-marching inviscid and viscous solutions for unsteady flows around vibrating blades. *J. of Turbomach.* 116, 469–476.
- Hinze, J. (1975). *Turbulence*. McGraw-Hill Book Company, New York.
- Hunt, J., C. Wray, and P. Moin (1988). Eddies, streams, and convergence zones in turbulent flows. Technical Report CTR-S88, Center for Turbulence Research, Stanford University.
- Lakshminarayana, B. (1996). *Fluid Dynamics and Heat Transfer of Turbomachinery*. John Wiley and Sons New York.
- Langston, L. (1980). Crossflows in a turbine cascade passage. *J. Eng. Gas Turbines Power* 102, 866–874.
- Launder, B. and D. Spalding (1974). The numerical computation of turbulent flows. *Comput. Meth. Appl. Mech. Eng.*, 269–289.

- Mårtensson, H., B. Laumert, and T. Fransson (2003). Aeromechanical aspects on unsteady flow in turbines. In *33rd AIAA Fluid Dynamics Conference and Exhibit*, Number AIAA-2003-3997, Orlando, Florida. AIAA.
- Mehta, U. (1996). Guide to credible computer simulations of fluid flows. *AIAA Journal* 12(5), 940–948.
- Meneveau, C. and J. Katz (2002). A deterministic stress model for rotor-stator interactions in simulations of average-passage flow. *J. Fluids Eng.* 124, 550–554.
- Menter, F. (1991). Influence of freestream values on $k-\omega$ turbulence model predictions. *AIAA Journal* 30(6), 1657–1659.
- Menter, F. (1992). Performance of popular turbulence models for attached and separated adverse pressure gradient flows. *AIAA Journal* 30(8), 2066–2072.
- Menter, F. (1994). Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA Journal* 32(8), 1598–1605.
- Menter, F. (2002). CFD best practice guidelines for CFD code validation for reactor-safety applications. Technical Report FIKS-CT-2001-00154, Evaluation of Computational Fluid Dynamic Methods for Reactor Safety Analysis Project (ECORA Project).
- Nadarajah, S. and A. Jameson (2006). Optimum shape design for unsteady three-dimensional viscous flows using a non-linear frequency domain method. In *24th Applied Aerodynamics Conference*, Number AIAA-2006-3455, San Francisco, California. AIAA.
- Northall, J. (2006). The influence of variable gas properties on turbomachinery compu-

- tational fluid dynamics. *J. of Turbomach.* 128, 632–638.
- Paulon, J., Z. Zhang, P. Jia, and J. Meng (1992). Influence of unsteady effects on the measurements in a transonic axial compressor. *J. of Turbomach.* 114, 510–516.
- Payne, S. (2001). *Unsteady Loss in a High Pressure Turbine Stage*. Ph. D. thesis, University of Oxford.
- Rai, M. (1987). Navier-stokes simulations of Rotor/Stator interaction using patched and overlaid grids. *J. Propul. Power* 3(5), 387–396.
- Rai, M. and N. Madavan (1990). Multi-airfoil navier-stokes simulations of turbine rotor-stator interaction. *J. of Turbomach.* 112, 377–384.
- Roache, P. (1998). *Verification and Validation in Computational Science and Engineering*. Hermosa Publishers, Albuquerque, New Mexico, USA.
- Roe, P. (1986). Characteristic based schemes for the euler equations. *Ann. Rev. Fluid Mech.* 18, 337–365.
- Roulund, A., B. Mutlu Sumer, J. Fredsoe, and J. Michelsen (2005). Numerical and experimental investigation of flow and scour around a circular pile. *J. Fluid Mech.* 534, 351–401.
- Schobeiri, M. (2005). *Turbomachinery Flow Physics and Dynamic Performance*. Springer-Verlag Berlin.
- Sharma, O. and T. Butler (1987). Predictions of endwall losses and secondary flows in axial flow turbine cascades. *J. of Turbomach.* 109, 229–236.
- Shih, T.-H., W. Liou, A. Shabbir, Z. Yang, and J. Zhu (1994). A new $k-\epsilon$ eddy viscos-

- ity model for high reynolds number turbulent flows-model developement and validation. Technical Report NASA-TM-106721, Institue for Computational Mechanics in Propulsion and Center for Modeling of Turbulence and Transition Lewis Research Center.
- Sleiman, M. (1999). *Simulation of 3-D Viscous Compressible Flow in Multistage Turbomachinery by Finite Element Methods*. Ph. D. thesis, Concordia University.
- Spalart, S. and S. Allmaras (1992). A one-equation turbulence model for aerodynamic flows. *AIAA Journal*, 5–21.
- Speziale, C. (1996). *Modeling of Turbulent Transport Equations, Simulation and Modeling of Turbulent Flow*. Oxford University Press, New York.
- Stieger, R. (2002). *The Effects of Wakes on Separating Boundary Layers in Low Pressure Turbines*. Ph. D. thesis, Cambridge University.
- Stieger, R., S. Hollis, and H. Hodson (2003). Unsteady surface pressures due to wakr induced transition in a laminar separation bubble on a LP turbine cascade. In *Proceedings of ASME Turbo Expo 2003*, Number 2003-GT-38303, Atlanta, Georgia, USA. ASME.
- Takeishi, K., M. Matsuura, S. Aoki, and T. Sato (1990). An experimental study of heat transfer and film cooling on low aspect ratio turbine nozzles. *J. of Turbomach.* 112, 488–496.
- Tavoularis, S. (2005). *Measurement in Fluid Mechanics*. 40 West 20th Street, New York, NY 10011-4211, USA: Cambridge University Press.

- Van der Weide, E., A. Gopinath, and A. Jameson (2005). Turbomachinery applications with the time spectral method. In *35th AIAA Fluid Dynamics Conference and Exhibit*, Number AIAA-2005-4905, Toronto, Ontario. AIAA.
- Van Wylen, G. and R. Sonntag (1978). *Fundamentals of Classical Thermodynamics* (2nd ed.). John Wiley and Sons New York.
- Vandoormaal, J. and G. Raithby (1984). Enhancements of the SIMPLE method for predicting incompressible fluid flows. *Numer. Heat Transfer* 7, 147–163.
- Wilcox, D. (2000). *Turbulence Modeling for CFD* (2nd ed.). DCW Industries.
- Xiao, X., A. McCarter, and B. Lakshminarayana (2001). Tip clearance effects in a turbine rotor: Part 1-pressure field and loss. *J. of Turbomach.* 123, 296–304.
- Xun, Z., H. Wanjin, and L. Zhiqiang (2005). Experimental investigation of energy loss in straight and bowed cascades with aft-loaded profiles. *J. of Aerospace Power* 19, 118–125.
- Yakhot, V. and S. Orszag (1986). Renormalization group analysis of turbulence. 1. basic theory. *J. Sci. Comput.* 1(1), 3–51.
- Yao, J., R. Davis, J. Alonso, and A. Jameson (2002). Massively parallel simulation of the unsteady flow in an axial turbine stage. *J. Propul. Power* 18, 465–471.
- Zwart, P. (1995). Grid turbulence in compressible flow. Master’s thesis, University of Ottawa, Ottawa, Ontario, Canada.

Appendix A. Simulations with Variable Tip Gap

During the early phase of simulations, scaling was only applied along the axial coordinate a and the tangential coordinate θ . The resulting tip gap geometry is shown in Figure .1. The tip gap size at the leading edge remains constant in all cases after scaling. However, at the trailing edge (and the entire length of the blade) the gap size was affected by scaling. In the downscaled case (TTM5:3), the tip gap at the trailing edge of the rotor blade is reduced with respect to the unscaled case, whereas, in the upscaled case (TTM4:3), the tip gap size at the trailing edge of the rotor blade is increased.

Turbine performance results for the various cases with the variable tip gap size and the uniform tip gap size are shown in Table .1. The additional cases with a variable tip gap are labeled with the subscript *vtg* (variable tip gap). Isocontours of the entropy loss coefficient are shown in plane C1 in Figure .2. Comparison of the time-averaged power output shows that the power output increases as the tip gap size decreases. The smaller the tip gap size also results in increased pressure ratio $\bar{\bar{P}}_{01}/\bar{\bar{P}}_{03}$ and increased temperature change $(\bar{\bar{T}}_{01} - \bar{\bar{T}}_{03})/\bar{\bar{T}}_{01}$. When comparing the isentropic efficiency and the loss on plane

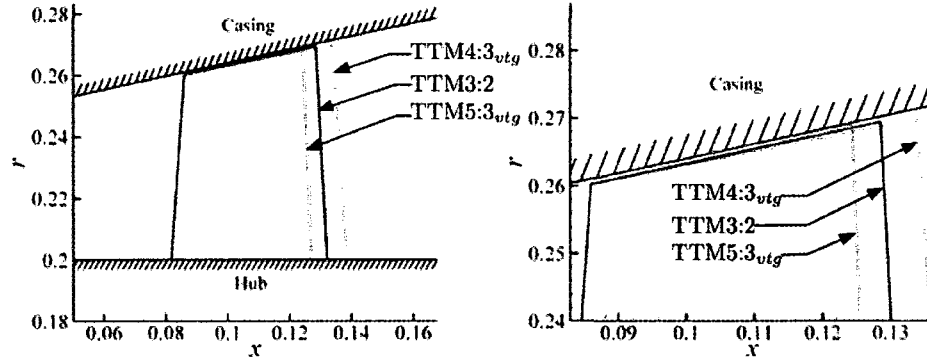


Figure .1: Side profile of a TTM rotor blade and scaled cases with variable tip gap

Table .1: Time averaged turbine performance parameters for scaled and unscaled cases

Case	$\frac{\bar{W}}{\bar{P}_{01} D_{hub}^2 \sqrt{R \bar{T}_{01}}}$	η_{tt}	$\frac{\bar{P}_{01}}{\bar{P}_{03}}$	$\frac{(\bar{T}_{01} - \bar{T}_{03})}{\bar{T}_{01}}$
TTM3:2	0.098	0.74	3.5	0.22
TTM5:3	0.105 (+7.0%)	0.88 (+19%)	3.3 (-5.5%)	0.25 (+15%)
TTM5:3 _{vtg}	0.110 (+12%)	0.90 (+22%)	3.4 (-2.9%)	0.27 (+23%)
TTM4:3	0.106 (+8.5%)	0.88 (+19%)	3.4 (-3.1%)	0.26 (+16%)
TTM4:3 _{vtg}	0.103 (+5.1%)	0.85 (+15%)	3.2 (-8.6%)	0.24 (+9.1%)

C1, there is a distinct rise in the isentropic efficiency when the tip gap size is reduced (losses are reduced). This is to be expected, because the two parameters are derived using essentially the same variables, namely the total pressure $\bar{P}_{03}$ and the total temperature $\bar{T}_{03}$.

The results show that a smaller tip gap generally results in better performance, which is to be expected. However, a tip clearance gap is required to ensure that the rotor blade tips do not contact the turbomachine casing. Contact could be a result of machining uncertainty, blade expansion due to high fluid temperatures or other deformation of the blade. The significant effect of the tip gap size on turbine performance makes isolating the effect of scaling impossible. Therefore, in the main simulations, the scaled rotor tip

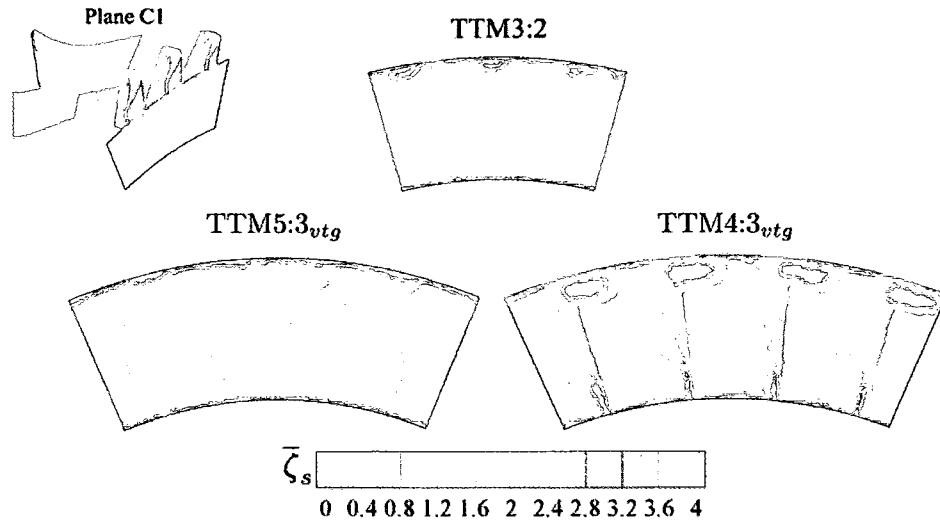


Figure .2: Predicted isocontours of entropy loss coefficient on plane C1

profile was adjusted such that the tip gap size remained uniform and identical to that in the unscaled case.

Appendix B. Post-Processing of the Results

Introduction

The post-processing procedure involved the following three steps.

1. Scaling the rotor blades.
2. Converting FLUENT files into Tecplot files.
3. Calculating relative time-averages and standard deviations.

The Tecplot files can also be imported into Tecplot to obtain instantaneous results. Scaling of the rotor blade is accomplished with the Matlab function *scale.m*. Converting the FLUENT file into Tecplot files is done with the Matlab function *fluentmac.m*. The Matlab function *fluentavg.m* calculates the relative time-averages and standard deviations. Sub-programs called by *fluentavg.m* include the function to read the original Tecplot files (*readtecplot.m*) and write the new Tecplot files (*writetecplot.m*) and a function to interpolate for a value over a 3-D surface (*surfinterp3d.m*).

Rotor Blade Scaling Code

Function scale.m

```
function scale(Lambda,Lambdascale,numprofiles)
% matlab function scale(Lambda,Lambdascale) takes rotor profile data
% in the form of:
%
%radius      x1  y1  z1  x2  y2  z2
%#           #   #   #   #   #   #
%#           #   #   #   #   #   #
%etc.
%note: headers must be deleted from the input file before using this
%function
%
%where x1, y1, z1 are the Cartesian coordinates that define one
%surface of the rotor and x2, y2, z2 are the Cartesian coordinates that
%define the other surface of the rotor. The input data is groups such
%that 21 profiles are given (each with a constant radius value). Profiles
%with the smallest radial value are given first. Output file are separated
%by profiles and suction or pressure side. Therefore a total of 42 data
%files are written. 21 profiles X 2 surfaces per profile.
%
%The inputs (Lambda,Lambdascale,numprofiles) required are the ratio between
%the number of rotor blades and stator vanes in the actual turbine (Lambda)
%and the ratio between the number of rotor blades and stator vanes required
%in the model (Lambdascale). numprofiles is an integer indicating how many
%profiles define the blade surface. Therefore, if 39 rotor blades and 13
%stator vanes are required in the model then n=3.
q=Lambda/Lambdascale;
rotor=load('rotor.dat');
for i=1:numprofiles
    %Convert from inches to mm
    Rad(:,i)=rotor(52*(i-1)+1:52*i,1)*25.4;
    X1(:,i)=rotor(52*(i-1)+1:52*i,2)*25.4;
    Y1(:,i)=rotor(52*(i-1)+1:52*i,3)*25.4;
    Z1(:,i)=rotor(52*(i-1)+1:52*i,4)*25.4;
    X2(:,i)=rotor(52*(i-1)+1:52*i,5)*25.4;
    Y2(:,i)=rotor(52*(i-1)+1:52*i,6)*25.4;
    Z2(:,i)=rotor(52*(i-1)+1:52*i,7)*25.4;
    %join suction and pressure surfaces at leading edge and trailing edge
    X1(52,i)=X2(50,i);
    X2(52,i)=X1(50,i);
    Y1(52,i)=Y2(50,i);
```

```

Y2(52,i)=Y1(50,i);
Z1(52,i)=Z2(50,i);
Z2(52,i)=Z1(50,i);
%Scale rotor blade along spanwise planes
X1s(:,i)=X1(:,i).*q;
Y1s(:,i)=Y1(:,i).*q;
Z1s(:,i)=Z1(:,i).*q;
X2s(:,i)=X2(:,i).*q;
Y2s(:,i)=Y2(:,i).*q;
Z2s(:,i)=Z2(:,i).*q;

%shift scaled rotor blade so that leading edge is coincident with
%unscaled case
Xshift=X1(1,i)-X1s(1,i);
Yshift=Y1(1,i)-Y1s(1,i);
X1s(:,i)=X1s(:,i)+Xshift;
Y1s(:,i)=Y1s(:,i)+Yshift;
X2s(:,i)=X2s(:,i)+Xshift;
Y2s(:,i)=Y2s(:,i)+Yshift;

%output and compare scaled and unscaled case
plot3(X1(:,i),Y1(:,i),Z1(:,i),'k',X2(:,i),Y2(:,i),Z2(:,i),'k')
plot3(X1s(:,i),Y1s(:,i),Z1(:,i),'--r',X2s(:,i),Y2s(:,i),Z2(:,i),'--r')
hold on
end

for i=1:numprofiles
x(52*(i-1)+1:52*i,1)=X1s(:,i);
y(52*(i-1)+1:52*i,1)=Y1s(:,i);
z(52*(i-1)+1:52*i,1)=Z1(:,i);
x2(52*(i-1)+1:52*i,1)=X2s(:,i);
y2(52*(i-1)+1:52*i,1)=Y2s(:,i);
z2(52*(i-1)+1:52*i,1)=Z2(:,i);
r(52*(i-1)+1:52*i,1)=Rad(:,i);

file1=[X1s(:,i) Y1s(:,i) Z1(:,i)];
file2=[X2s(:,i) Y2s(:,i) Z2(:,i)];
rotorscaleside1=[Rad(:,i) file1];
rotorscaleside2=[Rad(:,i) file2];
N=num2str(i);
fname1=['PW210s-scaled-rotor-profile' N '-side1.dat'];
fname2=['PW210s-scaled-rotor-profile' N '-side2.dat'];
save(fname1,'rotorscaleside1','-ascii');
save(fname2,'rotorscaleside2','-ascii');

```

end

end

Exporting FLUENT .cas and .dat Files to Tecplot .dat Format

The function *fluentmac.m* creates a text file that can be read into FLUENT as a journal file. FLUENT runs the predetermined series of commands outlined in the text file which in this case includes the generation of new iso-surfaces (one new surface is shown as an example), exporting the data to the required Tecplot data file format for use in *fluentavg.m*, and saving the new FLUENT case and data files in zip format (.gz). The function *fluentmac.m* can be customized for any FLUENT problem by changing any line with an arrow to the right of it. Lines without arrows to the right must remain unchanged for the program to generate a proper journal file. The program requires the name of the output filename *fname* as input and the FLUENT filenames must be of the format TTM-19580, where a number indicates the timestep *i* and can be preceded and followed by any text as long as the text remains constant throughout the series. The difference between the timestep numbers between consecutive filenames must be constant. That is TTM-19580, TTM-19600, TTM-19620 is allowed but TTM-19580, TTM-19600, TTM-19630 is not. The line for $i = 19580 : 20 : 23180$ indicates the timestep number (19580) of the first file in the series, the change in timestep number (20) between consecutive files, and the timestep number (23180) of the final file. In addition, the program also requires that the full path of the FLUENT filename be given for reading (in this case F:\TTM-*i*) and writing (in this case F:\TTM-*i*.cas.gz).

With respect for generation of iso-surfaces, the FLUENT manual contains all the required information for generating a iso-surface using the command line. The surfaces or zones that are exported can be increased or decreased by addition or subtraction of additional 'surface/zone to export' lines. The names of the zones in the Matlab program must exactly match the name of the zone in the FLUENT file for the journal file to work. In a similar manner to the surfaces/zones, variables to export can be increased or decreased by addition or subtraction of 'variable to export' lines. Again the variable names in the Matlab program must exactly match the name of the variable in the FLUENT for the journal file to work.

Function fluentmac.m

```
function fluentmac(fname)
fid=fopen(fname, 'wt' );
for i=19580:20:23180 %<-----filename number range
    N=num2str(i);
    S={'/file/rcd';...
        ['F:\TTM-' N];... %<-----read fluent filename
        '/surface/iso-surface';
        'x-coordinate';... %<-----iso-surface of?
        'axial-rotor-10';... %<-----new surface name
        ' ';... %<-----from surface? enter space for none
        'fluid-32';... %<-----from zone?
        ' ';...
        '0.085029';... %<-----iso-value
        ' ';...
        '/file/export/tecplot';...
        ['F:\TTM-' N]; %tecplot filename
        'axial-rotor-10';... %<-----surface/zone to export
        'hub1';... %<-----surface/zone to export
        'hub2';... %<-----surface/zone to export
        'hub3';... %<-----surface/zone to export
        'planeb1';... %<-----surface/zone to export
        'planec1';... %<-----surface/zone to export
        'rotor-blade';... %<-----surface/zone to export
        'rotor-blade:002';... %<-----surface/zone to export
```

```

'rotor-blade:003';... %<-----surface/zone to export
'rotor-blade:004';... %<-----surface/zone to export
'rotor-span10';... %<-----surface/zone to export
'rotor-tip';... %<-----surface/zone to export
'rotor-tip:028';... %<-----surface/zone to export
'rotor-tip:029';... %<-----surface/zone to export
'rotor-tip:030';... %<-----surface/zone to export
'stator-hub';... %<-----surface/zone to export
'stator-out';... %<-----surface/zone to export
'stator-span10';... %<-----surface/zone to export
'stator-vane';... %<-----surface/zone to export
'stator-vane:037';... %<-----surface/zone to export
'stator-vane:040';... %<-----surface/zone to export
' ';...
'n';...
'absolute-pressure';... %<-----variable to export
'axial-coordinate';...%<-----variable to export
'axial-velocity';... %<-----variable to export
'entropy';... %<-----variable to export
'mach-number';... %<-----variable to export
'radial-coordinate';... %<-----variable to export
'radial-velocity';... %<-----variable to export
'spanwise-coordinate';... %<-----variable to export
'tangential-velocity';... %<-----variable to export
'temperature';... %<-----variable to export
'total-pressure';... %<-----variable to export
'total-temperature';... %<-----variable to export
'turb-kinetic-energy';... %<-----variable to export
'veLOCITY-magnitude';... %<-----variable to export
'vorticity-mag';... %<-----variable to export
'wall-shear';... %<-----variable to export
'q';...
'n';...
'/file/wcd';...
['F:\TTM-' N '.cas.gz'];... %<-----write fluent filename
'y'};
for j=1:length(S)
line=S{j};
fprintf(fid, '%s \n', line);
end
end
fclose(fid);
end

```

Matlab Code to Calculate Relative Time Averages and Standard Deviations

The Matlab program *fluentavg.m* calculates the relative (relative to the motion of the domain of which the plane is in) time averages and relative standard deviations (RMS values) of any chosen properties and on any number of 3-D planes for a turbomachinery problem. It does this by rotationally shifting all planes such that the absolute position of all planes is in the exact same global position as the planes in the first file. 3-D surface interpolation, using Shepard's method, is then done to average (and calculate the RMS value) the data from all planes on an structured or unstructured grid. All programs referred to in this Appendix follow the instructions.

For the present study, the CFD simulation was performed in FLUENT 6.3.26 and the data were exported from FLUENT into the required Tecplot data file format using *fluentmac.m* detailed in Appendix C. Once the required Tecplot data files are generated (one Tecplot data file for each timestep) the files must all be placed in the same path (location on the computer). FLUENT should have save the Tecplot data file using the same name as the FLUENT file with “_tec” appended to the end of the file. That is, if the FLUENT case file was “TTM-19580.cas” then the Tecplot data file for the same timestep should be “TTM-19580_tec.dat”. In order for the program *fluentavg.m* to function, some limitations must be met. These include

- Input files must be of the Tecplot data file format.
- Only surfaces can be inputted into the program (no volume zones).

- A surfaces/zone cannot exist in a rotating and fixed domains at the same time, that is surfaces in stationary domains and in rotating domains must be separate.
- Axis of rotation must be the x-axis.
- Turbomachine must be rotating with constant velocity and time change.
- Difference in timestep number between consecutive files must be constant.
- The program has not been tested with turbomachine geometry that has nodes with a z-coordinate less than 0. Therefore it is recommended that the turbomachine geometry not exceed the (x, y, z) limits $(-\infty \text{ to } \infty, -\infty \text{ to } \infty, 0 \text{ to } \infty)$.
- The program has only been tested with 3-D turbomachinery domains

The function *fluentavg.m* can be customized by altering lines of code which have an arrow to the right of them. All other lines should be left unchanged. As shown below the program requires the number of surfaces/zones "numsurfs", the timestep number of the first file "ifs", the timestep number of the final file "ffs" and the difference in timestep number between consecutive files "deltafs".

The full path of the Tecplot data files for reading is required (in this case the read filename is located in between [' '] which must remain; F:\Tecplot\TTM-' N ' _tec.dat where ' N ' takes the place of the timestep number of the filename). In Matlab if the name of the filename is variable (ie. changes with timestep) then the filename must be located in between square brackets and quotes ['filename' N '.dat']. If the filename is not variable then the square brackets can be omitted 'filename19380.dat'. Other required filenames include

name for temp files, the output filename for time averaged data and the output filename for RMS data.

Tecplot Data File Format

The required input files are Tecplot data files (.dat files). The Tecplot file must be of the following form¹ for the function *readtecplot.m* to work

```
TITLE = "Sample finite-element data"
VARIABLES = X, Y, Z, "A", "B"
ZONE T="zone-1", N=5, E=4, ET=TRIANGLE, F=FEBLOCK

x1 x2 x3 x4 x5

y1 y2 y3 y4 y5

z1 z2 z3 z4 z5

A1 A2 A3 A4 A5

B1 B2 B3 B4 B5

1 2 3
1 3 4
1 4 5
1 5 2

ZONE T="zone-2", N=6, E=4, ET=QUADRILATERAL, F=FEBLOCK

x1 x2 x3 x4 x5 x6

y1 y2 y3 y4 y5 y6

z1 z2 z3 z4 z5 z6

A1 A2 A3 A4 A5 A6

B1 B2 B3 B4 B5 B6
```

¹more information can be found at <http://www.msi.umn.edu/software/tecplot/tutorial/dataformats.html>

```

1 2 3 6
1 3 4 5
1 4 5 2
1 5 2 6

```

The number of variables in the above case is 5, but is only constrained by the processor power. Similarly the number of zones shown above is 2, but is only limited by processor power. Each zone is divided into 5 blocks (equal to the number of variables). The number of nodes N and elements E in "zone-1" are 5 and 4 respectively. Similarly for "zone-2", the number of nodes and elements are 6 and 4. There is no known relation between the number of nodes and the number of elements. Each block contains the information for one variable at every node in the zone. The block for the variable A in "zone-1" would be

```

A1 A2 A3 A4 A5

```

Following the blocks is the connectivity matrix which indicates which nodes are connected to form the elements. Each row of the connectivity matrix defines one element. Therefore, the number of rows in the connectivity matrix is equal to the number of elements E in the zone. The connectivity matrix for "zone-1" is

```

1 2 3
1 3 4
1 4 5
1 5 2

```

The variable ET determines the type of element. Currently, the program *readtec-plot.m* only understand element types "TRIANGLE" and "QUADRILATERAL". If the element type is "TRIANGLE" then the number of columns in the connectivity matrix is 3. Conversely, If the element type is "QUADRILATERAL" then the number of columns in

the connectivity matrix is 4. The structure of the data file must be of block format (as laid out above), that is that "F=FEBLOCK".

Function fluentavg.m

```
function fluentavg
%Program calculate absolute and relative time averages for a turbomachine
%some requirements:
%- Input file format must be tecplot format exported from fluent
%- Only surfaces can be inputed (no volume domains)
%- axis of rotation must be the x-axis
%- turbomachine must be rotating with constant velocity and time change
%between time step must be constant
clear all clf global

%% load data

numsurfs=30;    %<-----number of surface/zones
ifs=19580;    %<-----input timestep number of first file
ffs=23180;    %<-----input timestep number of final file
deltafs=20; %<-----input difference in timestep number between
            %consecutive files
timesteps=1+(ffs-ifs)/deltafs;    %number of timesteps or data files

% load first data file of series
N=num2str(ifs);    %generate number sequence for corresponding input
                %data files
fname=['F:\Tecplot\TTM-' N '_tec.dat']; %<-----read filename
disp(['working on file ' fname])
[varBlock1,CMBlock1,paramBlock,variables]=readtecplot(fname,numsurfs);
numvar=length(variables);

disp('Calculating time-average')
%Generate initial sumRELavg matrix
for Block=1:numsurfs
    vars1=varBlock1{Block};
    temp=zeros(1,length(vars1));
    sumRELavg=cell(size(temp));
    clear temp
    for v=4:numvar
        sumRELavg{v}=vars1{v};
    end
end
```

```

        sumRELblock{Block}=sumRELavg;
        clear vars1 sumRELavg sumRMS
    end

%% Sum flow parameters from current time step with total from previous
%%time steps to find the average
for i=ifs:deltafs:deltafs:ffs
    N=num2str(i);    %generate number sequence for corresponding input
                    %data files
    fname=['F:\Tecplot\TTM-' N '_tec.dat']; %<-----read filename
    disp(['working on file ' fname])
    [varBlock2,CMBBlock2,paramBlock2]=readtecplot(fname,numsurfs);
    %% Check which blocks/zones are rotating and by how much
    for Block=1:numsurfs
        vars1=varBlock1{Block};
        vars2=varBlock2{Block};
        y1=vars1{:,2};
        z1=vars1{:,3};
        y2=vars2{:,2};
        z2=vars2{:,3};
        theta1=mean(atan(y1./z1));
        theta2=mean(atan(y2./z2));
        rotmat=theta1-theta2;
        rotBlock{Block}=rotmat;
        clear y1 z1 y2 z2 vars1 vars2 theta1 theta2 rotmat
    end

    for Block=1:numsurfs
        disp(['working on zone ' num2str(Block) '/' num2str(numsurfs)])
        sumRELavg=sumRELblock{Block};
        vars1=varBlock1{Block};
        vars2=varBlock2{Block};
        x1=vars1{:,1};
        y1=vars1{:,2};
        z1=vars1{:,3};
        x2=vars2{:,1};
        y2=vars2{:,2};
        z2=vars2{:,3};
        deltatheta=rotBlock{Block};
        if deltatheta==0
            for v=4:numvar
                flowprop(:,v-3)=vars2{:,v};
            end
            varsinterp=surfinterp3d(x2,y2,z2,flowprop,x1,y1,z1);
        end
    end
end

```

```

        for v=4:numvar
            sumRELavg{v}=sumRELavg{v}+varsinterp(:,v-3);
        end

    else
        %Calculate relative average
        r(:,1)=sqrt(z2.^2+y2.^2);
        theta(:,1)=atan(y2./z2);
        zshift(:,1)=sqrt((r(:,1).^2)/((tan(theta(:,1))+...
        deltatheta).^2+1));
        for k=1:length(theta)
            if (theta(k,1)+deltatheta)<0
                yshift(k,1)=-sqrt(r(k,1).^2-zshift(k,1).^2);
            else
                yshift(k,1)=sqrt(r(k,1).^2-zshift(k,1).^2);
            end
        end
        for v=4:numvar
            flowprop(:,v-3)=vars2(:,v);
        end
        varsinterp=surfininterp3d(x2,yshift,zshift,flowprop,x1,y1,z1);
        for v=4:numvar
            sumRELavg{v}=sumRELavg{v}+varsinterp(:,v-3);
        end

        Interp{Block}=varsinterp;
        sumRELblock{Block}=sumRELavg;
        clear r theta zshift yshift varsshift varsinterp flowprop;...
        x1 y1 z1 x2 y2 z2 sumRELavg sumRMS vars1 vars2 deltatheta
    end
end
fname=['F:\Tecplot\TTM-' N '-temp']; %<-----temp file filename
save(fname, 'Interp');
end

% calculate avg
for Block=1:numsurfs
    sumRELavg=sumRELblock{Block};
    for v=4:numvar
        RELavg{v}=sumRELavg{v}./timesteps;
    end
    vars1=varBlock1{Block};
    x1=vars1(:,1);
    y1=vars1(:,2);

```

```

    z1=vars1{:,3};
    RELavg{:,1}=x1;
    RELavg{:,2}=y1;
    RELavg{:,3}=z1;

    RELavgBlock{Block}=RELavg;
    clear x1 y1 z1 sumRELavg sumRMS RELavg RMS
end

disp('writing tecplot file for time average')
fout='F:\Tecplot\TTM-TimeAVG_tec.dat'; %<-----output filename for
                                     %time-averaged data
writetecplot(fout,numsurfs,RELavgBlock,CMBlock1,paramBlock,variables);

disp('Calculating RMS')
%Generate initial sumRMS matrices
for Block=1:numsurfs
    vars1=varBlock1{Block};
    temp=zeros(1,length(vars1));
    RELavg=RELavgBlock{Block};
    sumRMS=cell(size(temp));
    clear temp
    for v=4:numvar
        sumRMS{v}=(vars1{v}-RELavg{v}).^2;
    end

    sumRMSblock{Block}=sumRMS;
    clear vars1 RELavg sumRMS
end

%% Sum flow parameters from current time step with total from previous time
%steps to find the RMS
for i=ifs:deltafs:deltafs:ffs
    N=num2str(i); %generate number sequence for corresponding input
                  %data files
    fname=['F:\Tecplot\TTM-' N '-temp']; %<-----temp file filename
    disp(['working on file ' fname])
    load(fname, 'Interp');

    for Block=1:numsurfs
        disp(['working on zone ' num2str(Block) '/' num2str(numsurfs)])
        varsinterp=Interp{Block};
        RELavg=RELavgBlock{Block};
        sumRMS=sumRMSblock{Block};
    end
end

```

```

        for v=4:numvar
            sumRMS{v}=sumRMS{v}+(varsinterp(:,v-3)-RELavg{v}).^2;
        end

        sumRMSblock{Block}=sumRMS;
        clear r theta zshift yshift varsshift varsinterp flowprop;...
        x1 y1 z1 x2 y2 z2 RELavg sumRMS vars1 vars2 deltatheta
    end
    delete([fname '.mat'])
end

% calculate RMS
for Block=1:numsurfs
    sumRMS=sumRMSblock{Block};
    for v=4:numvar
        RMS{v}=sqrt(sumRMS{v}./timesteps);
    end
    vars1=varBlock1{Block};
    x1=vars1{:,1};
    y1=vars1{:,2};
    z1=vars1{:,3};
    RMS{:,1}=x1;
    RMS{:,2}=y1;
    RMS{:,3}=z1;

    RMSBlock{Block}=RMS;
    clear x1 y1 z1 sumRELavg sumRMS RELavg RMS
end

disp('writing tecplot file for RMS')
fout='F:\Tecplot\TTM-RMS_tec.dat'; %<-----output filename for RMS data
writetecplot(fout,numsurfs,RMSBlock,CMBlock1,paramBlock,variables);

disp('finished')

end

```

Function readtecplot.m

```

function [varBlock,CMBlock,parametersBlock,variables]=...
readtecplot(file,Numsurfs)
%Inputs required:
% Numvars=numbers of variables output from fluent + 3

```

```

% (for x,y,z coordinates)
% Numsurfs=numbers of surfaces output from fluent
fid = fopen(file);
%read title
temp=textscan(fid,'%s',1,'delimiter','');
title=textscan(fid,'%s',1,'delimiter','');

%% read in variables
temp=textscan(fid,'%s',1,'delimiter','=');

varscan=textscan(fid,'%q',1,'delimiter','');
variablestemp(1,1)=varscan{1};

i=1;
while (strcmp(variablestemp(i), 'ZONE', 4)==0) && (i<1000)
    varscan=textscan(fid,'%q',1,'delimiter','');
    variablestemp(i+1,1)=varscan{1};
    i=i+1;
end
variables(:,1)=variablestemp(1:i-1,1);
zone(1,1)=varscan{1};
clear variablestemp varscan
Numvars=length(variables);

%% Read in Data
for Block=1:Numsurfs
%read block parameters
if Block > 1
    zonetemp=textscan(fid,'%q',1,'delimiter','');
    zone(1,1)=zonetemp{1};
end
temp=textscan(fid,'N=%u E=%u %s %s','delimiter','');

for j=1:2
    param(1,j)=temp(j);
end
for j=3:4
    param(1,j)=temp{j};
end
parameters=[zone param];
%parameters
parametersBlock{Block}=parameters;
%parametersBlock
N=param{1};

```

```

E=param{2};

    for i=1:Numvars
        temp=textscan(fid,'%f',N);
        var(:,i)=temp;
    end
    varBlock{Block}=var;
    if strcmp(param{3}, 'ET=TRIANGLE')==1
        connect=textscan(fid,'%u %u %u',E, 'CollectOutput', 1);
        connectivtymatrix=connect{:};
        CMBlock{Block}=connectivtymatrix;
    elseif strcmp(param{3}, 'ET=QUADRILATERAL')==1
        connect=textscan(fid,'%u %u %u %u',E, 'CollectOutput', 1);
        connectivtymatrix=connect{:};
        CMBlock{Block}=connectivtymatrix;
    end
end
fclose(fid);

end

```

Function writetecplot.m

```

function writetecplot(file,numsurfs,varBlock,CMBlock,paramBlock,variables)
fid = fopen(file, 'wt');
fprintf(fid,'%s \n','TITLE = "title"');
fprintf(fid,'%s','VARIABLES =');

fprintf(fid,' %s',variables{1});
for i=2:3
    fprintf(fid,', %s',variables{i});
end

for i=4:length(variables)
    fprintf(fid,', "%s"',variables{i});
end

fprintf(fid,'\n');
for i=1:length(paramBlock)
    param=paramBlock{i};
    vars=varBlock{i};
    cm=CMBlock{i};
    fprintf(fid,'%s", N=%u, E=%u, %s, %s\n', param{:});
    fprintf(fid,'\n');
end

```

```

for j=1:length(variables)
fprintf(fid,'%e %e %e %e %e\n', vars{:,j});
fprintf(fid,'\n');
%fprintf(fid,'\n');
end

for j=1:length(cm)
    if strcmp(param{4}, 'ET=TRIANGLE')==1
        fprintf(fid,'%u %u %u\n', cm(j,1:3));
    elseif strcmp(param{4}, 'ET=QUADRILATERAL')==1
        fprintf(fid,'%u %u %u %u\n', cm(j,1:4));
    end
end
fprintf(fid,'\n');
%fprintf(fid,'\n');
end
fclose(fid);
end

```

3-D Surface Interpolation Using Shepard's Method

Function surfinterp3d.m

```

function [Propi]=surfinterp3d(x,y,z,Prop,xi,yi,zi)
w=size(Prop);
Propi=zeros(length(xi),w(2));
for i=1:length(xi)
    if i<=length(x)
        rtest=sqrt((xi(i)-x(i)).^2+(yi(i)-y(i)).^2+(zi(i)-z(i)).^2);
    end
    if (rtest>1e-5) | i>length(x)
        %find the closest points
        r(:,1)=sqrt((xi(i)-x).^2+(yi(i)-y).^2+(zi(i)-z).^2);
        sizeprop=size(Prop);
        %r(:,2:sizeprop(2)+1)=Prop;
        rmax=max(r(:,1));

        for j=1:4
            [rmin,imin]=min(r(:,1));
            Psort(j,:)=Prop(imin,:);
            rsort(j,1)=r(imin,1);
        end
    end
    Propi(i,:)=Psort;
end

```

```

        r(imin,1)=rmax;
    end

    %interpolation by shepards method
    %calculate weighting
    R=rsort(4,1);
    h=rsort(1:4,1);
    if rsort(1,1)==0;
        Propi(i,:)=Psort(1,:);
    else
        denom=(R-h(1,1)/(R*h(1,1)))^2+(R-h(2,1)/(R*h(2,1)))^2+...
            (R-h(3,1)/(R*h(3,1)))^2+(R-h(4,1)/(R*h(4,1)))^2;
        w(1)=(R-h(1,1)/(R*h(1,1)))^2/denom;
        w(2)=(R-h(2,1)/(R*h(2,1)))^2/denom;
        w(3)=(R-h(3,1)/(R*h(3,1)))^2/denom;
        w(4)=(R-h(4,1)/(R*h(4,1)))^2/denom;
        %interpolate

        Propi(i,:)=w(1).*Psort(1,:)+w(2).*Psort(2,:)+...
            w(3).*Psort(3,:)+w(4).*Psort(4,:);

    end
    clear r rsort R h w rmin imin rmax
else
    Propi(i,:)=Prop(i,:);
end
end

end

end

```