Numerical Analysis of Partial Admission in Axial Turbines

Narmin Baagherzadeh Hushmandi

Doctoral Thesis

2010
ABSTRACT

Numerical analysis of partial admission in axial turbines is performed in this work. Geometrical details of an existing two stage turbine facility with low reaction blades is used for this purpose. For validation of the numerical results, experimental measurements of one partial admission configuration at design point was used. The partial admission turbine with single blockage had unsymmetrical shape; therefore the full annulus of the turbine had to be modeled numerically.

The numerical grid included the full annulus geometry together with the disc gaps and rotor shrouds. Importance of various parameters in accurate modeling of the unsteady flow field of partial admission turbines was assessed. Two simpler models were selected to study the effect of accurate modeling of radial distribution of flow parameters. In the first numerical model, the computational grid was two dimensional and the radial distribution of flow parameters was neglected. The second case was three-dimensional and full blades’ span height was modeled but the leakage flows at disc cavity and rotor shroud were neglected. Detailed validation of the results from various computational models with the experimental data showed that modeling of the leakage flow at disc cavities and rotor shroud of partial admission turbines has substantial importance in accuracy of numerical computations. Comparison of the results from two computational models with varying inlet extension showed that modeling of the inlet cone has considerable importance in accuracy of results but with increased computational cost.

Partial admission turbine with admission degree of $\varepsilon = 0.524$ in one blocked arc and two opposing blocked arcs were tested. Results showed that blocking the inlet annulus in one single arc produce better overall efficiency compared to the two blocked arc model. Effect of varying axial gap distance between the first stage stator and rotor rows was also tested numerically for the partial admission turbine with admission degree of $\varepsilon = 0.726$. Results showed higher efficiency for the reduced axial gap model.

Computation showed that the main flow leave the blade path down to the disc cavity and re-enter into the flow channel downstream the blockage, this flow would pass the rotor with very low efficiency. First stage rotor blades are subject to large unsteady forces due to the non-uniform inlet flow. Plotting the unsteady forces of first stage rotor blades for partial admission turbine with single blockage showed that the blades experience large changes in magnitude and direction while traveling along the circumference. Unsteady forces of first stage rotor blades were plotted in frequency domain using Fourier transform. The largest amplitudes caused by partial admission were at first and second multiples of rotational frequency due to the existence of single blockage and change in the force direction.

Results obtained from the numerical computations showed that the discs have non-uniform pressure distribution especially in the first stage of partial admission turbines. The axial force of the first rotor wheel was considerably higher when the axial gap distance was reduced between the first stage stator and rotor rows. The commercial codes used in this work are ANSYS ICEM-CFD 11.0 as mesh generator and FLUENT 6.3 as flow solver.
PREFACE

The thesis is based on four publications which are listed below and appended at the end of the thesis.

1. Baagherzadeh Hushmandi, Narmin; Fransson, Torsten; 2009
"Effects of Multi-Blocking and Axial Gap Distance on Performance of Partial Admission Turbines – A Numerical Analysis"; Submitted to ASME Journal of Turbomachinery 26 Oct, 2009; Recommended for Publication 18 Jan, 2010; Revised Paper Submitted 25 Mar 2010; Paper Number TURBO-09-1185

2. Baagherzadeh Hushmandi, Narmin; Fridh, Jens; Fransson, Torsten; 2009
"Unsteady Forces of Rotor Blades in Full and Partial Admission Turbines"; Submitted to ASME Journal of Turbomachinery 27 May, 2009; Accepted for Publication 18 Jan, 2010; Final Version Approved 16 Feb, 2010; Paper Number TURBO-09-1064

3. B. Hushmandi, Narmin; Hu, Jiasen; Fridh, Jens; Fransson, Torsten; 2008

4. B. Hushmandi, Narmin; Hu, Jiasen; Fridh, Jens; Fransson, Torsten; 2007
"Numerical Investigation of Partial Admission Phenomena at Midspan of an Axial Steam Turbine"; Published at Proceedings of 7th European Conference on Turbomachinery, Fluid Dynamics and Thermodynamics, Athens, Greece and Presented by the First Author; pp. 885-895

Contribution of the various authors is as follows:

- Paper 1: First author was main author, research idea and computational works were done by the first author. Second author acted as mentor and reviewer.
- Paper 2: First author was main author, research idea and computational works were done by the first author. Second author did the experimental work (without participation of first author) and reviewed the paper. Third author acted as mentor and reviewer.
- Paper 3: First author was main author, research idea and computational works were done by the first author. Second author acted as numerical mentor and reviewer. Third author did the experimental work (without participation of first author) and reviewed the paper. Fourth author acted as reviewer.
- Paper 4: First author was main author, research idea and computational works were done by the first author. Second author acted as numerical mentor and reviewer. Third author did the experimental work (without participation of first author) and reviewed the paper. Fourth author acted as reviewer.
# CONTENTS

CONTENTS ................................................................................................................................. 1

List of Figures ........................................................................................................................................ 3

List of Tables ......................................................................................................................................... 7

Nomenclature ........................................................................................................................................ 8

1. Introduction ....................................................................................................................................... 11
   1.1 Losses in Full Admission Turbines ............................................................................................... 12
   1.2 Additional Losses in Partial Admission Turbines ......................................................................... 13
   1.3 Dynamic Forces of Rotor Blades in Partial Admission Turbines .................................................. 15
   1.4 Summary ....................................................................................................................................... 15

2. State of the Art ................................................................................................................................... 16
   2.1 Summary ....................................................................................................................................... 21

3. Objectives ......................................................................................................................................... 22

4. Methodology ....................................................................................................................................... 23

5. Experimental Test Facility .................................................................................................................. 25
   5.1 Turbine Facility .............................................................................................................................. 26
   5.2 Pressure Measurements .................................................................................................................. 29
   5.3 Temperature Measurement ............................................................................................................ 33
   5.4 Torque Measurements .................................................................................................................... 33
   5.5 Measurement of Mass Flow ........................................................................................................... 34
   5.6 Measurement Uncertainties .......................................................................................................... 34
   5.7 Time Averaging in Experimental Measurements ......................................................................... 35
   5.8 Performance of the Turbine ........................................................................................................... 35

6. Numerical Models and Validations ...................................................................................................... 37
   6.1 Computational Grid ......................................................................................................................... 37
   6.2 Boundary Conditions ...................................................................................................................... 40
   6.3 Traverse Pressure ............................................................................................................................ 40
   6.4 Excluding the Leakage Flows ....................................................................................................... 51
   6.5 Two-Dimensional Model ............................................................................................................... 54
   6.6 Effect of mesh resolution near blade profiles ............................................................................... 56
6.7 Extension of Numerical Inlet ........................................................................................................ 57
7. Selected Results ........................................................................................................................................ 62
  7.1 Turbine Performance in Various Partial Admission Configurations ............................................... 62
  7.2 Unsteady Forces of First Stage Rotor Blades .................................................................................. 64
  7.3 Relative Flow Angles .......................................................................................................................... 70
  7.4 Axial Force of Rotor Wheels .............................................................................................................. 73
  7.5 Coefficient of Loss .............................................................................................................................. 80
8. Conclusions and Discussion ................................................................................................................ 85
  8.1 Summary and Discussion .................................................................................................................. 85
  8.2 Conclusions and Future Work .......................................................................................................... 86
9. References ........................................................................................................................................ 88
10. Acknowledgement ............................................................................................................................... 92
APPENDIX A Important Geometrical Dimensions based on Rotor Blade Chord ...................................... 93
APPENDIX B Governing Equations ......................................................................................................... 95
APPENDIX C Truncation Errors ............................................................................................................... 111
APPENDIX D Contours of Normalized Static Pressure with Different Inlet Extensions.......................... 112
LIST OF FIGURES

Fig 1-1 Schematic mechanism arranged for sequential opening of the inlet valves ...... 11
Fig 1-2 Schematic representation of end-wall boundary layer rollup into a horseshoe vortex (Gaugler; Russel; [1982]) ......................................................................................... 12
Fig 1-3 h-s diagram of throttling by (a) pressure reduction valve (b) and throttling by a control stage; (Fridh et al. [2004]) ........................................................................ 14
Fig 2-1 Efficiency of Small Impulse Turbines, (Ohlsson [1962]) .................................. 16
Fig 2-2 Loss coefficient comparison of partial admission turbines using various formulae and test points, (Yahya et al. [1968]) ................................................................. 17
Fig 2-3 Aerodynamic force on one rotor blade in 10 blade passing period in a turbine with admission degree of ε = 0.5, (He [1997]) ................................................................. 19
Fig 2-4 Normalized efficiency vs. velocity ratio, bold lines are the single stage measurements and non-bold lines are two-stage measurements (Fridh et al. [2004]) ......................................................................................................................................................................................... 20
Fig 2-5 Total temperature near blade surfaces for rotational speed of 7500 rpm with a mass flow rate of 0.7 kg/s (Herzog et al. [2005]) .................................................... 21
Fig 5-1 Test turbine and wind tunnel room at KTH, (Fridh et al., [2004]) ...................... 25
Fig 5-2 Two stage turbine with measurement cross sections indicated, (Fridh et al.; [2004]) .................................................................................................................. 27
Fig 5-3 Turbine facility seen in the direction of flow, (Fridh et al. [2004]) ....................... 28
Fig 5-4 Total pressure probe upstream the rotor blades, (Pfefferle [2004]) ................. 29
Fig 5-5 Stationary static and total pressure probes at cross section 2, (Södergård et al. [1989]) .................................................................................................................. 29
Fig 5-6 Traverse unit of test turbine at KTH, (Pfefferle [2004]) .................................... 30
Fig 5-7 Schematic drawing of turbine section 2 with total and static pressure probes and the traverse total pressure probe, Pfefferle [2004] .................................................. 31
Fig 5-8 Normalized dynamic pressure profiles for all operating points; Pfefferle [2004] 32
Fig 5-9 Position of thermocouples at section 1 and section 8, Södergård et al.; [1989] 33
Fig 5-10 Torque measuring device, the turbine is at the left hand side, (Pfefferle [2004]) .................................................................................................................. 33
Fig 5-11 Mass flow measuring orifice (Pfefferle [2004]) ................................................. 34
Fig 5-12 Enthalpy-entropy diagram for a multistage turbine, (Horlock [1966]) .......... 35
Fig 6-1 Computational Grid for the Partial Admission Turbine Simulations (Extended Inlet) ............................................................................................................. 37
Fig 6-2 Computational grid, (A) Around the blades, (B) Disc cavity between S1&R1, (C) First stage shroud, (D) Complete grid clipped with 45 degrees arc (short inlet) 39
Fig 6-3 Static pressure at cross section 3, Computational results at hub and casing compared with experimental measurement data ......................................................... 41
Fig 6-4 Computed total pressure at hub, midspan and casing of section 3 ................... 42
Fig 6-5 Numerical and experimental total pressure at midspan of first stage stator leading edge .......................................................... 42
Fig 6-6 Numerical and experimental values of static pressure at hub and casing of cross section 4 .......................................................... 43
Fig 6-7 Total pressure at cross section 4, numerical results at hub, midspan and casing are compared with the experimental midspan values ........................................... 44
Fig 6-8 Static pressure contours and velocity vectors at a cross section passing the cavity between S1 & R1 (View: Upstream to Downstream) ........................................45
Fig 6-9 Static pressure contours and velocity vectors at a cross section passing the cavity between R1 & S2 (View: Upstream to Downstream) .........................46
Fig 6-10 Numerical and experimental static pressure at hub and casing of cross section 5 ..............................................................................................................47
Fig 6-11 Numerical total pressure at hub, midspan and casing of cross section 5 ....47
Fig 6-12 Static pressure contours and velocity vectors at a cross section passing the cavity between S2 & R2 (View: Upstream to Downstream) ......................48
Fig 6-13 Static pressure contours and velocity vectors at a cross section passing the Cavity between second rotor disc & Exhaust casing (View: Upstream to Downstream) ..........................................................................................................................49
Fig 6-14 Static pressure at Cross section 6 ....................................................................50
Fig 6-15 Total pressure at Cross section 6 ....................................................................50
Fig 6-16 Computational grid of the two stage partial admission turbine without leakage flow modeling .............................................................................51
Fig 6-17 Static pressure at cross section 3 (Simple-3D model and experimental data)52
Fig 6-18 Static pressure at cross section 4 (Simple-3D model and experimental data). 52
Fig 6-19 Computed and measured torque on the rotor shaft of the two-stage partial admission turbine ..............................................................................................................53
Fig 6-20 Geometry of the two-dimensional grid, upstream to downstream view (left) ..54
Fig 6-21 Normalized static pressure at cross section 3, 2D numerical & experimental data ..................................................................................................................55
Fig 6-22 Normalized static pressure at cross section 4, two-dimensional numerical results compared to experimental data ..............................................................56
Fig 6-23 Tangential force of one rotor blade travelling along the circumference, using 2D models and difference mesh resolution near the blade profiles ..................57
Fig 6-24 Static pressure at cross section 2 ....................................................................58
Fig 6-25 Total pressure at Cross section 2 ....................................................................59
Fig 6-26 Static pressure at casing of cross section 4 ....................................................60
Fig 6-27 Static pressure at hub of cross section 4 .........................................................60
Fig 6-28 Normalized radial dynamic pressure profile at cross section 2 ....................61
Fig 7-1 Efficiency of the two stage turbine at various partial admission configurations . .63
Fig 7-2 Efficiency of first stage from the two stage turbine at various partial admission configurations ........................................................................................................64
Fig 7-3 Computed tangential forces of first stage rotor blades in full admission ..........65
Fig 7-4 Tangential forces of first stage rotor blades at full admission in frequency domain .......................................................................................................................66
Fig 7-5 Computed axial forces of first stage rotor blades at full admission .................66
Fig 7-6 Axial forces of first stage rotor blades at full admission in frequency domain ...67
Fig 7-7 Computed tangential forces of first stage rotor blades at partial admission ......67
Fig 7-8 Tangential forces of first stage rotor blades in partial admission turbine in frequency domain ........................................................................................................68
Fig 7-9 Computed axial forces of first stage rotor blades at partial admission ..........69
Fig 7-10 Axial forces of first stage rotor blades in partial admission in frequency domain .......................................................................................................................70
Fig 7-11 Relative flow vectors around a rotor blade at midspan of partial admission turbine with reduced axial gap (Admitted channel) .................................................. 70
Fig 7-12 Relative flow angles upstream and downstream of first stage rotor’s midspan, design gap ........................................................................................................ 71
Fig 7-13 Relative flow angles at upstream and downstream of first stage rotor’s midspan, Reduced gap model .......................................................................................... 72
Fig 7-14 Relative flow angles upstream and downstream of first stage rotor’s midspan, Increased gap model ............................................................................................ 73
Fig 7-15 Axial force of the first and the second stage rotor wheels ................................ 73
Fig 7-16 Static pressure contours and velocity vectors at a cross section passing the disc cavity and main flow between S1 & R1 (Design gap, $\varepsilon = 0.762$), View upstream to downstream .................................................................................. 74
Fig 7-17 Static pressure contours and velocity vectors at cross section passing the disc cavity and main flow between S1 & R1 (reduced gap, $\varepsilon = 0.762$), View upstream to downstream ................................................................. 77
Fig 7-18 Static pressure contours and velocity vectors at cross section passing the disc cavity and main flow between S1 & R1 (Single Blockage, $\varepsilon = 0.524$), View upstream to downstream .................................................................................. 77
Fig 7-19 Static pressure contours and velocity vectors at cross section passing the cavity and main flow between R1 & S2 (Single Blockage $\varepsilon = 0.524$), View upstream to downstream .................................................................................. 78
Fig 7-20 Static pressure contours and velocity vectors at cross section passing the disc cavity and main flow between S1 & R1 (Double Blockage $\varepsilon = 0.524$), View upstream to downstream .................................................................................. 80
Fig 7-21 Static pressure contours and velocity vectors at cross section passing the cavity and main flow between R1 & S2 (Double Blockage $\varepsilon = 0.524$), View upstream to downstream .................................................................................. 80
Fig 7-22 Loss coefficient ($\varepsilon = 0.524$ in two arcs), at a cross section downstream of first rotor, passing the disc cavity and main flow, View upstream to downstream .... 81
Fig 7-23 Loss coefficient ($\varepsilon = 0.524$ in one arc), at a cross section downstream of R1, passing the disc cavity and main flow, View upstream to downstream ............ 82
Fig 7-24 Loss coefficient ($\varepsilon = 0.524$ in two arcs), at a cross section downstream of R2, passing the disc Cavity and main flow, View upstream to downstream .......... 83
Fig 7-25 Loss coefficient ($\varepsilon = 0.524$ in one arc), at a cross section downstream of R2, passing the disc Cavity and main flow, View upstream to downstream .......... 84

Fig A. 1 Schematic picture of first stage rotor blade ................................................. 94

Fig B. 1 Zones in the turbulent boundary layer for a typical incompressible flow over a smooth flat plate, Greitzer; et al. [2004] after Cebeci and Bradshaw [1977] ... 106
Fig B. 2 Rotor-stator interaction, stationary guide vanes with rotating blades (FLUENT User’s Guide, 2006) ...................................................................................... 110
Fig D. 1 Contours of Static Pressure normalized with Inlet Total Pressure at the Entering End of the Blockage, extended grid (lower) and shorter grid (upper) ............... 112
Fig D. 2 Contours of Static Pressure Normalized with Inlet Total Pressure, at the Leaving End of the Blockage, for Extended Grid (Lower) and Shorter Grid (Upper) .... 113
Fig D. 3 Contours of Static Pressure Normalized with Inlet Total Pressure, at the Admission Channel, for Extended Grid (Lower) and Shorter Grid (Upper) ..... 114
LIST OF TABLES

Table 4-1 Various computational cases chosen for investigations of the current work.. 24
Table 5-1 Main data for air supply system, (Södergård et al.; [1989]) ......................... 26
Table 5-2: Main data for turbine, (Södergård et al.; [1989]) ...................................... 26
Table 5-3 Test object characteristics at midspan design point, (Fridh et al. [2004]) ..... 28
Table 5-4 Turbine operating points for radial traverse measurement of total pressure at inlet .................................................................................................................. 32
Table 5-5 Measurement uncertainties as reported by Södergård et al; [1989].......... 34
Table 6-1 Input data .................................................................................................... 40

Table A. 1 Geometrical Dimensions for 2D, Simple 3D and Full 3D models .......... 93
Table A. 2 Rotor Blade Profile Data.............................................................................. 93
# NOMENCLATURE

<table>
<thead>
<tr>
<th>Latin</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$a$</td>
<td>Speed of sound, coefficient in linearized equations</td>
</tr>
<tr>
<td>$A$</td>
<td>Area</td>
</tr>
<tr>
<td>$B$</td>
<td>Peripheral length of admission arc</td>
</tr>
<tr>
<td>$c$</td>
<td>Absolute velocity, cell</td>
</tr>
<tr>
<td>$c_i$</td>
<td>Complex Fourier coefficient</td>
</tr>
<tr>
<td>$C_{ax}$</td>
<td>Axial blade chord</td>
</tr>
<tr>
<td>$c_p$</td>
<td>Specific heat capacity at constant pressure</td>
</tr>
<tr>
<td>$c_s$</td>
<td>Theoretical isentropic velocity</td>
</tr>
<tr>
<td>$D$</td>
<td>Diameter</td>
</tr>
<tr>
<td>$D$</td>
<td>Fluid domain</td>
</tr>
<tr>
<td>$E$</td>
<td>Total energy</td>
</tr>
<tr>
<td>$E_t$</td>
<td>Total energy per unit volume</td>
</tr>
<tr>
<td>$F$</td>
<td>Force</td>
</tr>
<tr>
<td>$F_y$</td>
<td>Tangential force per unit span</td>
</tr>
<tr>
<td>$f$</td>
<td>Body force per unit mass</td>
</tr>
<tr>
<td>$f(x)$</td>
<td>Face, an arbitrary continuous function, $dF/(HdB)$, force intensity on rotor blades</td>
</tr>
<tr>
<td>$g_i$</td>
<td>Gravity vector in the $i$th direction</td>
</tr>
<tr>
<td>$h$</td>
<td>Enthalpy</td>
</tr>
<tr>
<td>$H$</td>
<td>Blade height</td>
</tr>
<tr>
<td>$I$</td>
<td>Turbulent intensity</td>
</tr>
<tr>
<td>$I_f$</td>
<td>Face mass flux</td>
</tr>
<tr>
<td>$i, j, k$</td>
<td>Unit vectors in Cartesian Coordinates</td>
</tr>
<tr>
<td>$k$</td>
<td>Thermal conductivity, turbulence kinetic energy per unit mass</td>
</tr>
<tr>
<td>$l$</td>
<td>Turbulent length scale</td>
</tr>
<tr>
<td>$L$</td>
<td>Axial length of rotor blade chord, Longitudinal length of rotor channel ([Yahya] 1968 and Ohlsson [1962])</td>
</tr>
<tr>
<td>$M$</td>
<td>Mach number, Momentum</td>
</tr>
<tr>
<td>$M_w$</td>
<td>Molecular weight</td>
</tr>
<tr>
<td>$m$</td>
<td>Mass flow</td>
</tr>
<tr>
<td>$N_{faces}$</td>
<td>Number of faces enclosing the cell</td>
</tr>
<tr>
<td>$p$</td>
<td>Unsteady state at partial admission</td>
</tr>
<tr>
<td>$P$</td>
<td>Pressure, Shaft power</td>
</tr>
<tr>
<td>$Pr$</td>
<td>Prandtl number</td>
</tr>
<tr>
<td>$Q$</td>
<td>Heat produced per unit volume (by external agencies)</td>
</tr>
<tr>
<td>$r$</td>
<td>Displacement vector</td>
</tr>
<tr>
<td>$R$</td>
<td>Universal gas constant, Cell volume</td>
</tr>
<tr>
<td>$Re$</td>
<td>Reynolds number</td>
</tr>
<tr>
<td>$S$</td>
<td>Effective temperature, Source term</td>
</tr>
<tr>
<td>$t$</td>
<td>Time, Tangential or total</td>
</tr>
<tr>
<td>$T$</td>
<td>Temperature, Time (He; 1997)</td>
</tr>
<tr>
<td>$T_0$</td>
<td>Reference temperature</td>
</tr>
<tr>
<td>$T_P$</td>
<td>Blade passing period</td>
</tr>
<tr>
<td>$U$</td>
<td>Mean blade speed at midspan</td>
</tr>
<tr>
<td>$u_1, u_2, u_3$</td>
<td>Velocity components in a generalized coordinate system</td>
</tr>
</tbody>
</table>
\( \vec{V} \) Velocity vector
\( V \) Magnitude of velocity
\( \mathcal{V} \) Volume of computational cell
\( W \) Power
\( v_t \) Characteristic velocity
\( x \) Axial
\( x_1, x_2, x_3 \) General curvilinear coordinates
\( z \) Number of admission arcs

Greek
\( \alpha \) Inverse effective Prandtl number
\( \alpha_1 \) Absolute flow angle at rotor inlet
\( \beta \) Coefficient of thermal expansion
\( \beta_1 \) Relative flow angle at rotor inlet
\( \Gamma \) Diffusion coefficient
\( \delta_{ij} \) Kronecker delta
\( \varepsilon \) Turbulence dissipation rate
\( \varepsilon \) Admission degree = (Admitted inlet arc length / Total inlet arc length)
\( \eta \) Efficiency
\( \phi \) Scalar quantity, Flow coefficient
\( \gamma \) Specific heat ratio (= \( c_p/c_v \))
\( \Delta \) Difference
\( \mu \) Dynamic viscosity coefficient
\( \mu_0 \) Reference viscosity
\( \nu \) Kinematic viscosity, Isentropic velocity ratio
\( \Pi_{ij} \) Stress tensor
\( \rho \) Density
\( \tau \) Shear stress
\( \tau_{ij} \) Viscous stress tensor
\( \omega \) Angular velocity
\( \Omega \) Characteristic swirl number
\( \zeta \) Loss coefficient

Subscripts and Superscripts
\( ax \) Axial
\( ave \) Average
\( b \) Buoyancy
\( c \) Constant
\( dyn \) Dynamic
\( eff \) Effective
\( f \) face
\( H \) Hydraulic
\( in \) Inlet
\( max \) Maximum value
\( mid \) mispan
\( n \) Normal, normalized
\( nb \) neighbor
\( out \) outlet
\( p \) Constant pressure
\( s \) static
\( s, ss \) States at the end of isentropic processes
**Settling chamber**

Turbulent, total

Tangential component

Total to static

Axial component

Total

Position between stator and rotor (Ohlsson; 1962)

Two cells sharing the face

States 1 and 2

Measurement sections 1 to 8

Fluctuation in turbulent flow

Fluctuation in turbulent flow

**State Points**

1. Entry to stage
2. Exit from stage
3. Entry to turbine
4. Exit from turbine

**Over bars**

Averaged quantity or time-averaged quantity

Mass-averaged variable

Tensor form

Vector form

**Abbreviations**

2D Two-Dimensional

3D Three-Dimensional

4a First single stage turbine

4b Second single stage turbine

4ab Two stage turbine

**Entering End** One side of the blockage where rotor blades enter the blocked channel (As illustrated in Figure below)

**Exp** Experimental

**Calc** Calculations

**CFD** Computational Fluid Dynamics

f Function

Inc Increased gap distance between the first stage stator and rotor rows

KTH Kungliga Tekniska Högskolan (Royal Institute of Technology)

**Leaving End** One side of the blockage where rotor blades exit the blocked channel (As illustrated in Figure below)

Mid Design gap distance between the first stage stator and rotor rows

Num Numerical value

Red Reduced gap distance between the first stage stator and rotor rows

RNG Renormalization group

T.E. Truncation error
1. INTRODUCTION

Dimensions of turbine blades are function of the volumetric flow rate passing through the machine. Entropy generation is usually higher for small turbines compared to the larger turbines due to the increased viscous interactions. It could be beneficial to keep the turbines dimensions large and applying partial admission. The inlet annulus in the control stage of a partial admission turbine is divided into a number of individually controlled segmental arcs, i.e. admission arcs where the mass flow can vary from zero to full value depending on the turbine load. Flow in a partial admission turbine can be blocked at one or more segmental arcs of the turbine's inlet annulus. This results in reduced inlet mass flow to regulate the power and could generate higher part-load efficiencies due to maintained high pressure at the turbine inlet. Figure (1-1) shows the first stage stationary periphery of a steam turbine which is divided into a number of arcs of admission each having separate control valve. Control valves are opened in sequence. At partial admission, some arcs are fully active, some can be throttling and some are inactive. To apply partial admission in a turbine that does not have a control stage, flow can be blocked at the leading edge of several guide vane passages by placing a physical blockage.

![Fig 1-1 Schematic mechanism arranged for sequential opening of the inlet valves (Singh [2006])](image-url)
Performance of rotating machinery is limited by various loss mechanisms within their unsteady flow field. The losses are usually categorized into several main groups based on their sources of production. There are additional forms of losses in the partial admission turbines due to the non-uniform inlet flow. The important loss mechanisms in the full and partial admission turbines are listed and described briefly below.

1.1 Losses in Full Admission Turbines

- Endwall Loss
Endwall losses occur on all surfaces of the turbomachinery. Most importantly, the endwall losses occur when the boundary layer formed on the endwall of hub and casing hit the blades and get separated. The separated boundary layer then forms the so-called horseshoe vortex in the blades passages. Inside the passages, the horseshoe vortex moves from the pressure-side of the blade towards the suction-side of the neighbor blade and forms the endwall passage vortex. Figure (1-2) shows the schematic view of endwall vortex in a stationary blade row. Denton [1993] stated that the endwall losses normally account for 1/3 of the overall losses in the full admission turbines.

![Fig 1-2 Schematic representation of end-wall boundary layer rollup into a horseshoe vortex (Gaugler; Russel; [1982])](image)
- **Profile Loss**
Profile losses occur due to the viscous interaction between the boundary layer around the blade profile and the main flow. Factors affecting the profile loss are the main flow velocity and the blade surface roughness. Profile loss also depends on the inlet and outlet flow angles and pitch to chord ratio. The profile loss is considered to be two-dimensional; therefore the loss can be predicted by cascade tests or by two-dimensional computations, Denton [1993].

- **Leakage Flow Loss**
Another form of loss is generated by viscous interaction between the flow over blade tips in the unshrouded rotor blades or the flow from the blade shrouds and the main flow. This loss is also generated by viscous interaction of the leakage flow of stator gaps and the main flow.

- **Mixing Loss**
Mixing loss occur due to the shear strain between the fluid streams within the turbomachinery. Even inside the main flow of the stationary parts of a turbine, the fluid shear stress exists and mixing losses occur. Mixing losses are relatively higher at the separated flow zones, at the wakes of blades’ trailing edge and at the places where fluids with different velocities mix together (Sectors ends of partial admission turbines).

The above mentioned losses are comparable in size and each could be responsible to one third of the total loss in the full admission turbines, Denton [1993]. There are also other forms of losses which are smaller in size and are mentioned here

- **Disc friction loss**
Disc friction loss occurs due to the friction generated by centrifuging of the gas between the rotating disk and the fluid-filled stationary casing.

- **Losses due to shock waves**
Losses generated by shock waves are important especially in transonic turbines. A shock wave contains large normal stresses which contribute to irreversibility in the flow.

### 1.2 Additional Losses in Partial Admission Turbines

Axial turbines are suitable to be used over a large range of operating points. If the required power output is so small that the full admission turbine would give blades of very small aspect ratio for the design mass flow rate, partial admission can be used. Efficiency may be better in a partial admission turbine with larger blades than the full admission turbine with smaller blades having the same mass flow rate.

In another case if the turbine is already designed, at the lower power outputs, the mass flow can be regulated by partial admission instead of reducing the pressure ratio across the full admission turbine. As shown below, this approach could give better efficiency.

Two ways of power regulation is shown schematically in an h-s diagram in Fig (1-3). Figure (1-3a) shows the flow through a stage with a pressure reduction valve and Fig 1-3b shows the flow through a control stage of a turbine with two arcs, one throttled and
one open. In Fig 1-3a, pressure of the fluid is reduced to \( p_1 \), then expanded to pressure \( p_2 \) through the stage. State 2 in Fig 1-3a is the condition before the second stage in the turbine using a pressure reduction valve. In Fig 1-3b, some of the fluid is throttled to the lower pressure \( p_1 \) then expanded to pressure \( p_2 \) and the rest of the fluid go through the open arc. Fluid condition at the exit of control stage (condition 2c in Fig 1-3b) has lower entropy compared to the fluid condition at the exit of pressure reduction valve (condition 2 in Fig 1-3a); therefore entropy generation is lower using a control stage.

When partial admission is applied to a turbine, additional forms of losses are produced in the flow field. The principle partial admission losses of axial turbines are as follows according to Horlock [1966],

- **Expansion Losses**
  Sector end losses occur at the sides of the blocked channel. When the blades enter into the blocked channel from the active sector, the fluid expands rapidly due to the low pressure formed downstream the blocked channel and loses its momentum. This type of loss depends on factors like spacing of stator vanes, mean blade diameter, admission degree, mass flow, total enthalpy drop over the turbine and ratio of inlet and outlet relative velocities of rotor.
- **End of Sector Losses**
  Mixing of the high pressure fluid from the active channel and the stagnant fluid of the blocked channel reduces the tangential momentum of the flow and creates additional losses. When the blades are entering the admitted channel at the other side of the blockage, the stationary fluid inside the blade passages is mixed by the high momentum fluid coming from the open channel; this mechanism reduces the momentum of the through-flow. Its value depends on factors like isentropic velocity ratio, mean blade diameter, mass flow rate, admission degree and as Doyle [1962] suggests to the efficiency of the nozzles.

- **Windage Losses**
  Centrifuging of the fluid in the non-admitted part of the wheel increases the overall losses. This type of loss is called windage losses and its value is related to the mean blade speed, the blade length, mean diameter of the blades, ambient density and viscosity of fluid.

### 1.3 Dynamic Forces of Rotor Blades in Partial Admission Turbines

Another aspect of applying partial admission into the rotating machinery is the dynamic forces of rotor blades. The circumferential non-uniform inlet flow, imposes cyclic loading and un-loading, especially into the control stage rotor blades. The dynamic forces on the rotor blades may result in unexpected failures and breakdowns during operation; therefore accurate prediction of the forces is of high importance in design process of a turbine intended for partial admission operation.

### 1.4 Summary

From the brief introduction, it is apparent that several complicated loss mechanisms are present in the flow field of rotating machinery specially those operating under partial admission. Historically there have been several efficiency prediction methods which have related the losses into the geometrical dimensions and flow characteristics. The applicability of these methods is limited to a specific range of operation since the real flow is not resolved. A further shortcoming of loss prediction methods is that, it is not possible to predict the dynamic forces of partial admission turbines correctly which may be considerably large compared to the full admission turbines. Validated computational methods are therefore becoming a powerful tool to predict the details of unsteady flow field in such conditions. A main part of the current work consists of evaluating the various parameters and simplification methods which can be applied without changing the nature of the flow. After satisfactory agreement is achieved, the computational model is used to show the flow mechanism and explain the sources of losses by the flow analysis in various partial admission configurations.
2. STATE OF THE ART

Several experimental investigations have been performed to enhance physical understanding of the non-uniform flow field and the losses involved in partial admission turbines. However the numerical work in this area are limited to simple models due to limitations of available numerical techniques to predict the highly turbulent flow in such conditions and the large computational power required for modeling the unsteady flow in multi-passage, multi-stage partial admission turbines.

Ohlsson [1962] addressed the partial admission of an axial flow impulse turbine by a theoretical approach. A number of simplifications were made in the theoretical derivation. Flow was assumed incompressible, frictionless, small stator and rotor pitches ($\alpha_1 = \text{const}$ and $\beta_1 = \text{const}$), impulse turbine with constant rotor channel area, no leakage between stator and rotor and no friction in the flow. Figure 2-1 shows the total to static efficiency of an impulse partial admission turbine vs. the isentropic velocity ratio, obtained from the theoretical method as was presented in Ohlsson [1962]. In Figure, B is the peripheral length of admission arc and L is the rotor channel longitudinal length. $U$ is the peripheral speed and the quantity $c_s$ is the theoretical isentropic velocity. It is seen that the maximum efficiency occurs at the velocity ratio of 0.47 for the full admission turbine and maximum efficiency of partial admission turbines occurs at lower velocity ratios.

![Fig 2-1 Efficiency of Small Impulse Turbines, (Ohlsson [1962])](image)
Ohlsson [1962] showed that the losses at the exit side of the blocked channel were much smaller than the losses at the entering side. Also, it was shown that blocking the flow in one arc had smaller losses compared to multiple blocked arcs because of avoiding the repeated entering losses in single blockage models. Losses from the leakage flows were not considered in the calculations of efficiency, while due to the strong peripheral and radial pressure gradients in partial admission turbines, the leakage flow losses could be important. Doyle [1962] suggested that the losses at the sector ends of the blockage should also depend on the efficiency of the nozzles, which was neglected in the works of Ohlsson [1962] and Suter and Traupel [1959].

Experimental investigation of a single stage turbine was done by Klassen [1968] to determine the effect of different degrees of admission on the performance of a partial turbine. The tests were done with cold air and covered admission degrees of $\varepsilon = 1$, $\varepsilon = 0.51$, $\varepsilon = 0.31$ and $\varepsilon = 0.12$. The losses were divided into rotor pumping losses and all other losses. Investigations showed that the performance decreased as the admission degree was reduced. Klassen made the assumption that the rotor pumping and Windage losses were proportional to the percentage of inactive arc and suggested that all the other losses were constant.

Yahya et al. [1969] did theoretical investigation of a flat plate rotor developed to model partial admission. The theory was adapted to a real turbine rotor operating in partial admission. Experimental measurements and theoretical results were compared for rotors with three different blade pitches.

![Graph](image-url)  

*Fig 2-2 Loss coefficient comparison of partial admission turbines using various formulae and test points, (Yahya et al. [1968])*
The predicted loss coefficient by this method showed larger values than the theoretical results obtained by Suter and Traupel [1959] and Stenning [1953] as depicted in Fig (2-2). In the Figure, B is the length of admission channel and L is the rotor channel length. The difference between the wheels A1, A2 and A3 was only on the rotor blade pitch where A1 had the smallest and A3 had the biggest rotor blade pitch dimensions.

Macchi et al. [1985] reviewed the important loss correlations presented previously by various authors for the partial admission turbines. Using an efficiency optimization method, they obtained efficiency of various partial and full admission turbines operating under similar conditions. They concluded that the results of turbines with moderate size parameters were fairly similar while differences occurred when the compressibility effects or very small sizes were present.

Boulbin et al. [1992] did experimental measurements to obtain the unsteady forces and moments due to cyclic loading and un-loading in partial admission turbines. A partial admission turbine with one blocked arc was used in the study. The turbine was a radial flow machine fitted with rotating nozzles and stationary impulse blades. Two nozzle arcs were filled up in order to recreate partial admission condition. Hydraulic analogy in a rotating water table facility was used between the two dimensional unsteady flow of gas and the two dimensional free-surface flow of water over horizontal surface.

Experimental values of tangential force obtained from the water table were compared to one dimensional unsteady computations, using the characteristic method. The predicted peak forces of partial admission using one-dimensional method was in good agreement with experimental measurements but the general trend showed some discrepancies. It was shown by experimental measurements that the peak force of partial admission decreased with increased nozzle exit Mach number. Boulbin showed that the largest force peak at partial admission operation could reach up to two times of the force at full admission.

Lewis [1993] did experimental measurements of partial admission on a four stage turbine facility. Four partial admission configurations ($\varepsilon = 0.75, \varepsilon = 0.5$ in one blocked arc and two arcs and $\varepsilon = 0.25$) were tested. Results showed that in all of the partial admission cases, the unsteady flow field was almost equalized after the second stage. The author also concluded that the use of multiple flow segments was preferable to one segment because the latter result in significant performance losses.

By development of computational techniques for highly turbulent flows and increased capacity of computers, interest in detailed numerical investigation of the unsteady flows in partial admission turbines increased. He [1997] presented quasi three-dimensional unsteady computations of an axial turbine with admission degree of $\varepsilon = 0.5$ in one blocked arc and two arcs and full admission configurations for single and two stages. He, also presented the unsteady forces of one rotor blade with admission degree of $\varepsilon = 0.5$ and compared with the corresponding experimental data presented by Boulbin, et al [1992] on a water table (Fig 2-3). He, concluded that the trend of forces are in satisfactory agreement. Numerical investigations showed that the efficiency of a single stage turbine of impulse blading at a given admission degree was higher with a blocking arrangement of one segment rather than two. He, explained this effect as the extra
mixing loss for multi segmental blocks. However when a reaction stage was added into the aforementioned impulse stage, the overall performance of the partial admission turbines with one blocked arc and two arcs was almost equal. This showed that the decay rate of circumferential non-uniformities could be more important for performance of a turbine which resulted in efficiency gain in the multi blockage turbine.

![Graph showing aerodynamic force on one rotor blade in 10 blade passing period in a turbine with admission degree of ε = 0.5, (He [1997])]  

Wakeley and Potts [1997] did numerical computations of quasi-3D unsteady flow field for a two-stage partial admission turbine using a multi-row, multi-passage Navier-Stokes solver (same numerical code as He [1997]) and compared the results with experimental measurements of Lewis [1993] and Boulbin et al. [1992]. They suggested that the quasi-three dimensional method was not accurate enough to predict the strong three-dimensionality of the flow in partial admission turbines and full three dimensional, unsteady viscous analyses were required to capture the accurate flow.

Experimental and numerical analysis were presented by Skopek et al. [1999] for a partial admission axial steam turbine stage. It was shown that the axial distance between the nozzle and rotor blades had substantial influence on the loss level of the stage with partial admission. When the admission degree decreased, the optimal value of the velocity ratio also decreased and the distribution of pressure in the circumferential and the radial directions changed.

Bohn et al. [2003] did experimental measurements of a partial admission turbine with admission degree of ε = 0.8 in a scaled down multistage turbine. The inhomogeneous flow structure and temperature attenuation in the stages of the turbine specially the control stage was analyzed. The results showed that the flow inhomogeneity at the inlet of the multistage part was significant, but the flow was equalized after the third stage. While, most of the flow equalization took place within the first turbine stage while the
guide vanes were the main driver for this process. However, the temperature inhomogeneity did not attenuate significantly even at the outlet of the multistage turbine.

Aerodynamic and efficiency measurements on a two stage axial turbine with low reaction blades at different degrees of partial admission were presented by Fridh et al. [2004]. Objectives were to find out the steady and unsteady aerodynamic losses generated by partial admission. They showed that the total to static turbine efficiency dropped with lowering the admission degree (Fig 2-4). In the Figure, $\varepsilon$ is the admission degree, $z$ is the number of admission arcs and $4a$, $4b$ and $4ab$ represent the efficiency of first single stage, second single stage and the two stage configurations, respectively. Results from the steady traverse measurements of static pressure downstream of the blockage showed strong disturbances. Measurements also showed that the static pressure wake resulting from blockage moves almost axially through the turbine while the temperature wake moves in the direction of particle trace.

![Graph](image)

*Fig 2-4 Normalized efficiency vs. velocity ratio, bold lines are the single stage measurements and non-bold lines are two-stage measurements (Fridh et al. [2004])*

Dorney et al. [2004] performed numerical simulations of a full and partial admission turbine using a full 3D Navier-Stokes solver. Results of analysis showed that the simplified models which take advantage of periodic boundaries to simulate the flow of the full admission turbines do not give accurate results for partial admission turbines. In order to assess the performance of turbines in such conditions, the full-annulus of a partial admission turbine need to be modeled numerically.
Herzog et al. [2005] did experimental and numerical investigations on a four-stage axial partial admission turbine at low Mach numbers with varying rotational speeds and mass flows. The aim was to investigate the windage effect at some extreme part load conditions. When the steam mass flow was low, it could not cool down the rotating blades of the turbine thus the kinetic energy of the blades was transferred into thermal energy. Numerical computations were conducted using a 3D Navier-Stokes solver. Total temperature on the blade surfaces of a partial admission turbine with rotational speed of 7500 rpm and mass flow of 0.7 kg/s is shown in Fig (2-5) using numerical calculation data. As it is seen, the highest temperatures were developed in the last third of the turbine. Temperature increase and the peak temperature could be estimated well with the CFD calculations especially in the first three stages.

![Fig 2-5 Total temperature near blade surfaces for rotational speed of 7500 rpm with a mass flow rate of 0.7 kg/s (Herzog et al. [2005])]  

2.1 Summary

From the brief summary, it is seen that the knowledge about the actual flow field of partial admission turbines is rather limited in the open literature. Even though, extensive experimental measurements data are available but the development of numerical models to predict the flow field of partial admission turbines was confined to simple cases. From the validation tests, several researchers have concluded that the simplified numerical techniques and the periodic boundaries are unable to capture the actual unsteady flow of these types of turbines. The choice of admission configuration (single or multi blocked arcs); admission degree and the optimum gap distance between the control stage stator and rotor blades are the main issues in the design of a partial admission turbine. While two dimensional and quasi three dimensional numerical studies of partial admission turbines are performed but full three dimensional numerical studies are considerably limited in the field. Since the flow field in partial admission turbines is extremely unsteady in nature and usually un-symmetrical, a full three dimensional study considering the leakage flow at rotor tip and stator gap would give more reliable information to improve the design.
3. OBJECTIVES

Flow in a partial admission turbine is blocked in one or more segmental arcs of the inlet annulus, therefore the flow going into the downstream blade rows has induced periodic unsteadiness in addition to the unsteady flow of full admission turbines. Numerical modeling of the partial admission turbines requires inclusion of the circumferential non-uniformity of the flow as well as the three-dimensional effects in the span direction. Further improvement of the numerical analysis in partial admission turbines would be to include the modeling of leakage flows in the disc cavities and rotor shroud due to the strong interaction between the main flow and the leakage flows.

The main objectives of the current work were as follows:

- To predict the effect of various admission configurations on the performance and aerodynamics of the turbine by means of validated numerical methods. The various partial admission configurations included the following test cases, partial admission turbine with a single blockage and admission degree of $\varepsilon = 0.762$, partial admission turbine with admission degree of $\varepsilon = 0.524$ in one blocked arc and two opposing blocked arcs, partial admission turbine with single blockage and admission degree of ($\varepsilon = 0.762$) with varying axial gap distances between the first stage stator and rotor rows and the full admission turbine.

- To determine the magnitude and frequency of unsteady forces of first stage rotor blades in partial admission turbines and to compare with the corresponding force components of full admission turbines.

- To determine the importance of interaction between the disc leakage flow and the main flow in the partial admission turbines which also could suggest some practical ways to improve the performance by reducing the leakage flow re-entry into the main flow. Reduction of the leakage flow re-entry would however delay the compensation of the pressure wake in the downstream blade rows and would reduce the second stage efficiency.

- Detailed analysis of the loss mechanisms for the two stage partial admission turbine in various admission configurations and to determine the sources of efficiency losses.
4. METHODOLOGY

Numerical computations were started by modeling of the two stage partial admission turbine with an admission degree of $\varepsilon = 0.762$ in a single blocked arc. For this configuration, measurements of static pressure at various cross sections along the domain were available at design point from a previous experimental work and were used as the validation data.

Due to existence of single blockage, the full annulus of the turbine had to be modeled numerically. The three-dimensional computational model included the rotor shroud and disc cavity leakage flows and the initial numerical results showed the importance of accounting for the leakage flow and the main flow interactions. Validation tests showed very good agreement between the computational results and experimental measurement data at most of the cross sections along the domain but the cross section downstream the first stage rotor row. As was pointed out by He [1995], flow in the first stage rotor row of a two stage partial admission turbine is inherently unsteady and separation of flow takes place in this region. In order to improve the quality of results at this region, the mesh density was increased in the whole circumference of first stage. Furthermore, the inlet boundary was extended from 100% $L_{ax}$ to 250% $L_{ax}$ upstream of nozzles leading edge to study the effect of inlet extension on the flow characteristics of the downstream stages.

Importance of accounting for the three-dimensional effects in modeling of the flow in partial admission turbines was tested by comparison to simpler numerical models. The first model was two dimensional and was created at geometrical midspan of the turbine. Results showed discrepancies in the absolute values between the experimental measurement data and numerical computations, but acceptable agreement was obtained in tendencies. Effect of accounting for the low Reynolds number flow was studied using this model. As was pointed out by Denton [1993] the profile loss of turbine blades can be considered two-dimensional and it can be quantified using cascade tests or two-dimensional computations. The two-dimensional assumption for the profile loss is not totally true for partial admission turbines; however comparison of the results obtained with a two-dimensional model which can resolve the low Reynolds flow around the blade profiles and a model which omits the low Reynolds flow can be beneficial. For this purpose the unsteady tangential forces of first stage rotor blades in one complete cycle was investigated for the aforementioned two-dimensional computational cases.

In another simplified computational model, the domain was extended in the span-wise direction from hub to the casing of the blades but the leakage flows at the rotor shroud and disc cavities were excluded. Computational results showed better agreement with the experimental data than the two-dimensional model but still large discrepancies could be observed. Outcome of this model confirmed that the modeling of leakage flows in partial admission turbines has substantial importance in improving the accuracy of results.

RNG, $k-\varepsilon$ Turbulence model was used together with the non-equilibrium near wall function for computations of the unsteady forces of first stage rotor blade in full and
partial admission turbines. However in some of the computational cases; e. g. the extended inlet partial admission turbine, convergence could not be achieved by this Turbulence model, therefore turbulence modeling was changed to more advanced Large Eddy Simulation (LES) model. The inlet contraction in the extended inlet model has large aspect ratio, therefore the time step size was reduced to half (100 time steps in one stator passage passing period) to model the development of turbulent eddies of the flow in time correctly. LES Turbulence model was also used for obtaining the efficiency of the turbine and the loss mechanisms for various partial admission configurations.

Various partial admission configurations were selected as listed in Table (4-1) to study the various aspects of partial admission turbines. In all of the computational test cases mentioned below, the inlet boundary is placed at 100% upstream of nozzle’s leading edge. Performance change of the partial turbine by varying the axial gap distance between the first stage stator and rotor wheels is studied by the computational cases, 1, 2 and 3 and effect of blocking the inlet annulus in one and two arcs is investigated using the test cases 4 and 5. Full admission configuration is modeled for comparison purposes.

<table>
<thead>
<tr>
<th></th>
<th>Number of admission arcs (z)</th>
<th>Rotational Speed (ω)</th>
<th>Total to static pressure ratio $P_{t2}/P_{s7}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Partial admission $\varepsilon = 0.76$</td>
<td>Design Value</td>
<td>4450 rpm</td>
</tr>
<tr>
<td>2</td>
<td>Partial admission $\varepsilon = 0.76$</td>
<td>120% Design Value</td>
<td>1</td>
</tr>
<tr>
<td>3</td>
<td>Partial admission $\varepsilon = 0.76$</td>
<td>80% Design Value</td>
<td>1</td>
</tr>
<tr>
<td>4</td>
<td>Partial admission $\varepsilon = 0.524$</td>
<td>Design Value</td>
<td>1</td>
</tr>
<tr>
<td>5</td>
<td>Partial admission $\varepsilon = 0.524$</td>
<td>Design Value</td>
<td>2</td>
</tr>
<tr>
<td>6</td>
<td>Full admission $\varepsilon = 1$</td>
<td>Design Value</td>
<td>-</td>
</tr>
</tbody>
</table>
5. EXPERIMENTAL TEST FACILITY

An existing turbine facility was used for investigations of this work. The equipment was manufactured in a cooperation between STAL-LAVAL Co. (Today: SIEMENS Industrial Turbomachinery AB) at Finspång, Sweden and Department of Thermal Engineering (Today: Division of Heat and Power Technology) at Royal Institute of Technology (KTH), Stockholm, Sweden. STAL was responsible for manufacturing and installation of the turbine including the power brake, thermocouples and the pressure measurement system. KTH was responsible for design and installation of the air supply system including the compressor, air cooler, pipes, cooling water system, equipment for the flow measurements and control of the air supply system. The whole installation was in operation in spring 1985, Södergård et al. [1989]. Figure (5-1) shows the position of test turbine and the other components of the system where the experimental measurements presented in this report were performed.

![Diagram of test turbine and wind tunnel room at KTH](image)

**Fig 5-1 Test turbine and wind tunnel room at KTH, (Fridh et al., [2004])**

Air is supplied by two parallel screw compressor units. The air to the inlet of the compressor has the atmospheric temperature and humidity. The high temperature of the air after the compressor is cooled to selected temperature in water cooled heat exchangers. Part of the humidity is condensed but some part (about $4\% \frac{kg\,H_2O}{kg\,Dry\,air}$) remains in the air, then the air passes a flow straightener. Water used in compressor
and heat exchanger is cooled in a cooling tower outside the building. Flow is then straightened in a long pipe with the inner diameter of 300 mm where an ISO mass flow measuring flange is mounted. Air passes the settling chamber of the turbine through a 90 (deg) bend and thereafter passes the turbulence grid and a honey combs before being led into the turbine, Pfefferle [2004]. After the air is passed through the turbine, it is led to a 500 mm diameter tube to the outdoor chimney. A suction fan in the outlet compensates the friction losses. The fan is also used to control the outlet static pressure of the turbine and therefore to keep the pressure ratio across the turbine constant. The main data for the air supply system is listed in Table (5-1) and for the turbine system in Table (5-2) from Södergård et al. [1989].

### Table 5-1: Main data for air supply system, (Södergård et al.; [1989])

<table>
<thead>
<tr>
<th><strong>Compressor type</strong></th>
<th>Atlas Copco ZA6+6</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Maximum working pressure</strong></td>
<td>400 k Pa</td>
</tr>
<tr>
<td><strong>Air volume flow at atmospheric pressure</strong></td>
<td>3.95 (m^3/s)</td>
</tr>
<tr>
<td><strong>Air, mass flow</strong></td>
<td>4.7 (kg/s)</td>
</tr>
<tr>
<td><strong>Number of compressor stages</strong></td>
<td>2</td>
</tr>
<tr>
<td><strong>Power input to compressor shaft</strong></td>
<td>968 (kW)</td>
</tr>
<tr>
<td><strong>Power input at no load</strong></td>
<td>366 (kW)</td>
</tr>
<tr>
<td><strong>Air outlet temperature, full power</strong></td>
<td>180 °C</td>
</tr>
<tr>
<td><strong>Cooling water consumption</strong></td>
<td>2.3 (kg/s)</td>
</tr>
<tr>
<td><strong>Sound pressure level at 1m distance</strong></td>
<td>85 (dB(A))</td>
</tr>
<tr>
<td><strong>Air cooler, cooling capacity</strong></td>
<td>180 °C - 30 °C</td>
</tr>
</tbody>
</table>

### Table 5-2: Main data for turbine, (Södergård et al.; [1989])

<table>
<thead>
<tr>
<th><strong>Number of turbine stages</strong></th>
<th>1 - 3</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Maximum outer diameter of a bladed turbine disc</strong></td>
<td>500 (mm)</td>
</tr>
<tr>
<td><strong>Inner diameter of a disc</strong></td>
<td>280 (mm)</td>
</tr>
<tr>
<td><strong>Maximum speed of rotation</strong></td>
<td>9000 (rpm)</td>
</tr>
<tr>
<td><strong>Type of water brake</strong></td>
<td>Froude FO 209</td>
</tr>
<tr>
<td><strong>Braking power at 9000 (rpm)</strong></td>
<td>750 (kW)</td>
</tr>
<tr>
<td><strong>Maximum braking torque at 7000 (rpm)</strong></td>
<td>1017 (Nm)</td>
</tr>
<tr>
<td><strong>Maximum water flow to the brake</strong></td>
<td>3 (kg/s)</td>
</tr>
<tr>
<td><strong>Type of sump draining pump</strong></td>
<td>Flygt BS 2051</td>
</tr>
<tr>
<td><strong>Torque measurement device</strong></td>
<td>Torquemeters Ltd. ET 250 LS</td>
</tr>
</tbody>
</table>

### 5.1 Turbine Facility

The turbine facility contain a turbine unit, water brake system, torque measuring equipment, a common frame for the three mentioned parts, oil supply system and control devices. Figure (5-2) shows a drawing of the two-stage turbine unit, measurement sections (2 to 7) are indicated in the Figure as was presented in Fridh et al.; [2004].
The airflow is accelerated in a converging channel to the guide vanes. The ring-shaped walls of the housing are exchangeable which means that different test objects can be mounted. The diameter of the blade tip can be up to 500 mm and the diameter of the blade root can be down to 280 mm. The midspan characteristics of the existing test object are as given in Table (5-3). The guide vanes are fastened to the periphery of the nozzle diaphragm and can be revolved about ±15 degrees and adjusted about 110 mm axially. It is also possible to make traverse measurements by revolving guide ring.
### Table 5-3 Test object characteristics at midspan design point, (Fridh et al. [2004])

<table>
<thead>
<tr>
<th></th>
<th>Stage 1 Stator</th>
<th>Stage 1 Rotor</th>
<th>Stage 2 Stator</th>
<th>Stage 2 Rotor</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of Blades</td>
<td>42</td>
<td>58</td>
<td>42</td>
<td>58</td>
</tr>
<tr>
<td>Hub Diameter (m)</td>
<td>0.355</td>
<td>0.355</td>
<td>0.355</td>
<td>0.355</td>
</tr>
<tr>
<td>Tip to Hub Diameter Ratio</td>
<td>1.13</td>
<td>1.17</td>
<td>1.15</td>
<td>1.19</td>
</tr>
<tr>
<td>Pitch to Chord Ratio</td>
<td>0.82</td>
<td>0.81</td>
<td>0.83</td>
<td>0.82</td>
</tr>
<tr>
<td>Aspect Ratio</td>
<td>0.67</td>
<td>1.18</td>
<td>0.77</td>
<td>1.32</td>
</tr>
<tr>
<td>Static Pressure Ratio</td>
<td>1.22</td>
<td></td>
<td>1.23</td>
<td></td>
</tr>
<tr>
<td>Mean Velocity Ratio</td>
<td>0.47</td>
<td></td>
<td>0.47</td>
<td></td>
</tr>
<tr>
<td>Flow Coefficient</td>
<td>0.35</td>
<td></td>
<td>0.36</td>
<td></td>
</tr>
<tr>
<td>Reynolds Number×1E-05</td>
<td>4.3</td>
<td>2.0</td>
<td>3.9</td>
<td>1.8</td>
</tr>
<tr>
<td>Mean Reaction</td>
<td>0.16</td>
<td></td>
<td>0.17</td>
<td></td>
</tr>
<tr>
<td>Shaft Speed (rpm)</td>
<td>-</td>
<td>4450</td>
<td>-</td>
<td>4450</td>
</tr>
<tr>
<td>Flow Turning (deg)</td>
<td>76</td>
<td>133</td>
<td>95</td>
<td>134</td>
</tr>
<tr>
<td>Relative Mach Number at TE</td>
<td>0.48</td>
<td>0.30</td>
<td>0.49</td>
<td>0.32</td>
</tr>
</tbody>
</table>

After passing the turbine blades, the air flow is guided through the annular space between two cylinder walls. At the end of the annular channel there is a plate with many bores. This restriction blocks the flow disturbances from downstream to the turbine flow. The turbine is designed such that it is possible to change the leakage flow around the guide vane from negative to positive values therefore to measure the effect of leakage flow on the measured performance.

**Fig 5-3** Turbine facility seen in the direction of flow, (Fridh et al. [2004])
5.2 Pressure Measurements

Turbine is equipped with a number of pressure probes. Static pressure probes consist of holes at hub and casing of measurement cross sections where the flow velocity is zero. The total pressure probes are placed at midspan, Fig (5-4).

![Total pressure probe upstream the rotor blades, (Pfefferle [2004])](image)

At cross section 2 (position of cross sections are shown in Fig 5-2), eight static pressure probes and six total pressure probes are distributed as shown schematically in Fig (5-5).

![Stationary static and total pressure probes at cross section 2, (Södergård et al. [1989])](image)

The central housing covers the test object. Two traversing pressure probes were installed on the upper part of the central housing. Each instrument had two electric stepping motors for traversing and for turning the pressure probes, Södergård et al. [1989].

The probe was fixed in the guidance pipe of the traverse unit, which could be radially traversed and turned around its own axis. The traverse unit worked with radial steps of 1
μm, and angular steps of 0.01(deg) and it could be controlled remotely from a control room, Pfefferle [2004]. The traversing unit for measurement of pressure at various cross section of the turbine facility is shown in Fig (5-6).

![Traverse unit of test turbine at KTH, (Pfefferle [2004])](image)

**Fig 5-6 Traverse unit of test turbine at KTH, (Pfefferle [2004])**

Fridh et al. [2004] stated, “In order to perform traversing measurements, traversing probes were installed at cross sections 3 to 7. The static hub and casing pressures were obtained by turning the first stator disc during operation. The position of the downstream taps (that were not fixed in the stator disc) was thereby changed relative to the blockage. Due to test rig limitations it was only possible to turn the disc approximately 3 stator passages, and only four static pressure taps were available at the casing and four at the hub, therefore the test was repeated with the blockage shifted in tangential position until data was obtained around the circumference with a circumferential resolution of one degree.”

The inlet flow of the turbine is passed through a 90 (deg) bending before entering into the settling chamber and then passes a turbulence grid. Therefore, the resulting flow has three-dimensional distribution of flow properties. It is desirable to obtain a circumferential and radial total pressure distribution from the measurements at the inlet of the turbine inorder to be able to perform accurate numerical simulation. However, the structural limitations in the test facility does not allow circumferential traverse measurements at cross section 2 and placing several total pressure probes around the circumference was not desirable due to the disturbance it could cause to the main flow.

It is however possible to do a radial and yaw angle traverse by moving the probe in the radial direction and lining it with the flow at only one fixed position. Since the distribution
of pressure at cross section 2 may vary along the circumference, static wall pressure and midspan total pressure from 6 other circumferential positions distributed evenly along the circumference was obtained to adapt the data along the whole circumference; Pfefferle, [2004]. Schematic drawing of the cross section 2 with mounted static and total pressure probes is shown in Fig. (5-7).

Pfefferle, [2004] did the total pressure measurements over a range of operating points for a single stage full admission turbine as presented in Table (5-4). The total pressure values were normalized for various operating points according to Eq. (5-1). At each radial position, the respective dynamic pressure was divided with the dynamic pressure at midspan.
Table 5-4 Turbine operating points for radial traverse measurement of total pressure at inlet

<table>
<thead>
<tr>
<th>Operating Point</th>
<th>Pressure ratio over first stage</th>
<th>Velocity ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Not used</td>
<td>Not used</td>
</tr>
<tr>
<td>2</td>
<td>1.23</td>
<td>0.47</td>
</tr>
<tr>
<td>3</td>
<td>1.23</td>
<td>0.35</td>
</tr>
<tr>
<td>4</td>
<td>1.29</td>
<td>0.43</td>
</tr>
<tr>
<td>5</td>
<td>1.23</td>
<td>0.55</td>
</tr>
<tr>
<td>6</td>
<td>Not used</td>
<td>Not used</td>
</tr>
<tr>
<td>7</td>
<td>1.23</td>
<td>0.65</td>
</tr>
<tr>
<td>8</td>
<td>1.23</td>
<td>0.40</td>
</tr>
<tr>
<td>9</td>
<td>1.35</td>
<td></td>
</tr>
</tbody>
</table>

Results showed that the normalized curves from all the operating points were reduced to a single curve as shown in Fig (5-8).

Fig 5-8 Normalized dynamic pressure profiles for all operating points; Pfefferle [2004]

Pfefferle [2004], used the normalized curve as obtained above to get radial profiles for the other circumferential positions where only two static pressure and one total pressure taps were mounted. He also measured the flow angles for various operating points and showed that the flow angles are close to zero. Linear function was used to connect the radial pressure profiles around the circumference and to approximate a total pressure distribution at cross section 2 for all the operating points.
5.3 Temperature Measurement

Total temperature was measured using eight thermocouples distributed evenly at two radial positions, upstream of cross section 2 (at cross section 1) and downstream of cross section 7 (at cross section 8). Position of the thermocouples is shown schematically in Fig (5-9).

![Fig 5-9 Position of thermocouples at section 1 and section 8, Södergård et al.; [1989]](image)

5.4 Torque Measurements

A torque measuring unit is connected to the turbine facility via coupling. Figure (5-10) shows a photo of the torque measurement device. The torquemeter works with a torsion shaft. The device is temperature compensated for the higher flexibilities due to increased temperature.

![Fig 5-10 Torque measuring device, the turbine is at the left hand side, (Pfefferle [2004])](image)
5.5 Measurement of Mass Flow

Mass flow is measured with a standard orifice plate in the long pipe between condensate water separator and turbine and calculated according to ISO Standards. Figure (5-11) shows inside the pipe where the differential pressure is measured.

![Mass flow measuring orifice](Pfefferle [2004])

5.6 Measurement Uncertainties

Södergård et al. [1989] report the following uncertainties in Table (5-4) based on the results from calibration tests:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Uncertainty</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure</td>
<td>± 0.5 %</td>
</tr>
<tr>
<td>Temperature</td>
<td>± 0.1 °C</td>
</tr>
<tr>
<td>Air mass flow</td>
<td>± 0.2 %</td>
</tr>
<tr>
<td>Torquemeter</td>
<td>± 0.2 %</td>
</tr>
<tr>
<td>Speed</td>
<td>± 0.1 %</td>
</tr>
</tbody>
</table>

Accuracy of the pressure measurement system used in the experimental work presented in Fridh et al. [2004] was better than the values of Table (5-5); accuracy of the rest of the flow parameters are the same as above. Uncertainty of mass flow measurements is ± 0.35% and uncertainty of efficiency measurements is ± 0.9% around the design point efficiency.
5.7 Time Averaging in Experimental Measurements

According to Pfefferle, [2004], the measurement data is continuously logged every second. All measured values are time averaged samples, recorded over approximately 25 seconds, using the last 25 data logs. The reason for this is mainly due to the fact that the inlet temperature is subject to oscillations of about +/-0.3°C. One of the reasons for these oscillations could be due to the control circuit of the water-cooled heat exchanger. The oscillations cause small changes in other turbine parameters, but with the time averaging over 25 seconds the measured values can be seen as steady, since the periodic time of the mentioned oscillations is smaller than 25 seconds.

5.8 Performance of the Turbine

In most of the turbines, flow can be considered adiabatic since the heat losses are negligible compared to the work output. If the flow is assumed to be adiabatic then for a given pressure drop, the second law of thermodynamics can be used to show that the maximum work output is achieved if the turbine is isentropic. Figure (5-10) shows the enthalpy-entropy diagram for a typical stage (1-3) for a multi-stage turbine (A-B). For a multi-stage turbine in which the fluid is exhausted to a given static pressure, it is convenient to define the efficiency as total to static efficiency, Horlock [1966].

$$\eta_0 = \frac{h_{0A} - h_{0B}}{h_{0A} - h_{BS}}$$  \hspace{1cm} (5-2)

![Enthalpy-entropy diagram for a multistage turbine, (Horlock [1966])](image-url)
And the total to static efficiency of one stage (stage 1-3 in the Figure) is given as

$$\eta_{ts} = \frac{h_{01} - h_{03}}{h_{01} - h_{3ss}} \quad (5-3)$$

The overall performance of the turbine is evaluated mechanically by measuring mass flow, turbine shaft output, temperature before the turbine and pressure before and after the turbine. The total enthalpy drop $\Delta h_0$ can be obtained as:

$$\Delta h_0 = h_{0A} - h_{0B} = \frac{W_{\text{shaft}}}{m} = \frac{M \omega}{m} \quad (5-4)$$

where

- $\Delta h_0 = h_{0A} - h_{0B}$ = Total enthalpy drop across the turbine
- $W_{\text{shaft}}$ = Shaft power
- $m$ = mass flow
- $M$ = Shaft momentum output
- $\omega$ = Angular velocity

The ideal gas law determines the density. Knowing the equation of state for an ideal gas

$$\frac{T_{BS}}{T_{0A}} = \left(\frac{P_{BS}}{P_{0A}}\right)^{\frac{(\gamma - 1)}{\gamma}} \quad (5-5)$$

The isentropic, enthalpy drop $(\Delta h_s = h_{0A} - h_{BS})$ is obtained as follows, Lakshminarayana [1996]

$$\Delta h_s = h_{0A} - h_{BS} = c_p T_{0A} \left[ 1 - \left(\frac{P_{BS}}{P_{0A}}\right)^{\frac{(\gamma - 1)}{\gamma}} \right] \quad (5-6)$$

where

- $h_{0A} - h_{BS}$ = Isentropic enthalpy drop from across the turbine
- $c_p$ = Specific heat at constant pressure
- $T_{0A}$ = Total temperature at turbine inlet
- $T_{BS}$ = Static temperature at turbine outlet
- $P_{BS}$ = Static pressure at turbine outlet
- $P_{0A}$ = Total pressure at turbine inlet
- $\gamma$ = Specific heat ratio

The isentropic velocity ratio is defined as below while $U$ is the mean blade speed at midspan.

$$v = \frac{U}{\sqrt{2\Delta h_s}} \quad (5-7)$$
6. NUMERICAL MODELS AND VALIDATIONS

In this chapter, the numerical models used for analysis of the partial admission turbine and the boundary conditions are introduced. The computational results are then validated against the available experimental measurement data. A parametric study is followed to show the influence of various factors in accurate modeling of partial admission turbines.

6.1 Computational Grid

Computational grid for the two stage turbine is fully structured and consists only of hex type of mesh. The computational domain was divided into smaller sub-domains and meshed separately, then attached together to form the final grid (Fig 6-1). In the experimental measurements of partial admission, a physical blockage filled one partial volume of the inlet annulus from the turbulence grid up to the leading edge of the first stage guide vanes. In the numerical computations, partial admission was simulated by removing a volumetric part of the grid from the inlet up to the leading edge of the first stage guide vanes. Since the effect of thermal exchange between blockage and the flow is negligible, it is acceptable to remove the blocked volume of the grid instead of simulating a solid blockage.

![Computational Grid for the Partial Admission Turbine Simulations (Extended Inlet)](image)

In the axial direction, two different inlet extensions were tested. The first inlet boundary was placed at 100% $L_{ax}$ upstream of nozzles leading edge (also called short inlet grid), the inlet boundary at the second computational grid was placed at 250% $L_{ax}$ upstream of
nozzles leading edge (Extended inlet grid). Where \( L_{ax} \) is the axial length of rotor blade chord. Outlet boundary was placed at \( 243\% L_{ax} \) downstream of section 7. Table (A-1) in Appendix A gives positioning of the various numerical boundaries based on the rotor blade chord and important geometrical dimensions of the turbine. The main results of this work are obtained using the short inlet grid; this selection is justified by comparison of the results obtained with varying inlet extension in section (6-6). Complete geometry of the numerical grid (with extended inlet) for simulation of the two stage partial admission turbine is shown in Fig (6-1). Ten out of forty two stator passages are blocked in the Figure.

Since the inlet flow was subsonic, the total pressure was sufficient at the inlet as the boundary condition. At the outlet, the uniform static pressure condition was used. The non-slip boundary condition was used at the walls and the stator-rotor rows were coupled together using the sliding interfaces. Air was assumed to obey the perfect gas law. Sutherland’s law was used to relate the coefficients of viscosity and thermal conductivity to the thermodynamic variables and flow was governed by the Navier-Stokes equations.

Only H type of mesh was used for the first stage stator vanes due to the existence of blockage but O type of mesh was used for the downstream blade rows. Non-slip wall condition was enforced on the walls. Figure (6-2) shows closer view of the various parts of the computational grid. One part of the mesh on the walls of first stage stator and rotor blades is shown in Fig (6-2A). Mesh resolution in the first stage was finer than the second stage due to stronger vortices resulting from admission configuration at the turbine inlet.

The full geometry of the labyrinth seals with direct coupling of disc cavity and shroud leakage flows were included in the computational model. Figure (6-2B) shows part of the volume cells from the gap distance between the first stage stator and rotor discs. Computational grid was blocked by wall boundary right after the labyrinth seals and there was no leakage flow to outside.

The stator disc wall was stationary but the volume cells and the rotor disc wall were rotating with the same angular velocity of the rotors. Figure (6-2C) shows an axial cut of the computational mesh from the first stage rotor shroud. While the upper casing wall was kept stationary, the volume cells and the lower shroud wall were rotating with the same angular speed of the rotors. The shroud was attached to the rest of the grid by interfaces. Figure (6-2D) shows the complete computational grid (with short inlet extension) clipped with 45 degrees angle.
Fig 6-2 Computational grid, (A) Around the blades, (B) Disc cavity between S1&R1, (C) First stage shroud, (D) Complete grid clipped with 45 degrees arc (short inlet)
6.2 Boundary Conditions

Measurements were done over a range of operating points for the single and two stage turbine operating at full and partial admission during the course of an experimental work done by Fridh, et al. [2004]. Data from the measurements are used as the input values for the boundary conditions and validation of computational results of this work. For the partial admission turbine, only the pressure probes at the admission arc were taken into account. The following values were used and the flow properties were assumed uniform in the boundary conditions.

- Total pressure (Spatially averaged from section 2)
- Total temperature (Spatially averaged upstream of section 2)
- Flow direction (Normal to boundary)
- Turbulent parameters (4% turbulence intensity was estimated from the experiments at inlet)

At the numerical outlet, constant static pressure was used. The following parameters were specified at outlet

- Static pressure (Spatially averaged from section 7)
- Backflow conditions (total temperature, ...)

Since the cross sectional area in the outlet flow channel was constant, the outlet boundary condition was taken as the average static pressure from measurements of cross section 7. Table (6-1) gives the main input data into the computational grid.

<table>
<thead>
<tr>
<th>Table 6-1 Input data</th>
</tr>
</thead>
<tbody>
<tr>
<td>$P_{in}/P_{out}$</td>
</tr>
<tr>
<td>Admission degree</td>
</tr>
<tr>
<td>$T_{in} (K)$</td>
</tr>
<tr>
<td>$T_{out} (K)$</td>
</tr>
<tr>
<td>N (rpm)</td>
</tr>
</tbody>
</table>

6.3 Traverse Pressure

The circumferential static and total pressure values are presented and validated against the experimental measurement data at five cross sections along the domain for the partial admission turbine. All the pressure values are normalized with the measured average total pressure at cross section 2 (Numerical inlet total pressure value). Position of the cross sections is shown in Fig (5-2). Figure (6-3) shows the circumferential static pressure at section 3. The time averaged numerical and experimental data are plotted at hub and casing. The static pressure measurement at hub was done at few coarse points. Deep static pressure drop is seen at the entrance to the blocked channel where emptying of the rotor passages occurs. The pressure drop is compensated fast and the static pressure values are higher at the other side of the blockage. Numerical and experimental static pressure curves at cross section 3 show very good agreement both in tendency and absolute values.
Computed total pressure values are plotted at hub, midspan and casing of cross section 3 in Fig (6-4). The three-dimensional effect in the flow is clearly visible. The total pressure values at casing indicate higher velocities in this area. Disturbances in the total pressure field are indication of sources of losses in the flow.

Total pressure measurement values was not available at cross section 3 but it was available at the trailing edge of a few guide vanes from the first stage stator row. Measurements were done close to the blockage sidewalls at midspan. The measured data are compared in Fig (6-5) with the computed total pressure values at the stator’s trailing edge midspan. Stator wakes are clearly visible in the computed total pressure curve. The measured total pressure values show lower values than the computed total pressure close to the stator’s trailing edge; this could be an indication of higher computed velocities than the real flow at this region.
Fig 6-4  Computed total pressure at hub, midspan and casing of section 3

Fig 6-5  Numerical and experimental total pressure at midspan of first stage stator leading edge
Numerical and experimental values of circumferential static pressure at hub and casing of cross section 4 (downstream of first stage rotor row) is plotted in Fig (6-6). The deep pressure drop in the static pressure field seen at cross section 3 has disappeared to a large extent at this section. While the tendency of numerical and experimental curves has good agreement, differences in absolute values. Computed static pressure at the admitted channel of cross section 4 shows higher values compared to the experimental measurement data. It should be mentioned that the results presented in this section were taken from the computations with short inlet grid. It is shown in section (6-6) that by extending the inlet grid, agreement between the computational results and experimental data improves but with the need for a more advanced Turbulence model and much finer time step size.

![Numerical and experimental values of static pressure at hub and casing of cross section 4](image)

**Fig 6-6 Numerical and experimental values of static pressure at hub and casing of cross section 4**

Computed total pressure values are plotted at hub, midspan and casing of cross section 4 in Fig (6-7) using the computational results. The experimental measurements of total pressure existed only at midspan of part of the circumference. It is seen from the computed total pressure that the flow is considerably unsteady and three dimensional downstream the blockage at position of cross section 4. Agreement between the computed and measured total pressure at this section is acceptable.
When partial admission is applied at the inlet of a turbine, pressure and velocity fields at the disc cavities are also affected by the non-uniform flow field. As a consequence of the cavity formed downstream the blockage, the fluid inside the disc cavities are sucked into the main flow, therefore the distribution of the static and dynamic pressure fields inside the disc cavities become non-uniform which in turn imposes large unsteady loads on the discs. Static pressure contours on the cross sections perpendicular to the main flow direction and passing the disc cavities are presented in this section.

Axial position of the presented sections is exactly at the middle of the cavity and the pressures are non-dimensionalized with the inlet total pressure. The absolute velocity vectors are plotted on four radial positions of the discs. Scales of vectors are proportional to the magnitude of absolute velocities but a large numbers of vectors are skipped to increase the clarity of the Figures. The presented contours and vectors are instantaneous. Position of the blockage is shown by the blockage side walls and the smaller arc between the two walls is blocked. The direction of view is from upstream towards downstream.

Figure (6-8) shows the normalized static pressure contours on a cross section passing from the disc cavity between the first stage stator and rotor rows. The low static pressure region seen in the blade passage corresponds to the maximum pressure drop in Fig (6-3).
The fluid inside the disc cavities are sucked into the main flow channel downstream the blockage, flow is then re-entering into the disc in the admitted channel. This process would reduce the turbine's performance since the momentum of the main flow is reduced by interactions with the low momentum leakage flow.

Figure (6-9) shows the static pressure contours on a cross section perpendicular to the main flow and passing the disc cavity between the first stage rotor and second stage stator rows. The fluid inside the disc cavity is seen to be sucked into the main flow more visibly in this section. At some region, there is a split in the rotational flow direction and the velocity vectors are directed opposite to the rotational direction. The reason for fast compensation of the static pressure at cross section 4 can be explained better by the phenomena seen in Fig (6-9).
Circumferential static pressure at hub and casing of cross section 5 is plotted in Fig (6-10). The effect of the blockage has disappeared to a large extent in the static pressure field of this section and it is only seen as a slight disturbance in the static pressure field. The stator wakes are clearly visible in the numerical curves.

Figure (6-11) shows the computed total pressure values at hub, midspan and casing of cross section 5. It is clear from the Figure that the disturbance in the dynamic pressure field is still considerably large downstream the second stage stator row.
Fig 6-10 Numerical and experimental static pressure at hub and casing of cross section 5

Fig 6-11 Numerical total pressure at hub, midspan and casing of cross section 5
At the disc cavity between the second stage stator and rotor rows, the static pressure contours and velocity vectors on four radial positions are shown in Fig (6-12). The slight disturbance seen in the circumferential static pressure of Fig (6-10) is also visible in the main flow channel of Fig (6-12). It is seen that the velocity vectors inside the disc cavity are moving inline with the rotational direction and the cavity flow does not have considerable interaction with the main flow in this region.

Figure (6-13) shows the contours of static pressure in the disc cavity between the second stage rotor and exhaust casing together with the velocity vectors at four radial positions. The static pressure distribution in the disc cavity has become more evenly distributed and the velocity vectors are moving in line with the rotational direction.

**Fig 6-12 Static pressure contours and velocity vectors at a cross section passing the cavity between S2 & R2 (View: Upstream to Downstream)**

Figure (6-13) shows the contours of static pressure in the disc cavity between the second stage rotor and exhaust casing together with the velocity vectors at four radial positions. The static pressure distribution in the disc cavity has become more evenly distributed and the velocity vectors are moving in line with the rotational direction.
The circumferential static pressure values at cross section 6 downstream the second stage rotor row is plotted in Fig (6-14). It is seen that the disturbance in the static pressure field is almost completely equalized downstream the second stage.

Similarly, the computed total pressure at hub, midspan and casing of cross section 6 is plotted in Fig (6-15). The disturbance in the dynamic pressure field is also equalized to a large extent downstream the second stage of the partial admission turbine. Since the traverse measurements existed for one operational point of partial admission (design point), the validation was confined to that case only.
**Fig 6-14 Static pressure at Cross section 6**

- **Experimental Casing**
- **Numerical Casing**
- **Numerical Hub**

**Fig 6-15 Total pressure at Cross section 6**

- **Numerical Casing**
- **Numerical Midspan**
- **Numerical Hub**
6.4 Excluding the Leakage Flows

Effect of improvement in the computational results with the disc cavity and shroud leakage flow modeling in a partial admission turbine is demonstrated in this section. The computational grid with leakage flow modeling is named “Full-3D” and without leakage flow “Simple-3D”. The differences between the Simple-3D and the Full-3D computational models are outlined here. In the Simple-3D model, the disc cavity and shroud leakage flows are excluded and the outlet is placed at position of cross section 7 which is shorter than the Full-3D model. The inlet boundary is placed at the same position of short inlet (100% \(L_{\text{ax}}\) upstream of nozzles leading edge). The total to static pressure ratio along the domains is similar and the admission degree is the same as the Full-3D model (10 out of 42 stator passages blocked in one arc at leading edge). Turbulence model used is RNG, \(k-\varepsilon\) with non-equilibrium near wall modeling. Figure (6-16) shows the complete geometry of the computational grid of Simple-3D model.

![Computational grid of the two stage partial admission turbine without leakage flow modeling](image)

Figures (6-17) and (6-18) show the circumferential static pressure plots at cross section 3 and 4 respectively. The computational results using the Simple-3D model are compared with the experimental measurement data at hub and casing. Results from validation of computed static pressure at cross section 3 and 4 show that the leakage flow modeling has considerable importance in improving the accuracy of numerical results.
Fig 6-17  Static pressure at cross section 3  (Simple-3D model and experimental data)

Fig 6-18  Static pressure at cross section 4  (Simple-3D model and experimental data)
It is also seen that the static pressure values at hub and casing are very close to each other while this was not the case when the interaction between the disc cavity and shroud leakage flows with main flow was modeled.

A comparison is made here between the measured total torque of the rotor shaft at the design speed for the admission turbine and the time averaged, computed torque on the turbine in Fig (6-19).

To obtain the velocity ratio, the blade speed at midspan was divided with the total to static enthalpy drop of the two stage partial admission turbine. Results are presented using computational grids with short and extended inlets. Extended inlet grid is introduced in section (6-6). The computed results are also presented for the grid excluding the leakage flow modeling (Simple-3D).

Each measurement point is time averaged from 20 sequential torque measurements at a single operating point. All the presented operating points are close to the design point. The admission degree and the pressure ratio along the turbine are similar for all the presented numerical and experimental cases.

![Graph showing computed and measured torque on the rotor shaft of the two-stage partial admission turbine]

The computational results with extended grid show slight improvement compared to the short inlet, however the difference is small. It is seen that the leakage flow modeling has improved the accuracy of the computed momentum considerably.
6.5 Two-Dimensional Model

In an axial turbine, if the radial distribution of the flow parameters could be assumed uniform and the flow at the midspan could be representative of the flow in the whole span, the two-dimensional model (2D) could be used. In the presence of strong swirling flows and three-dimensionality in the flow-field, the 2D model is not a reasonable choice and the numerical domain should be 3D. However, by using a 2D computational model, the tendency of variation of flow parameters and characteristics of the flow could be captured qualitatively with greater save on the computational resources.

Boundaries of the 2D grid were confined at midspan to one cell row with one millimeter height in the span direction. Boundary condition at the span direction was set to symmetry. Full annulus of the turbine was modeled and partial admission was simulated by cutting one part of the mesh from inlet up to the leading edge of the first stage guide vanes. Extension of numerical grid in the axial direction was from 100% \( L_{\text{ax}} \) upstream of nozzles’ leading edge to cross section 7.

Figure (6-20) shows the two-dimensional model of the two-stage turbine with single blockage. Geometry of the full annulus model is seen at the left side and a part of the computational grid in the right side.

Traverse static pressure at cross section 3 is plotted in Fig (6-21) using the 2D numerical data and experimental measurement values. The static pressure values are normalized with the inlet total pressure (Note that the normalization factor in the 2D results of the appended paper, Hushmandi et al.; [2007] is different).
It is seen that the two-dimensional numerical model has captured the tendency of experimental curve at most of the circumference. However, the effect of stator vanes is evened out downstream the blockage in the numerical results; this indicates that the fluid is emptied there while the experimental data shows the wakes clearly. Static pressure values at the admitted channel show higher values than the experimental measurement data and at the blocked channel show lower values. It is not possible for the fluid inside the admitted channel to interact with the blocked channel fluid as well as in the 3D model.

Figure (6-22), shows the normalized circumferential static pressure at cross section 4. Experimental hub and casing values is compared to the two dimensional numerical data. Effect of the blockage is decreased in the static pressure field of this section and the deep pressure drop has disappeared. The 2D numerical results have captured the tendency of experimental curve in this cross section to a good extent however there is still large difference in the absolute values.

Presented validation data from the two dimensional model show that the 2D model can capture the tendency of the experimental measurements with good accuracy however the absolute values are not in satisfactory agreement. It was suggested by other authors, (e. g. Wakeley and Potts; [1997]) that the effect of three-dimensionality was important in modeling the partial admission turbines because of the admission configuration and strong unsteadiness in the flow.
6.6 Effect of mesh resolution near blade profiles

Effect of mesh resolution on the results is evaluated with the two-dimensional model in Fig (6-23). In the coarse grid, the near wall mesh is coarse and the RNG, k-ε Turbulence model is used. In the finer grid, the near wall mesh resolution is higher and the near wall mesh density is about $y^+ = 1$ and SST, k-ω Turbulence model is used.

Tangential force for one of the rotor blades, travelling along the circumference is plotted using the two 2D grids. It is seen that the tangential force for both of the cases is similar in almost all of the circumference. The finer grid however show smaller force when the blade enters the blocked channel. This could be due to the ability of the finer grid to better modelling the separated flow in the sector end of the blockage.
Fig 6-23 Tangential force of one rotor blade travelling along the circumference, using 2D models and difference mesh resolution near the blade profiles

6.7 Extension of Numerical Inlet

In order to find out the extension of numerical inlet that can capture the distribution of flow parameters correctly, two different grids were tested. Inlet boundaries in the two computational grids were placed at 100% $L_{ax}$ and 250% $L_{ax}$ upstream of nozzles' leading edge and the grids were called, “short inlet” and “extended inlet” grids respectively. Outlet boundaries for both of the cases were similar and were placed at 243% $L_{ax}$ downstream of section 7. Since there was not measurements of total pressure available upstream of cross section 2, the average total pressure measurements from 6 total pressure probes in cross section 2 was used as the constant inlet boundary condition in both cases. Consequently, the total to static pressure ratio for both cases was the same while the extension of grid in one of the grids was 150% $L_{ax}$ longer at the inlet.

The mesh used for both of the computational cases were similar, however relatively finer mesh was used at the inlet part of the extended grid due to the large aspect ratio of the inlet cone. The RNG k-ε Turbulence model with non-equilibrium near-wall modeling was used to start the simulations in both cases. While the short inlet grid showed acceptable convergence behavior, the extended inlet grid showed physically un-realistic results.
(reversed flow at inlet). By switching to LES turbulence model, the reversed flow problem at the inlet was solved completely; however the solution was not stable. In a further attempt to improve the solution of the extended grid, the time step size was reduced to half (100 time steps in one stator pitch passing period). Acceptable convergence in the results was obtained thereafter.

Figure (6-24) shows the circumferential static pressure values at midspan with the two different inlet extensions and the experimental measurement data. It should be mentioned that the measurements were taken from 8 static pressure probes placed inside the admitted channel only and the obtained data were distributed evenly along the circumference; therefore the measured values seen inside the disc cavity does not represent the real static pressure at the blocked channel. The frequency of stator vanes is seen to be dominant in the static pressure field of short inlet model at cross section 2. However from the extended inlet data, it is seen that the dominant frequency in the static pressure seems to be a combination of stator and rotor blade numbers.

![Fig 6-24 Static pressure at cross section 2](image)

The circumferential values of total pressure at cross section 2 are plotted in Fig (6-25) for the short and extended inlet models. Total pressure was also measured in cross section 2 with 6 total pressure probes distributed evenly along the circumference at
midspan. Position of the blockage was changed in relation to the fixed total pressure probes during the measurements but the data presented here are taken from the measurements of probes placed completely inside the admitted channel. The purpose is to show the variations of total pressure amplitude at the admitted channel.

![Graph of total pressure at Cross section 2]

It is clearly seen from the computational results compared with the experimental data at cross section 2 that the extended inlet model can capture the disturbances in the static and total pressure fields more accurately while the short inlet grid is unable to capture these variations. Since the agreement between the computational results of short inlet grid and the experimental data was the worst in the static pressure values of cross section 4, the comparison is repeated here using the data obtained with the extended inlet grid.

Figures (6-26) and (6-27) show the static pressure plots at hub and casing of cross section 4. It is seen that the static pressures obtained with the extended grid show considerable improvement compared to the results obtained with the short inlet grid. It should however be mentioned that the computational time required to travel one stator pitch with extended grid (LES Turbulence model and finer time step) was twice the time required for short inlet grid (coarser time step, RNG k-ε Turbulence model).
Fig 6-26 Static pressure at casing of cross section 4

Fig 6-27 Static pressure at hub of cross section 4
The radial dynamic pressure normalized with the dynamic pressure at midspan are plotted at several angular positions inside the admitted channel in Fig (6-28). The selected angular arc equals to two stator pitches and the angles have equal separation arcs. The results can be compared qualitatively with the results obtained for single stage configuration by Pfefferle [2004], Fig (5-8). Pfefferle [2004], measured the total pressure profile at one angular position and corrected the curve by measurements from the static wall and midspan total pressure from the other fixed pressure probes along the circumference. By comparing the two Figures, it is seen that the tendency of radial pressure profile is different than that obtained by Pfefferle [2004].

![Fig 6-28 Normalized radial dynamic pressure profile at cross section 2](image-url)
7. SELECTED RESULTS

It was shown in chapter 6 that the effect of leakage flows of the disc cavities and rotor shroud is essential in accurate modeling of partial admission turbines. Therefore in this chapter, the three-dimensional computational grid including the leakage flows is employed for further investigations. The effect of blocking the inlet flow in a single arc and two opposing blocked arcs is examined for admission degree of $\varepsilon = 0.524$. In another partial admission configuration with admission degree of $\varepsilon = 0.726$, the gap distance between the first stage stator and rotor rows is varied by 20% compared to its design value and the effect on the turbine performance is investigated. In addition, the magnitude and frequency of the unsteady forces of first stage rotor blades are obtained for the partial admission turbine with a single blockage.

7.1 Turbine Performance in Various Partial Admission Configurations

Experimental total to static efficiency of the test turbine facility at various admission configurations are reported here from Fridh et al. [2004]. Experimental efficiencies are plotted over a range of flow coefficients while the numerical data are plotted at design operating point. Flow coefficient is defined as below.

$$\phi = \frac{c_{ax}}{U} \quad (7-1)$$

The total enthalpy drop is calculated using the torque exerted on the rotor shaft, Eq. (7-2). The density is determined by the ideal gas law and the isentropic total to static enthalpy drop is defined by Eq. (7-3).

$$\Delta h_t = \frac{P}{m} = \frac{M\omega}{m} \quad (7-2)$$

$$\Delta h_s = c_p T_{0,\text{in}} \left[1 - \left(\frac{p_{\text{out}}}{p_{\text{out},\text{in}}}\right)^{\frac{(y-1)}{y}} \right] \quad (7-3)$$

Figure (7-1) shows the total to static efficiency of the two stage turbine in various partial admission configurations. The efficiencies are normalized with the maximum experimental efficiency of the two stage full admission turbine. Computational results show that the partial admission configurations with admission degree of $\varepsilon = 0.524$ in a single blocked arc and two blocked arcs have almost identical two-stage efficiencies. The flow coefficient is slightly lower for the single blocked arc model while the operating...
point for both of the computational grids is similar. Experimental results show that the partial admission turbine with single blockage and admission degree of $\varepsilon=0.524$ has better efficiencies over a large range of flow coefficients compared to the double blockage model.

![Graph](image)

**Fig 7-1 Efficiency of the two stage turbine at various partial admission configurations**

Computational results for the partial admission turbine with admission degree of $\varepsilon=0.726$ and various gap distances between the first stage stator and rotor blades show that the reduced and design gap models have similar efficiencies at the design operating point. However the model with increased gap distance shows considerably lower efficiency.

First stage efficiency from the two stage turbine is calculated using the mass averaged flow parameters downstream the first stage and computed first stage rotor torque. Total to static efficiencies of the first stage for various computational cases are normalized with the experimental maximum full admission efficiency of the two stage turbine ($\eta_{\text{max}}$) and are plotted in Fig 5.

It is seen that the performance of the first stage is higher in the partial admission turbine with two blocked arcs than single blocked arc model having the same admission degree. Efficiency of the first stage for the partial admission turbine with admission degree of $\varepsilon=0.726$ and varying gap distances shows that the reduced gap model have higher efficiencies than the alternative gap distances. In the sections (7-3) to (7-5), it is attempted to explain the reason for the variation of efficiency in various configurations of partial admission turbine by detailed analysis of the flow field.
7.2 Unsteady Forces of First Stage Rotor Blades

Rotor blades in a partial admission turbine are under cyclic loading and unloading due to the periodic disturbance of the inlet flow. This process imposes unsteady forces with large amplitudes especially to the first stage rotor blades. It is rather difficult to build up responsive experimental devices to measure the unsteady forces of rotor blades therefore the computational tools are of primal importance to obtain the detailed forces in such conditions. The computed unsteady axial and tangential forces are presented in this section for the two stage turbine operating in full and partial admission with admission degree of $\varepsilon=0.726$.

Figure (7-3) shows the normalized unsteady tangential forces of the first stage rotor blades for the full-admission turbine. Several blades of the first stage rotor are followed for a quarter of circumference and the resulting forces are plotted together around the circumference to obtain the unsteady forces in the whole circumference of the full admission turbine. The force values are reduced with the average force inorder to identify the real amplitudes. The local peaks seen in the tangential forces are resulted from the stator wakes. It is difficult to identify the frequency content of disturbances from the time domain; therefore Fourier Transform is used.
The unsteady forces of rotor blades have a period of $2\pi$; this means if $f(x)$ represent the force function, for all $x$ in $-\infty < x < \infty$, $f(x + 2\pi) = f(x)$. It is useful to represent such a function by complex Fourier series as below inorder to identify the magnitude and frequency of force components.

$$ f(x) = \sum_{j=-\infty}^{\infty} c_j e^{ijx} \quad \text{where} \quad i = \sqrt{-1} \quad (7-4) $$

The complex Fourier coefficients can be computed by the integration below:

$$ c_j = \frac{1}{2\pi} \int_0^{2\pi} f(x)e^{-ijx} \, dx \quad (7-5) $$

The Fourier series are truncated and harmonics are used up to a finite number. Using the Fast Fourier Transform (FFT) algorithms in Matlab 7.8.0, approximate Fourier coefficients are computed. Amplitude vs. Frequency of the unsteady tangential forces on the first stage rotor blades of full-admission turbine is computed and plotted in Fig (7-4). As was expected, the largest amplitude is seen at 42 multiple of rotational frequency (the stator vane number).
The axial component of the unsteady force vector for the first stage rotor blades of full admission turbine are plotted in time domain in Fig (7-5) and in frequency domain in Fig (7-6). Similarly, wakes resulted from the stator vanes produce the largest amplitude of unsteady axial forces.
First stage rotor blades in a partial admission turbine change both magnitude and direction while travelling along the circumference. Normalized tangential forces of two adjacent first stage rotor blades travelling along the whole circumference in the two stage partial admission turbine is plotted in Fig (7-7).
Downstream the blockage, emptying of the rotor channel occurs. When the blades are about to enter into the blocked channel, the tangential force increase in direction of rotation due to the sudden pressure drop in the suction side of the blade surface. Inside the blocked channel, magnitude of the tangential force is small but disturbances occur with considerable amplitude due to the interaction of the main flow and leakage flow. At the exit of blocked channel, the rotor passages are refilled with the high pressure fluid and the tangential force return to the through flow magnitude. The effect of the blockage disturbance can be seen at about half of the circumference. It is seen that the adjacent blades have slightly different amplitudes downstream the blockage. This can also be observed from the axial force plots in Fig (7-9).

The unsteady tangential forces of partial admission in frequency domain for the two adjacent blades are plotted in Fig (7-8). The largest amplitudes are seen at the first and second multiples of the rotational frequency. The first multiple is due to the wake of the single blockage and the second is due to the change in the direction of the tangential force component at the sides of the blockage. The other significant frequencies that are added into the system due to partial admission are at 3, 5, 6, 9 and 10 multiples of rotational frequency. Although the magnitude of stator wakes is in the same order of full admission turbine, but it is hard to identify it due to the other larger amplitudes present in the frequency domain.

![Fig 7-8 Tangential forces of first stage rotor blades in partial admission turbine in frequency domain](image-url)
Normalized axial forces of two adjacent first stage rotor blades travelling along the circumference in the two stage partial admission turbine are plotted in Fig (7-9). A deep drop is seen in the axial force magnitude at the entrance to the blocked channel. Due to mixing process, the magnitude of axial force increase quickly downstream the blockage. The local peaks of axial force at the exit of the blocked channel indicate a strong mixing process at this region. This idea can be confirmed by looking at the change in the direction of tangential force at the same region in Fig (7-7).

The frequency content of the unsteady axial force of the two adjacent first stage rotor blades at partial admission turbine are plotted in Fig (7-10). The 2\textsuperscript{nd}, 3\textsuperscript{rd} and 1\textsuperscript{st} multiples of the rotational frequency are present in the axial force. Currently, there are not measurements of unsteady force available for the investigated turbine facility to compare with the computed forces; however the measured total torque on the rotor shaft around the design speed was presented for the two stage partial admission turbine and compared with the time averaged, computed torque on all the rotating walls of the turbine in Fig (6-19).

\begin{figure}[h]
\centering
\includegraphics[width=\textwidth]{fig7-9.png}
\caption{Computed axial forces of first stage rotor blades at partial admission} 
\end{figure}
7.3 Relative Flow Angles

Relative flow vectors are plotted in Fig (7-11) at midspan of a rotor blade placed at the admitted channel of partial admission turbine with reduced gap distance and admission degree of $\varepsilon=0.726$. The velocity vectors are scaled with the velocity magnitude. The relative velocities are considerably lower at the trailing edge of the blade suction side and the vectors are revered compared to through flow at some points.

Fig 7-11 Relative flow vectors around a rotor blade at midspan of partial admission turbine with reduced axial gap (Admitted channel)
The circumferential relative flow angles are plotted in Figures (7-12), (7-13) and (7-14) for the partial admission turbine with admission degree of $\varepsilon=0.726$ and varying axial gap distances between the first stage stator and rotor rows. Figure (7-12) shows the relative flow angles for the design axial gap model. At the admitted channel the flow turning is around 133 degrees. Downstream the blocked channel, considerable disturbances are seen in the flow angles. The disturbance is caused by redirection of the velocity vectors at the low pressure region downstream the blocked channel.

![Fig 7-12 Relative flow angles upstream and downstream of first stage rotor’s midspan, design gap](image)

Figure (7-13) shows the flow angles of the reduced axial gap distance model of partial admission turbine. The disturbances seen in the inlet flow angles are considerably lower downstream the blocked channel compared to the design axial gap model. However it is seen that the relative flow angles at the outlet of blade row are more disturbed compared to the design gap model in the admitted channel. This is observed in the trailing edge of the rotor blade shown in Fig (7-11) and can be due to the change of axial gap from the design value which would change the inlet flow angle to the rotor.
The partial admission turbine with increased axial gap distance between the first stage stator and rotor rows, shows considerable disturbance of the inlet relative flow angles downstream the blocked channel. Disturbances are also seen in the outlet flow angles of admitted channel.

The higher efficiency of first stage in the partial admission turbine with reduced axial gap can be related to the smaller disturbances of the flow at the rotor inlet. However the disturbances of the admitted channel flow at the rotor outlet would cause the efficiency to decrease in the downstream stage. The increased gap model had the lowest efficiency of the first stage; the largest disturbance of the flow at the rotor inlet was seen in Fig (7-14) of the partial admission turbine with increased axial gap distance. It is seen from Figures (7-12), (7-13) and (7-14) that the flow in a low reaction turbine migrate in the direction the nozzles send it.
Fig 7-14 Relative flow angles upstream and downstream of first stage rotor's midspan, Increased gap model

7.4 Axial Force of Rotor Wheels

Modelling the leakage flow at the disc cavities gives the possibility to obtain the axial force on the rotor discs. Normalized axial force of the first and the second stage rotor discs are presented in Fig (7-15) for the various partial admission configurations. The presented axial force is the sum of the total forces on the walls of all the components of the rotor wheel including the blades.

In the partial admission turbine model with admission degree of $\varepsilon=0.726$ and various gap distances between the first stage stator and rotor rows, it is seen that the model with reduced axial gap distance has higher axial force of the first disc than the other models. This could be due to the higher pressure in the axial gap between the first stage stator and rotor disc cavities giving rise to the net axial force of the rotor wheel.
In order to find out the reason for considerable change of the axial force when the axial gap distance is varied, static pressure contours and the velocity vectors at an axial section passing the main flow and the disc cavity between the first stage stator and rotor rows. Figures (7-16) and (7-17) are from the design and reduced axial gap models, respectively with single blocked arc and admission degree of $\varepsilon=0.726$. The static pressure values are normalized with the inlet total pressure. The velocity vectors are plotted in four radial positions of the disc. The selected radial positions between the first stage stator and rotor rows are placed at 0.51, 0.66, 0.82 and 0.97 of hub radius from the disc centre. Similarly, the selected radial positions between the first stage rotor and the second stage stator are placed at 0.49, 0.65, 0.82 and 0.99 of hub radius from the disc centre.

The vectors lengths are scaled with the magnitude of the velocity and both of the Figures (7-16) and (7-17) have the same scale factor for velocity. The blockage positions are shown with blockage side walls, the smaller arc between the blockage sidewalls is blocked and the direction of rotation is shown with an arrow in the Figures.
The static pressure values are considerably higher at the disc cavity in the partial admission turbine with reduced axial gap distance and the velocities are much lower. The leakage flow inside the disc cavities does not interact much with the high momentum flow of the blade passages in the partial admission turbine with reduced gap distance.
Fig 7-17 Static pressure contours and velocity vectors at cross section passing the disc cavity and main flow between S1&R1 (reduced gap, $\varepsilon = 0.762$), view upstream to downstream

When the axial force of the partial admission turbines with admission degree of $\varepsilon=0.524$ in a single blocked arc and double opposing blocked arcs are observed in at Fig (7-15), it is clear that the axial force of the first stage rotor wheel is considerably larger when a single blockage is used (Both cases have the design axial gap distance). The normalized static pressure contours together with the velocity vectors are plotted for the flow around the first stage rotor wheel for both configurations in Figures (7-18) and (7-19).
Direction of velocity vectors in Fig (7-19) shows that the main flow is entering the disc cavity at the admitted channel and re-enters into the main flow downstream the blockage. This process (mixing of the low momentum leakage flow with high momentum through flow) would considerably reduce the efficiency. However it would help to compensate the static pressure disturbance of the main flow.
Fig 7-19 Static pressure contours and velocity vectors at cross section passing the cavity and main flow between R1 & S2 (Single Blockage $\epsilon = 0.524$), View upstream to downstream

Similarly, the static pressure contours and velocity vectors are plotted for the flow around the first stage rotor wheel for the partial admission turbine with double opposing blockages and admission degree of $\epsilon = 0.524$ in Figures (7-20) and (7-21).
Comparison of Figures (7-20) and (7-21) shows that the pressure is higher in the disc cavity of partial admission turbine with single blockage giving rise to the net axial force of first rotor wheel as was seen in Fig (7-15). The higher pressure might be due to the less interaction between the leakage flow inside the disc cavities and the main flow.
Fig 7-21 Static pressure contours and velocity vectors at cross section passing the cavity and main flow between R1 & S2 (Double Blockage $\varepsilon = 0.524$), View upstream to downstream

7.5 Coefficient of Loss

In turbomachinery flow, the isentropic work is defined as the ratio of actual work to the isentropic work, therefore the only factor which can change the efficiency in an adiabatic machine is departure from isentropic process. There are different definitions of loss coefficients, a loss coefficient is defined here according to Denton [1993],

$$\zeta = \frac{T_{s,\text{out}} \Delta s}{h_{o,\text{out}} - h_{s,\text{out}}}$$

(7-6)

The loss coefficient is plotted downstream the first stage and second stage rotor rows for the partial admission turbines with admission degree of $\varepsilon = 0.524$ in one blocked arc and two opposing blocked arcs in Figures (7-22) to (7-25).
Comparing Figures (7-22) and (7-23), it is clear that the loss downstream the first stage rotor row for the partial admission turbine with two blocked arcs is lower than the single blocked arc model. It is seen that the mixing losses are large where the rotor blades exit the blocked channel. The partial admission turbine with single blockage has considerably larger losses in the disc cavity downstream of the first stage. The leakage flow enters into the blocked channel downstream the blockage and re-enters into the disc cavity flow at the admitted channel, this phenomena produces larger losses in the
first stage of the single blockage turbine which in turn affects the efficiency of the first stage.

Figure (7-15) showed higher overall efficiency for partial admission turbine with smaller blocked arc ($\varepsilon = 0.726$) compared to the turbine with larger blockage ($\varepsilon = 0.524$). This can be explained by the phenomena seen in Fig (7-23). The flow of the partial admission turbine with smaller blockage would have lower losses due to less interaction of the main flow and the leakage flow inside the disc cavities.

However, the losses are larger in the partial admission turbine with two blocked arcs downstream the second stage rotor row compared to the single blockage model (Figures (7-24) and (7-25)). The losses decrease the efficiency of second stage in double blockage turbine. The results of loss coefficient are in agreement with the efficiency plots of Figures (7-15) and (7-16).
Fig 7-24 Loss coefficient ($\varepsilon = 0.524$ in two arcs), at a cross section downstream of R2, passing the disc Cavity and main flow, View upstream to downstream
Fig 7-25 Loss coefficient ($c = 0.524$ in one arc), at a cross section downstream of R2, passing the disc Cavity and main flow, View upstream to downstream
8. CONCLUSIONS AND DISCUSSION

8.1 Summary and Discussion

Numerical analyses are performed in this work to assess the performance of partial admission turbines in various configurations. Furthermore, the unsteady forces of first stage rotor blades and the non-uniform pressure distribution of the rotor discs of partial admission turbines are analyzed extensively. For this purpose, geometrical dimensions of an existing two stage axial turbine with low reaction blades are used. For the numerical computations of this work, ANSYS Academic CFD package was used. The geometry and the computational mesh were produced in ICEM-CFD 11.0 and the solver was FLUENT 6.3.16 and FLUENT 12.1. Circumferential distribution of static pressure at various cross sections of the two stage turbine existed for one partial admission configuration from a previous experimental measurement work. In addition, performance measurements existed over a range of operating points for various partial admission configurations. The mentioned measurement data were used as the initial condition and validation data for the computational results of this work. Validation tests were performed to obtain the suitable computational grid that could capture the physics of the unsteady flow of partial admission turbines. The geometry of the validated computational grid was then modified to simulate various configurations of partial admission in axial turbines and to analyze the flow field in such conditions.

The partial admission configuration chosen for validation of computational grid had an admission degree of $\varepsilon=0.726$ in a single blocked arc. Since the geometry of the turbine with single blockage was unsymmetrical, the full annulus of the turbine had to be modeled numerically. The computational mesh was three-dimensional; furthermore, it included the leakage flows of rotor shroud and disc cavity clearance. The complete three-dimensional model contained around 10 million cells and the computational time to perform one complete revolution was around two months (continuous) on four parallel computational nodes, while each node had 8 GB of main memory and eight processors. Comparison of the computed circumferential static pressure with the corresponding experimental data at various cross sections along the domain showed very good agreement, however some discrepancies were seen downstream the first stage rotor row. In order to find out the sources of discrepancies, effect of various parameters were tested on the accuracy of the obtained computational results.

Effect of mesh resolution near the wall boundaries was tested using a simple two-dimensional model. The two-dimensional model contained one cell row in the span-wise direction with one millimeter thickness at the geometrical midspan of the turbine. Coarse mesh near the blade profiles together with the near-wall models was used in the first computational test case. In the second two-dimensional case, finer mesh near the blade profiles was used together with low Reynolds number Turbulence model. Unsteady tangential force component of a first stage rotor blade travelling along the circumference was plotted using the results of the coarse and the finer computational grids. Comparison of the tangential forces did not show considerable differences; therefore the low Reynolds number effects were neglected for further computations of this work.
Comparison of the circumferential static pressure at various cross sections along the domain showed that the simple two-dimensional numerical model could capture the tendency of experimental measurements but the absolute values showed large discrepancies. The two-dimensional model contained around 500 kilo to 1 million computational cells depending on the mesh resolution near the wall boundaries. The computational time to results of one complete revolution was in the range of two days on a single node with 8 GB of main memory and double processors.

Effect of modeling leakage flows was assessed by comparison to a simple three-dimensional model excluding the flow at the disc cavities and the rotor shroud. Comparison of the obtained computational results of the simple three-dimensional model with the experimental data showed discrepancies in the absolute values. Furthermore, when the forces of first stage rotor blades were plotted using the computational models with varying complexity, effect of main flow and leakage flow interactions on the force components downstream the blockage was observed while the simpler computational models were unable to capture the effect. The simple three-dimensional model excluding the leakage flows contained around 6 million computational cells and the computational time to perform one complete revolution was around two months on three parallel computational nodes, while each node had 8 GB of main memory and double processors.

Effect of grid extension at the numerical inlet was assessed for the three-dimensional computational model including the leakage flows. It was observed that the computational grid with extended inlet could improve the computational results at the cross section downstream the first stage. However, the time step size had to be reduced to half and the Turbulence model had to be changed to a more advanced model to accurately capture the turbulent eddies of the inlet contraction.

Finally, the validated computational grid was used as the base for computation of various configurations of partial admission turbines. The effect of blocking the inlet flow in a single arc and two opposing blocked arcs was tested for an admission degree of $\varepsilon=0.524$. Effect of gap distance between the first stage stator and rotor blade rows was tested by varying the gap distance by 20% compared to its design gap value. In additions, performance of the turbine in various partial admission configurations together with the physical analysis of the performance variation in different configurations was followed.

### 8.2 Conclusions and Future Work

The main conclusions and suggested future work are presented here:

- First stage rotor blades are under cyclic change of unsteady forces due to the admission configuration at the turbine inlet. Plotting the unsteady forces of the first stage rotor blade of a partial admission turbine with single blockage in the frequency domain showed that the largest amplitudes are at the first and second multiples of rotational frequency; this is due to the blockage disturbance and the change in direction of the force vector. The other dominant amplitudes of
tangential force were seen at 3rd, 5th and 6th multiple of rotational frequency. For axial forces, the 2nd, 3rd, 4th, 6th, 7th, 8th, 9th and 13th multiples of rotational frequencies were dominant.

- Effect of interaction between the main flow of the blade passages and the leakage flow of the disc cavities and rotor shrouds cannot be omitted in numerical analysis of partial admission turbines. In partial admission turbines, the rotor discs have non-uniform static and dynamic pressure distributions, especially in the first stage. A significant part of the fluid close to the hub in the admitted arc is sucked into the intra-stage disc cavity. The flow interacts with the cavity flow under the influence of rotor rotation before discharging into the low pressure region downstream of the blockage. If this flow could be reduced, the output from the first rotor could increase, but the second stage would have a more non-uniform flow and lower efficiency. This phenomenon also explains the reason for fast compensation of the static pressure drop downstream of the blockage.

- Gap flow modelling in the partial admission turbines also enables estimation of the thrust loads on the discs. Computational results of axial force on rotor wheels for various partial admission configurations showed that the blockage length, multi-blocking and axial gap distance between the first stage stator and rotor blades has direct impact on the disc force. As the axial gap distance is reduced than the design value, the axial force of rotor disc increases. Furthermore, the partial admission turbine with single blocked arc has larger disc force, than the double blocked inlet arc.

- Effect of multi-blocking and axial gap distance was analysed on performance of the two stage partial admission axial turbine. Performance results showed that blocking the inlet annulus in two arcs produce better efficiency at the first stage compared to the single blocked arc, however due to extra mixing losses; the efficiency of double blockage model deteriorates in the downstream stage.

- Computational results showed that changing the axial gap distance between the first stage stator vanes and rotor blades have considerable effect on performance of partial admission turbine. When the axial gap distance is reduced than the design value, the interaction between the leakage flow inside the disc cavity and the main flow decreases which in turn result in better efficiencies of the first stage. When the axial gap distance is increased, the efficiency of the first stage decreases due to the extra losses associated with mixing of main flow and disc cavity leakage flow. However, the overall efficiency of the reduced gap model decreases in the second stage.

- The future work of this thesis could be to perform the stress analysis of the rotor blades. Furthermore analysis of the disc forcing function with respect to disc mode shapes to find out if certain blockage configurations are dangerous for disc mode excitations.
9. REFERENCES


"Turbine Blade Forces due to Partial Admission"; RFM, Revue Francaise de Mecanique, n3, 1992, p. 203-208, ISSN 0373-6601

Cebeci, T. and Bradshaw, P.; 1977

Dannenhoffer, J. F. and Baron J. R., 1985
"Grid adaptation for the 2-D Euler equations"; AIAA 23rd Aerospace Sciences Meeting, January 14-17, 1985, Reno, Nevada

Denton, J. D.; 1993

Dorney, D. J.; Griffin, L. W.; Sondak, D. L.; 2004

Doyle, M. D. C.; 1962
"Theories for Predicting Partial Admission Losses in Turbines"; Journal of Aerospace science; 29 (4); 1962

ANSYS FLUENT 12.0; Theory Guide
Fluent Inc., April 2009

ANSYS FLUENT 12.0; User's Guide
Fluent Inc., April 2009

Fridh J. E.; Bunkute B.; Fakhraï R.; and Fransson T. H.; 2004
"An experimental study on partial admission in a two-stage axial air test turbine with numerical comparisons"; Proceedings of the ASME Turbo Expo 2004, v 5 B: Turbomachinery; Axial Flow Turbine Aerodynamics; Design Methods and CFD Modelling for Turbomachinery; Turbomachines and the Environment, 2004, p 1285-1297
"Dynamic Features of Partial Admission: Outcome from Rotating Measurements"; 7th European Conference on Turbomachinery Fluid Dynamics and Thermodynamics (ETC 7); March 5-9, 2007, Athens, Greece

Gaugler, R. E.; Russel, L. M.; 1982
"Flow Visualization Study of the Horseshoe Vortex in a Turbine Stator Cascade"; NASA TP 1884 c.1, June 1982

Greitzer, E. M.; 2004

He, L.; 1997

Herzog, N.; Gündogdu, Y.; Kang, G.; Seume, J.R.; Rothe, K.; 2005

"Solution of the 2D Navier-Stokes Equations on Unstructured Adaptive Grids"; Presented at the AIAA 9th Computational Fluid Dynamics Conference

Horlock, J. H.; 1966

Kim, S. E. and Choudhury, D.; 1995
"A Near-Wall Treatment Using Wall Functions Sensitized to Pressure Gradient"; In ASME FED Vol. 217, Separated and Complex Flows

Klassen H. A., 1968
"Cold air investigation of effects of partial admission on performance of 3.75 inch axial flow turbine"; NASA TN D-4700

Lakshminarayana, B.; 1996

Lauder, B. E.; Spalding, D. B.; 1974

Lewis, K. L.; 1993
"The Influence of Partial Admission on the Performance of a Multistage Turbine"; Internal report; Whittle Laboratory, Cambridge University, UK

Macchi, E.; Lozaa, G.; 1985
"Comparison of Partial vs. Full Admission for Small Turbines at Low Specific Speeds"; American Society of Mechanical Engineers, 8p; ISSN: 0402-1215

Ohlsson, G. O.; 1962
"Partial-Admission Turbines"; Journal of the Aerospace Sciences, v 29, n 9, Sept. 1962, p 1017-1023 + 1028

Pfefferle, D.; 2004
"Investigation of Efficiency Measurements at HPT Test Turbine, Development of an alternative Evaluation Method"; Studienarbeit 2004; Dept of Energy Technology, Royal Institute of Technology, Stockholm, Sweden

Rauch, R. D.; Batira, J. T. and Yang, N. T. Y; 1991

Roelke, R. J.; 1973
"Turbine Design and Application"; NASA SP-290, A. J. Glassman, Vol. 2, Chap. 8

Singh, O.; 2006

Shaanan, S.; Ferziger, J., H.; Reynolds, W. C.; 1975
"Numerical simulations of Turbulence in the presence of shear"; Report TF-6, Department of Mechanical Engineering, Stanford University, Stanford, California

Skopek, J.; Vomela, J.; Taje, L.; Polansky, J.; 1999

Smagorinsky, J.; 1963

Stenning, A. H.; 1953
"Design of Turbines for High Energy Fuel Low Power Output Applications"; Massachusetts Institute of Technology - Dynamic Analysis Control Laboratory - Report, Sept 30, 1953, n 79, 30p

**Suter, P.; Traupel, W.; 1959**
"Untersuchungen ueber den Ventilationsverlust von Turbinenraedern (Investigation on windage loss of turbine rotors)"; Zurich Eidgenoessische Technische Hochschule -Institut fuer Thermische Turbomaschinen-Mitteilungen, n 4, 1959, 39p

**Södergård, B.; Henriksson, K.; Kjellsrtöm, B.; Söderberg, O.; 1989**
"Turbine testing facility"; Technical report, TRITA-KRV-1989-03; ISSN 1100-7990; Department of Thermal Engineering, The Royal Institute of Technology, S-100 44 Stockholm, Sweden

**Tajc, L.; Bednar, L.; Polansky, J.; Kovarik, P.; 2005**
"Experimental and computational investigation of flow phenomena in a steam turbine with radial control stage and two axial stages"; Proceedings of 5th European Turbomachinery Conference, Lille, France

**Tannehill, J. C.; Anderson, D. A.; Pletcher, R. H.; 1997**

**Traupel W., 1977**
"Thermische Turbomaschinen"

**Wakeley, G. R.; Potts I., 1997**
"Origins of loss within a multistage turbine environment under conditions of partial admission"; Paper No. 97-GT-96; Presented at the International Gas Turbine & Aeroengine Congress & Exhibition, 1997, Orlando, Florida, USA

**Yahya, S. M; 1968**
10. ACKNOWLEDGEMENT

I would like to thank my academic supervisor Professor Torsten H. Fransson at the Chair of Heat and Power Technology, the Royal Institute of Technology (KTH), Stockholm, for giving me the opportunity to start this work and for his continues support and encouragement during the entire study.

Dr. Jiasen Hu, my main supervisor during the first two years of this study and my Master thesis work is gratefully acknowledged.

I would like to thank my co-supervisor Mr. Jens Fridh for providing all the experimental measurement data and to help in physical interpretation of the computational results. Mr. Per Almqvist is also acknowledged for his useful suggestions and participation in the reference group meetings.

Great thanks to Mr. Rolf Wikström, Mr. Åke Göransson and Mr. Arne Karlsson from Steam Turbine Division of Siemens Industrial Turbomachinery AB, Finspång, Sweden for their useful and fruitful discussions at the reference group meetings and via emails.

The financial support of this work from Division of Heat and Power Technology at KTH, Stockholm and Steam Turbine Division of Siemens Industrial Turbomachinery AB, Finspång are truly acknowledged.

This work is supported by grants of computing time from centre of Parallel computers (PDC) at the Royal Institute of Technology (KTH), Stockholm which is distributed by the Swedish National Allocation Committee.

Thanks to all colleagues at the department of Energy Technology for a constructive and friendly working environment.

Finally, my greatest thanks are to my family for always being with me, although physically far away.
APPENDIX A  IMPORTANT GEOMETRICAL DIMENSIONS BASED ON ROTOR BLADE CHORD

Table A. 1 Geometrical Dimensions for 2D, Simple 3D and Full 3D models

| Cross Section 2 | 46% L_{axx} Upstream of S1 |
| Cross Section 3 | 16% L_{axx} Downstream of S1 |
| Cross Section 4 | 40% L_{axx} Downstream of R1 |
| Cross Section 5 | 6% L_{axx} Downstream of S2 |
| Cross Section 6 | 40% L_{axx} Downstream of R2 |
| Cross Section 7 | 172% L_{axx} Downstream of R2 |
| Clearance First stage stator and rotor disks | 16% L_{axx} |
| Clearance First stage rotor and second stage stator disks | 20% L_{axx} |
| Clearance Second stage stator and rotor disks | 16% L_{axx} |
| Clearance Second stage rotor disk and Exhaust Casing | 20% L_{axx} |
| Short Numerical Inlet (2D, Simple 3D, Full 3D with short inlet) | 100% L_{axx} Upstream of sec2 |
| Extended Numerical Inlet | 250% L_{axx} Upstream of S1, LE |
| Numerical Exit (2D & Simple 3D model) | At sec 7 |
| Numerical Exit (Full 3D model) | 243% L_{axx} Downstream of Sec 7 |
| Clearance first stage stator and rotor blade profiles | 40% L_{axx} |
| Clearance first stage rotor and second stage stator blade profiles | 76.7% L_{axx} |
| Clearance second stage stator and rotor blade profiles | 40% L_{axx} |

Table A. 2 Rotor Blade Profile Data

<table>
<thead>
<tr>
<th>Station as Percent of Chord</th>
<th>Upper Wall as Percent of Chord</th>
<th>Station as Percent of Chord</th>
<th>Lower Wall as Percent of Chord</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.08</td>
<td>49.53</td>
<td>0.54</td>
<td>42.51</td>
</tr>
<tr>
<td>0.44</td>
<td>42.92</td>
<td>1.74</td>
<td>39.96</td>
</tr>
<tr>
<td>14.99</td>
<td>82.92</td>
<td>21.97</td>
<td>46.45</td>
</tr>
<tr>
<td>48.16</td>
<td>94.65</td>
<td>57.97</td>
<td>46.51</td>
</tr>
<tr>
<td>75.26</td>
<td>70.51</td>
<td>86.10</td>
<td>23.91</td>
</tr>
<tr>
<td>89.74</td>
<td>36.38</td>
<td>99.20</td>
<td>0.07</td>
</tr>
<tr>
<td>99.98</td>
<td>0.68</td>
<td>100.00</td>
<td>0.56</td>
</tr>
</tbody>
</table>

Leading Edge Tangent Circle Radius / Chord = 0.16
Trailing Edge Tangent Circle Radius / Chord = 0.04
Maximum Thickness / Chord = 0.46
Chord Angle = 25°
Fig A. 1 Schematic picture of first stage rotor blade
APPENDIX B GOVERNING EQUATIONS

How well a numerical method can predict the behavior of a physical system depends on the physical and mathematical constrains imposed. The behavior of the fluid flow is governed with the fundamental equations of fluid dynamics. It is not possible to solve these equations in their exact form for the continuous domain, thus the physical domain is discretized to finite elements or volumes and the derivatives are approximated by differences.

Several considerations are required to ensure that the discretized equations predict the exact solution of the governing equations closely. In this chapter, the fundamental equations of fluid dynamics as used in the code are introduced first. Then the numerical methods to obtain the approximate solution of these equations are listed and the accuracy of numerical methods is assessed.

B. 1 Governing Equations in Their Exact Form

The equations governing the fluid dynamics and the heat transfer are the following conservation equations applied to fluid flow, Tannehill et al. [1997]:

- Conservation of mass (Continuity equation)
- Conservation of momentum (Newton’s second law)
- Conservation of energy (First law of thermodynamics)

In order to close the system of equations, it is needed to add an additional state equation which relates the pressure, density and temperature.

The conservation of mass applied to an infinitesimal control volume which is fixed in space and does not have any sources yields the following formula:

\[ \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{V}) = 0 \quad \text{(B-1)} \]

The conservation of momentum applied to the fluid passing an infinitesimal control volume that is fixed in space, yield the following general form of momentum equation:

\[ \frac{\partial}{\partial t} (\rho \vec{V}) + \nabla \cdot \rho \vec{V} \vec{V} = \rho \vec{f} + \nabla \vec{P} \quad \text{(B-2)} \]

The term \( \nabla \cdot \rho \vec{V} \vec{V} \) represents the rate of momentum lost by convection per unit volume from the control surfaces. This term can be expanded as below:

\[ \nabla \cdot \rho \vec{V} \vec{V} = \rho \vec{V} \cdot \nabla \vec{V} + \vec{V} (\nabla \cdot \rho \vec{V}) \quad \text{(B-3)} \]
By substituting the above expansion into equation (B-2) and using the continuity equation to simplify the terms, the following general form for momentum equation is obtained:

\[ \rho \frac{D\vec{V}}{Dt} = \rho \vec{f} + \nabla \cdot \bar{P} \]  

(B-4)

In the above equation, \( \vec{f} \) is the force per unit mass. \( \bar{P} \) represents the components of stress tensor and consist of normal and shear stresses. For a Newtonian fluid in which stress at a point is linearly dependent on the rate of strain of the fluid, \( \bar{P} \) can be written as below. It is assumed that the bulk viscosity is negligible here since this study does not deal with the structure of shock waves.

\[ \bar{P}_{ij} = -p \delta_{ij} + \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} \frac{\partial u_k}{\partial x_k} = -p \delta_{ij} + \bar{\tau}_{ij} \quad i,j,k = 1,2,3 \]  

(B-5)

where \( \delta_{ij} \) is the Kronecker delta function (\( \delta_{ij} = 1 \) if \( i = j \) and \( \delta_{ij} = 0 \) if \( i \neq j \)).

To obtain the energy equation, the First Law of Thermodynamics is applied to the fluid passing an infinitesimal control volume fixed in the space.

\[ \frac{\partial E_t}{\partial t} + \nabla \cdot \left( E_t \vec{V} \right) = \frac{\partial Q}{\partial t} + \nabla \cdot \left( k_{eff} \nabla T \right) + \rho \vec{f} \cdot \vec{V} + \nabla \cdot \left( \bar{P} \vec{V} \right) \]  

(B-6)

where \( E_t \) is the total energy per unit volume:

\[ E_t = \rho (e + \frac{v^2}{2} + \text{potential energy} + \cdots) \]  

(B-7)

and \( k_{eff} \) is the effective conductivity (\( k_t + k \) where \( k_t \) is the turbulent thermal conductivity, defined according to the turbulence model being used), ANSYS FLUENT Theory Guide [2009].

**B. 2 Equation of State**

In order to close the system of governing equations and to make a relationship between the thermodynamic properties and the transport properties, the equations of state are required. From thermodynamics, we know that the state of a fluid is known if two thermodynamic properties are known.

Air is assumed to obey the perfect gas law in this study. The equation of state for perfect gas is as below:

\[ p = \rho RT \]  

(B-8)
where $R$ is the gas constant and for air at standard conditions, $R = 287 \, m^2/(s^2 K)$.

Sutherland’s law is used to relate the coefficients of viscosity and thermal conductivity to the thermodynamic variables.

$$
\mu = \mu_0 \left( \frac{T}{T_0} \right)^{3/2} \frac{T_0 + S}{T + S}
$$

(B-9)

The constants for air were considered as below:

- $\mu_0 = \text{Reference viscosity} = 1.716 \times 10^{-5} \, kg/m.s$
- $T_0 = \text{Reference temperature} = 273.11 \, K$
- $S = \text{Effective temperature} = 110.56 \, K$

Once the viscosity is known, thermal conductivity of the fluid can be obtained. It is possible to assume the thermal conductivity constant. For air, the constant thermal conductivity is assumed to be $0.0242 \, W/m.K$.

However if the gas law is used, the thermal conductivity can be defined using the kinetic theory, ANSYS FLUENT Theory guide [2009]

$$
k = \frac{15}{4} \frac{R}{M_w} \mu \left[ \frac{4}{15} \frac{c_p M_w}{R} + \frac{1}{3} \right]
$$

(B-10)

where $M_w$ is the molecular weight and the specific heat capacity for the fluid air is assumed to be constant $c_p = 1006.43 \, J/kg.K$ which is a good approximation for air at moderate pressures and temperatures.

**B. 3 Vector Form of Governing Equations**

The system of governing equations is combined into vector form before applying a numerical algorithm. The compressible governing equations in Cartesian coordinates without body forces, mass diffusion, chemical reactions or external heat sources can be written as below, Tannehill; et al. [1997].

$$
\frac{\partial \vec{U}}{\partial t} + \frac{\partial \vec{E}}{\partial x} + \frac{\partial \vec{F}}{\partial y} + \frac{\partial \vec{G}}{\partial z} = 0
$$

(B-11)

where $\vec{U}, \vec{E}, \vec{F}$ and $\vec{G}$ are vectors given by
\[
\overline{U} = \begin{bmatrix}
\rho

\rho u

\rho v

\rho w

E_t
\end{bmatrix}
\]

\[
\vec{E} = \begin{bmatrix}
\rho u

\rho u^2 + p - \tau_{xx}

\rho uv - \tau_{xy}

\rho uw - \tau_{xz}

(E_t + p)u - u\tau_{xx} - v\tau_{xy} - w\tau_{xz} + q_x.
\end{bmatrix}
\]

\[
\vec{F} = \begin{bmatrix}
\rho v

\rho v^2 + p - \tau_{xy}

\rho vw - \tau_{yz}

(E_t + p)v - u\tau_{xy} - v\tau_{yy} - w\tau_{yz} + q_y.
\end{bmatrix}
\]

\[
\vec{G} = \begin{bmatrix}
\rho w

\rho uw - \tau_{xz}

\rho vw - \tau_{yz}

\rho w^2 + p - \tau_{zz}

(E_t + p)w - u\tau_{xz} - v\tau_{yz} - w\tau_{zz} + q_z.
\end{bmatrix}
\]

The first row is the continuity equation while the next three rows are the momentum equations in x, y and z direction and the last row is the energy equation.

### B. 4 Reynolds Averaging Method for Governing Equations

When the Reynolds number of the flow is increased or there are external tabulators, the random motions in the flow increases. These random motions called “eddies” are of varying length and time scales. Currently it is not possible with the available computer resources to model the whole range of the turbulent eddies by Direct Numerical Simulation (DNS) else than for simple flows.

One way to reduce the computational cost is to filter the governing equations to contain the large eddies (greater than the size of the filter) and to model the smaller eddies. This method is called the Large Eddy Simulation (LES) and the associated computational resources required are much less than the DNS model for the same problem.

Another common way (especially in industrial applications) is to time-average or mass-average the governing equations. The equations obtained in this manner are called the Reynolds averaged Navier-Stokes (RANS) equations. However by averaging the equations of motion additional terms associated with the turbulence are raised in the equations. These terms must be related to the mean flow variables by turbulence modeling and additional assumptions should be made in order to close the problem.
The instantaneous variables in the governing equations are decomposed into mean and fluctuating terms in the RANS method. In the case of compressible flows the mass-weighted averaging is more convenient than the simple time-averaging. The mass-averaged variables are defined according to $\bar{f} = \rho \bar{f}/\bar{\rho}$, Tannehill; et al. [1997]. The mass-averaged velocity components are defined in the form below

$$u = \bar{u} + u'' \quad v = \bar{v} + v'' \quad w = \bar{w} + w'' \quad (B-13)$$

However the fluid properties such as density and pressure are time-averaged according to

$$\bar{\rho} = \bar{\rho}' + \bar{\rho}'', \quad \bar{p} = \bar{p}' + \bar{p}'' \quad (B-14)$$

The above variables are substituted into the instantaneous Navier-Stokes equations and are time averaged. The mass-weighted average of the continuity equations is obtained as below

$$\frac{\partial \bar{\rho}}{\partial t} + \frac{\partial}{\partial x_j} (\bar{\rho} \bar{u}_j) = 0 \quad (B-15)$$

And the complete Reynolds momentum equation in mass-weighted variables becomes

$$\frac{\partial}{\partial t} (\bar{\rho} \bar{u}_i) + \frac{\partial}{\partial x_j} (\bar{\rho} \bar{u}_i \bar{u}_j) = \frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j} (\bar{\tau}_{ij} - \rho \bar{u}_i \bar{u}_j) \quad (B-16)$$

If the viscosity fluctuations are neglected, $\bar{\tau}_{ij}$ becomes

$$\bar{\tau}_{ij} = \mu \left[ \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} \frac{\partial \bar{u}_k}{\partial x_k} \right] + \rho u_i u_j + \mu \left[ \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} \frac{\partial \bar{u}_k}{\partial x_k} \right] \quad (B-17)$$

The stress gradients can be decomposed into laminar-like $\bar{\tau}_{ij}\text{_{lam}}$ and turbulent $\bar{\tau}_{ij}\text{_{turb}}$ terms as below

$$\bar{\tau}_{ij}\text{_{lam}} = \mu \left[ \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} \frac{\partial \bar{u}_k}{\partial x_k} \right] \quad (B-18)$$

$$\bar{\tau}_{ij}\text{_{turb}} = \rho u_i u_j + \mu \left[ \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} \frac{\partial \bar{u}_k}{\partial x_k} \right] \quad (B-19)$$

In practical applications, the viscous terms with the double prime fluctuations are small and are neglected. In FLUENT, the second term in Eq. (5-18) involving the molecular
viscosity is expected to be much smaller than $\rho \overline{u_i u_j}$ and is neglected, ANSYS FLUENT Theory guide [2009].

By Reynolds-averaging method, new unknowns are added into the governing equations. The Reynolds equations cannot be solved in the form given and additional equations are needed involving the new unknowns to close the problem or some relations should be developed between the new unknown turbulent fluctuations and the mean flow motion. This is known as turbulence modeling.

### B. 5 Large-Eddy Simulation (LES)

In the Large Eddy Simulation model, the large eddies are resolved directly since they are more geometry dependent and the small eddies are modeled. The small eddies have less dependency on the geometry and they are more isotropic thus it is easier to find a universal model to describe them.

The general time-dependent Navier-Stokes equations can be filtered in either Fourier (wave-number) space or configuration (physical) space to obtain a set of equations. However the volume-averaged filter is more frequently used with finite volume method. Flow variables can be decomposed into large (resolved) and small (subgrid) scales, as follows

$$u_i = \overline{u_i} + \hat{u}_i \quad (B-20)$$

where $\overline{u_i}$ is the resolvable part and $\hat{u}_i$ is the subgrid part. The filtered variable is defined with the following integral

$$\overline{u_i} = \int_D u_i(\hat{x}) G(x, \hat{x}) d\hat{x} \quad (B-21)$$

Where $D$ is the fluid domain, $G$ is the filter function that determines the scale of the resolved eddies and $x, \hat{x}$ are the position vectors. In FLUENT, the finite-volume discretization itself implicitly provides the filtering operation, ANSYS FLUENT Theory guide [2009].

$$\overline{u_i}(x) = \frac{1}{V} \int_V u_i(\hat{x}) d\hat{x}, \quad \hat{x} \in V \quad (B-22)$$

Where $V$ is the volume of the computational cell and $G$ is defined as follows

$$G(x, \hat{x}) = \begin{cases} 1 & \hat{x} \in V \\ 0 & \text{otherwise} \end{cases} \quad (B-23)$$

In order to keep the equations concise, the filtered equations are developed for the incompressible Navier-Stokes equations of motion.
\[
\frac{\partial \bar{u}_i}{\partial x_j} = 0 \tag{B-24}
\]

\[
\frac{\partial \bar{u}_i}{\partial t} + \frac{\partial \bar{u}_i \bar{u}_j}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \bar{p}}{\partial x_i} + v \frac{\partial^2 \bar{u}_i}{\partial x_k \partial x_k} \tag{B-25}
\]

It is not possible to solve the above equations for both \( \bar{u}_i \) and \( \bar{u}_i \bar{u}_j \) so the convective flux is represented in terms of decomposed variables

\[
\bar{u}_i \bar{u}_j = \bar{u}_i \bar{u}_j + \tau_{ij} \tag{B-26}
\]

resulting

\[
\frac{\partial \bar{u}_i}{\partial t} + \frac{\partial \bar{u}_i \bar{u}_j}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \bar{p}}{\partial x_i} + v \frac{\partial^2 \bar{u}_i}{\partial x_k \partial x_k} - \frac{\partial \tau_{ij}}{\partial x_j} \tag{B-27}
\]

\( \tau_{ij} \) is the subgrid scale stress tensor and can be developed as follows

\[
\tau_{ij} = \bar{u}_i \bar{u}_j - \bar{u}_i \bar{u}_j = (\bar{u}_i + \bar{u}_i)(\bar{u}_j + \bar{u}_j) - \bar{u}_i \bar{u}_j = (\bar{u}_i \bar{u}_j - \bar{u}_i \bar{u}_j) + (\bar{u}_i \bar{u}_j + \bar{u}_i \bar{u}_j) + (\bar{u}_i \bar{u}_j) \tag{B-28}
\]

The first term in the right hand side inside the parenthesis is called Leonard stress, the second term, the cross-term stress and the third term, the Reynolds stress, Tannehill et al. [1997]. If averaging was used instead of filtering the first two terms would be zero and only the Reynolds stress term was modeled. Neglecting the Leonard stresses can be shown to be of the same order as the truncation error for second-order schemes, Shaanan et al., [1975].

The time-averaged fluctuations can be computed from the difference between the data obtained from LES and RAN calculations. That is \( \bar{u}_i = \bar{u}_i - \langle \bar{u}_i \rangle \) where the double prime is a time dependent fluctuation and the angle bracket show a time averaged variable. It should be noted that only the large eddies (larger than the filter size) are simulated with the LES method.

### B.6 Turbulence Modeling

Numerical simulation requires that the turbulent stress terms in the Reynolds equations are closed. This process is called turbulence modeling. It is not possible currently to find a universal turbulent model that can work well for all types of flows. Therefore the task is to find a turbulent model that has acceptable accuracy and reasonable amount of computational time for a certain type of flow.
Boussinesq Hypothesis

According to Boussinesq’s hypothesis, the turbulent shear stresses are related to the mean flow strain rate. By neglecting the bars over the mean flow variable, the Boussinesq hypothesis for the Reynolds stress is

$$-\rho u'_i u'_j = \mu_t \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \left( \rho k + \mu_t \frac{\partial u_k}{\partial x_k} \right)$$

(B-29)

where $\mu_t$ is the turbulent viscosity and $k$ is the kinetic energy of turbulence $k = \frac{u'_i u'_i}{2}$.

By kinetic theory analogy for gases, the turbulent viscosity can be modeled as

$$\mu_t = \rho v_t l$$

(B-30)

where $u_t$ and $l$ are characteristic velocity and length scales of the turbulence respectively. The RNG, $k - \epsilon$ turbulence model and subgrid turbulence modeling in LES, both use the Boussinesq’s hypothesis.

RNG, $k - \epsilon$ Model

In two equations turbulence models, two transport equations are solved and the turbulence velocity and length scales are determined. The two transport quantities in the $k - \epsilon$ model are the kinetic energy of turbulence $k$ and the dissipation rate $\epsilon$. The turbulent viscosity can be evaluated from the solution as $\mu_t = C_\mu \rho k^2/\epsilon$ and the length scale $l = C^{3/4}_\mu k^{3/2}/\epsilon$. The transport equations for the RNG, $k - \epsilon$ model is as follows, ANSYS FLUENT Theory guide [2009].

$$\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_j} \left( \alpha_k \mu_{eff} \frac{\partial k}{\partial x_j} \right) + G_k + G_b - \rho \epsilon - Y_M$$

(B-31)

$$\frac{\partial}{\partial t} (\rho \epsilon) + \frac{\partial}{\partial x_i} (\rho \epsilon u_i) = \frac{\partial}{\partial x_j} \left( \alpha_\epsilon \mu_{eff} \frac{\partial \epsilon}{\partial x_j} \right) + C_1 \epsilon \left( G_k + C_3 G_b \right) - C_2 \rho \frac{\epsilon^2}{k} - R_\epsilon$$

(B-32)

The model constants are derived analytically by the RNG theory $C_{1\epsilon} = 1.42$, $C_{2\epsilon} = 1.68$. $G_k$ represents the production of turbulence kinetic energy,

$$G_k = \mu_t \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) = 2\mu_t S_{ij}$$

(B-33)

$S_{ij}$ is the rate of mean strain tensor given by
\[ S_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad (B-34) \]

For the near-wall and low-Reynolds-number regions, turbulent viscosity is determined by scale elimination procedure through a differential equation

\[ d \left( \frac{\rho^2 k}{\sqrt{\epsilon \mu}} \right) = 1.72 \frac{\hat{\nu}}{\sqrt{\hat{\nu}^3 - 1 + C_v}} d \hat{\nu} \quad (B-35) \]

where \( \hat{\nu} = \frac{\mu_{eff}}{\mu} \) and \( C_v \approx 100 \). By integrating Eq. (B-34), the rate of change of the effective turbulent transport versus the effective Reynolds number is obtained. In the high Reynolds number limit, Eq. (B-34) becomes

\[ \mu_t = \rho c_\mu \frac{k^2}{\epsilon} \quad (B-36) \]

where \( c_\mu = 0.0845 \) using the RNG theory. The turbulent viscosity is modified in the RNG method if the main flow is swirl dominant. The modification takes the form

\[ \mu_t = \mu_{t0} f \left( \alpha_s, \Omega, \frac{k}{\epsilon} \right) \quad (B-37) \]

where \( \mu_{t0} \) is the value of turbulent viscosity calculated by either Eq. (B-34) or (B-35). \( \Omega \) is the characteristic swirl number evaluated within FLUENT and \( \alpha_s \) is the swirl constant that varies depending on if the flow is mildly swirling or is swirl dominated.

The inverse effective Prandtl numbers, is calculated using the formula below which is derived analytically from the RNG theory

\[ \frac{\alpha - 1.3929}{\alpha_0 - 1.3929} \cdot 0.6321 \cdot \frac{\alpha + 2.3929}{\alpha_0 + 2.3929} \cdot 0.3679 = \frac{\mu_{mol}}{\mu_{eff}} \quad (B-38) \]

Where \( \alpha_0 = k/\mu_c \) and in the high Reynolds number limit \( (\mu_{mol}/\mu_{eff} \ll 1) \), \( \alpha_k = \alpha_e \approx 1.393 \). The \( R_\epsilon \) term used in the transport equation for \( \epsilon \) is given as below

\[ R_\epsilon = \frac{C_\mu \rho \eta^3 (1 - \eta/\eta_0) \epsilon^2}{1 + \beta \eta^3} k \quad (B-39) \]

where \( \eta = S k/\epsilon, \eta_0 = 4.38, \beta = 0.012 \).

The buoyancy effect in the production of \( k \) and \( \epsilon \) terms are accounted if the gravity and temperature gradients are present in the flow. The generation of turbulence due to buoyancy is calculated by
\[ G_b = \beta g_i \frac{\mu_t}{Pr_t} \frac{\partial T}{\partial x_i} \]  

(B-40)

where \( Pr_t \) is the turbulent Prandtl number, \( Pr_t = 1/\alpha \) where \( \alpha \) is calculated using Eq. (B-37) and \( g_i \) is the gravity vector in the direction of \( i \). The coefficient of thermal expansion is calculated as below

\[ \beta = -\frac{1}{\rho} \left( \frac{\partial \rho}{\partial T} \right)_p \]  

(B-41)

Whereas for the ideal gases, Eq. (B-40) reduces to

\[ G_b = -g_i \frac{\mu_t}{\rho Pr_t} \frac{\partial \rho}{\partial x_i} \]  

(B-42)

\( C_{3\epsilon} \) determines how the transport quantity \( \epsilon \) is affected by the buoyancy effects. \( C_{3\epsilon} \) is calculated in FLUENT as below

\[ C_{3\epsilon} = \tanh \left| \frac{\nu}{u} \right| \]  

(B-43)

where \( \nu \) is the velocity vector parallel to the gravity vector and \( u \) is the component of the flow velocity in the direction perpendicular to the gravity vector.

The compressibility effects are accounted in the turbulence through a dilatation dissipation term, \( Y_M \)

\[ Y_M = 2 \rho \epsilon M_t^2 \]  

(B-44)

where \( M_t \) is the turbulent Mach number, defined as \( M_t = \sqrt{k/\alpha^2} \) and \( \alpha (\equiv \sqrt{\gamma RT}) \) is the speed of sound.

The convective heat transfer is modeled in FLUENT using the Reynolds analogy to momentum transfer, ANSYS FLUENT Theory guide [2009]. The modeled energy equation can be written as

\[ \frac{\partial}{\partial t}(\rho E) + \frac{\partial}{\partial x_i} \left[ u_i (\rho E + p) \right] = \frac{\partial}{\partial x_j} \left( k_{eff} \frac{\partial T}{\partial x_j} + u_i (\tilde{\tau}_{ij})_{eff} \right) \]  

(B-45)

where \( E \) is the total energy, \( k_{eff} \) is the effective thermal conductivity and \( (\tilde{\tau}_{ij})_{eff} \) is the viscous stress tensor defined as

\[ (\tau_{ij})_{eff} = \mu_{eff} \left( \frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right) - \frac{2}{3} \mu_{eff} \frac{\partial u_k}{\partial x_k} \delta_{ij} \]  

(B-46)
Subgrid Scale Models for LES

The subgrid scale stresses in the Large Eddy Simulation should be modeled. Since the subgrid scale stresses are small, a relatively simple model could be used. The first model was proposed by Smagorinsky [1963]. It takes the form of a mixing length model. The SGS stress tensor is represented by

\[ \tau_{ij} = 2 \mu_t S_{ij} \]  \hspace{2cm} (B-47)

where \( S_{ij} \) is the rate of stress tensor and

\[ \mu_t = \rho C_s^2 \sqrt{2 S_{ij} S_{ij}} \]  \hspace{2cm} (B-48)

Various values of the Smagorinsky constant \( C_s \) is reported between 0.1 to 0.24. A more complex model for LES is the dynamic Smagorinsky model. In the basic form, the dynamic Smagorinsky model takes the same form as the simple Smagorinsky model but allows the \( C_s \) constant to be calculated from two filter sizes. When this model is applied to the flows involving heat transfer, the turbulent Prandtl number is calculated as part of the solution.

B. 7 Near-Wall Flow Modelling

Flow close to the walls is affected with the presence of walls. The non-slip condition is assumed on the walls; therefore the flow velocity is zero. The velocity gradients close to the walls is large and the variation of velocity from zero to the value in fully turbulent region need to be modeled. There are two types of turbulence models based on their ability to resolve the viscosity affected near-wall region. The high-Reynolds number models can only predict the flow far from the walls. When using this type of models, appropriate type of near-wall modeling should be used in order to link the viscosity affected near-wall flow with the outer turbulent flow. The other type of turbulence models, called the low-Reynolds number models has functions that resolve the flow all the way down to the walls. In order to be able to resolve the viscosity affected near wall regions sufficient mesh density needs to be used close to the walls.

Experiments have shown that the area between the walls and the outer turbulent flow can be divided into three layers. The layer closest to the wall is called the viscous sublayer, viscous effects are significant in this layer and the flow can be considered laminar. The upper layer is called the buffer layer, the viscosity and turbulence effects are both present on this layer. The upper layer is called the fully turbulent layer in which the turbulent effects are dominating the viscous effects.

Figure B-1, shows the regions of a turbulent boundary layer for a flat plate with incompressible flow. Non-dimensional velocity \( u^+ \) is plotted against the non-dimensional
distance from the wall $y^+$. The diagram has logarithmic scale and the shear velocity is defined as below:

$$u_\tau = (\frac{\tau_w}{\rho_w})^{1/2}$$  \hspace{1cm} (B-49)

Near-wall models are based on empirical results. They are used to connect the flow variables between the wall and its adjacent cells. In this work, the non-equilibrium wall function is used together with the $RNG$, $k-\varepsilon$ turbulence model. Kim and Choudhury; 1995 showed that the above combination gives the best result for prediction of reattachment length, skin friction and static pressure coefficient in a backward-facing step flow.

**Non-Equilibrium Wall Function in RNG, $k-\varepsilon$ Model**

The non-equilibrium wall function is used together with the RNG turbulence model for most of the computations of this work. Non-equilibrium near-wall function is a two layer model based on a proposal from Kim and Choudhury [1995] in which the standard near-wall model, Launder and Spalding [1974], is further optimized to include the pressure gradient effects. In this model the log law for mean velocity is as follows according to ANSYS FLUENT Theory guide [2009]

---

**Fig B. 1 Zones in the turbulent boundary layer for a typical incompressible flow over a smooth flat plate, Greitzer; et al. [2004] after Cebeci and Bradshaw [1977]**
\[
\frac{\bar{U} C_{\mu}^{1/4} k^{1/2}}{\tau_w / \rho} = \frac{1}{\kappa} \ln \left( E \frac{\rho C_{\mu}^{1/4} k^{1/2} y}{\mu} \right)
\]  \hspace{1cm} (B-50)

where

\[
\bar{U} = U - \frac{1}{2 \bar{d} x} \frac{d p}{\rho k} \ln \left( \frac{y}{y_v} \right) + \frac{y - y_v}{\rho k} + \frac{y^2_v}{\mu} \]  \hspace{1cm} (B-51)

\( y_v \) is the physical viscous sublayer thickness

\[
y_v \equiv \frac{\mu y_v^2}{\rho C_{\mu}^{1/4} k_p^{1/2}}
\]  \hspace{1cm} (B-52)

\[
y^* \equiv \frac{\rho C_{\mu}^{1/4} k_p^{1/2} y_p}{\mu}
\]  \hspace{1cm} (B-53)

where

\( y_v^* \) = 11.225 in Eq. (B-52)
\( \kappa \) = von Karman Constanta (= 0.4187)
\( E \) = empirical constant (= 9.793)
\( U_p \) = mean velocity of the fluid at point P
\( k_p \) = turbulence kinetic energy at point P
\( y_p \) = distance from point P to the wall
\( \mu \) = dynamic viscosity of the fluid

The non-equilibrium wall function assumes that the flow close to the walls consist of two layers, the viscous sublayer and the fully turbulent layer. The profile assumptions for turbulent quantities are as follows:

\[
\tau_t = \begin{cases} 0, & y < y_v \\ \tau_w, & y > y_v \\ \end{cases}
\]

\[
k = \begin{cases} \left( \frac{y}{y_v} \right)^2 k_p, & y < y_v \\ k_p, & y > y_v \\ \end{cases}
\]

\[
\epsilon = \begin{cases} \frac{2 \nu k}{y^2}, & y < y_v \\ \frac{k^{3/2}}{C_i y^3}, & y > y_v \\ \end{cases}
\]  \hspace{1cm} (B-54)

where \( C_i = \kappa C_{\mu}^{-3/4} \). Using the profiles defined above, the cell averaged production of \( k \), \( \bar{G}_k \) and the cell-averaged dissipation rate \( \bar{\epsilon} \) can be estimated with the volume averages of the cells. For hexahedral cells the volume averages are approximated by depth averages as below:

\[
\bar{G}_k = \frac{1}{y_n} \int_0^{y_n} \tau_t \frac{\partial U}{\partial y} dy = \frac{1}{\kappa y_n \rho C_{\mu}^{1/4} k_p^{1/2}} \ln \left( \frac{y_n}{y_v} \right)
\]  \hspace{1cm} (B-55)

and
\[
\bar{e} \equiv \frac{1}{y_n} \int_0^{y_n} \varepsilon \, dy = \frac{1}{y_n} \left[ 2\nu + k_p^{1/2} \ln \left( \frac{y_n}{y^*} \right) \right] k_p
\]  \hfill (B-56)

where \( y_n \) is the height of the cell \( (y_n = 2y_p) \).

**Near Wall Modeling in LES**

If the near wall mesh is fine enough to resolve the laminar sublayer, the near wall shear stress is obtained from laminar stress-strain relation as

\[
\bar{\tau} = \frac{\rho u_t y}{\mu}
\]  \hfill (B-57)

If the centroid of the cell adjacent to the wall, falls within the logarithmic region of the boundary layer, the law of the wall is employed

\[
\frac{\bar{u}}{u_t} = \frac{1}{\kappa} \ln \left( \frac{\rho u_t y}{\mu} \right)
\]  \hfill (B-58)

where \( \kappa \) is the von Karman constant and \( \kappa = 0.41 \). If the centroid of the cell adjacent to the wall falls in the buffer layer, the laminar and turbulent laws of the wall are blended as follows

\[
u^+ = e^{\gamma} u^+_{lam} + e^{1 - \gamma} u^+_{turb}
\]  \hfill (B-59)

and the blending function is given as

\[
\Gamma = -\frac{0.01(y^+)}{1 + 5y^+}
\]  \hfill (B-60)

**B. 8 Boundary Conditions**

**Pressure Inlet**

The boundary condition at the inlet of numerical domain is taken as the known pressure condition. The flow rate and the velocities are to be determined from the calculations. The pressure values are obtained from experimental measurements as described in chapter 4.

At the pressure inlet the following information is specified, ANSYS FLUENT Theory guide [2009]:

• Total pressure
• Total temperature
• Flow direction
• Static pressure
• Turbulent parameters

When the inlet flow is subsonic, the static pressure value is ignored. The isentropic relations for ideal gas are defined as below for the inlet conditions

\[
P_0 = p_s (1 + \frac{\gamma - 1}{2} M^2)^{(\gamma-1)}
\]

(B-61)

\[
M \equiv \frac{V}{a} = \frac{V}{\gamma R T_s}
\]

(B-62)

\[
\rho = \frac{p_s}{R T_s}
\]

(B-63)

\[
\frac{T_0}{T_s} = 1 + \frac{\gamma - 1}{2} M^2
\]

(B-64)

**Specification of Turbulent Quantities at Inlet and Outlet**

The turbulent quantities were defined in terms of intensity and viscosity ratio at the inlet and outlet of numerical domain. "The turbulence intensity, \(I\), is defined as the ratio of the root-mean-square of the velocity fluctuations, \(u'\), to the mean flow velocity, \(u_{avg}\). The turbulence intensity at the core of a fully-developed duct flow can be estimated from the following formula derived from an empirical correlation for pipe flows", ANSYS FLUENT Theory guide [2009]

\[
I = \frac{\hat{u}}{u_{avg}} = 0.16 (Re_D)^{-1/8}
\]

(B-65)

And the turbulent viscosity ratio, \(\mu_t / \mu\), is directly proportional to the turbulent Reynolds number \((Re_t = k^2 / (\nu \sigma))\). The turbulent intensity was estimated from the experiments to have a uniform value of 4% at the inlet. At outlet the turbulent intensity is assumed to have uniform value of 2% and the viscosity ratio is 5.

**Pressure Outlet**

Static pressure is specified at the pressure outlet boundary condition. The static pressure value is used only when the flow is subsonic if the flow becomes supersonic locally, the pressure will be extrapolated from the interior and the specified value will not be used. The following information is entered at an outlet boundary condition, ANSYS FLUENT Theory guide [2009]
• Static pressure
• Backflow conditions (total temperature, backflow direction, ...)

Only the specified static pressure is used at the outlet and all the other conditions are extrapolated from the interior of the domain.

Wall and Symmetry Boundaries

The non-slip boundary condition is enforced at the wall bounded viscous flows. This means that the adjacent fluid sticks to the wall and moves with the same speed of wall if the wall is moving.

Symmetry boundary conditions are used when the physical geometry of interest, and the expected pattern of the flow solution, has mirror symmetry. They can also be used to model zero-shear slip walls in viscous flows. A zero flux of all quantities across a symmetry boundary is assumed, ANSYS FLUENT Theory guide [2009].

Sliding Interfaces

Sliding mesh is employed for the time-dependent solution of rotor-stator interaction. The cell zones slide (rotate) relative to one another along the grid interface in each time step. For the axial rotor-stator configuration in which the rotating and stationary parts are aligned axially, the interface is a planar section perpendicular to the axis of rotation as shown in Fig (B-4).

![Fig B. 2 Rotor-stator interaction, stationary guide vanes with rotating blades](FLUENT User’s Guide, 2006)
APPENDIX C  TRUNCATION ERRORS

In order to change the derivatives in the PDE's to finite differences, the Taylor's series expansion can be used, Tannehill et al.; [1997]. If \( f \) is a continuous function in the Cartesian coordinate, the Taylor's series expansion for \( f(x_0 + \Delta x, y_0, z_0) \) about the point \((x_0, y_0, z_0)\) gives

\[
\begin{align*}
  f(x_0 + \Delta x, y_0, z_0) &= f(x_0, y_0, z_0) + \frac{\partial f}{\partial x} \Delta x + \frac{\partial^2 f}{\partial x^2} \left( \frac{\Delta x^2}{2!} ight) + \ldots \\
  &+ \frac{\partial^{n-1} f}{\partial x^{n-1}} \left( \frac{\Delta x^{n-1}}{(n-1)!} \right) + \frac{\partial^n f}{\partial x^n} \left( \frac{\Delta x^n}{n!} \right)
\end{align*}
\]

(C-1)

The forward difference can be obtained by re-arranging Eq. (C-1) as follows

\[
\frac{\partial f}{\partial x} \bigg|_{x_0, y_0, z_0} = \frac{f(x_0 + \Delta x, y_0, z_0) - f(x_0, y_0, z_0)}{\Delta x} - \frac{\partial^2 f}{\partial x^2} \left( \frac{\Delta x^2}{2!} \right) - \ldots
\]

(C-2)

If the notation is changed to \( i, j, k \)

\[
\frac{\partial f}{\partial x} \bigg|_{i,j,k} = \frac{f_{i+1,j,k} - f_{i,j,k}}{\Delta x} + T.E.
\]

(C-3)

where \( \frac{f_{i+1,j,k} - f_{i,j,k}}{\Delta x} \) is the finite difference representation for \( \frac{\partial f}{\partial x} \bigg|_{i,j,k} \) and the truncation error (T.E.) is the difference between the partial derivative and its finite-difference representation. Here the truncation error is of the order of \( \Delta x \). This is written in the mathematical form \( O(\Delta x) \) which means \(|T.E.| < K|\Delta x|\) when \( \Delta x \to 0 \) and \( K \) is a positive, real constant and we can write

\[
\frac{\partial f}{\partial x} \bigg|_{i,j,k} = \frac{f_{i+1,j,k} - f_{i,j,k}}{\Delta x} + O(\Delta x)
\]

(C-4)

An approximation to the second order derivative is obtained as follows

\[
\frac{\partial^2 f}{\partial x^2} \bigg|_{i,j,k} = \frac{f_{i+1,j,k} - 2f_{i,j,k} + f_{i-1,j,k}}{(\Delta x)^2} + O[(\Delta x)^2]
\]

(C-5)

So we may expect that the T. E. of the second representation is smaller than the first for sufficiently small \( \Delta x \).
APPENDIX D  CONTOURS OF NORMALIZED STATIC PRESSURE WITH DIFFERENT INLET EXTENSIONS

Fig D. 1 Contours of Static Pressure normalized with Inlet Total Pressure at the Entering End of the Blockage, extended grid (lower) and shorter grid (upper)
Fig D. 2 Contours of Static Pressure Normalized with Inlet Total Pressure, at the Leaving End of the Blockage, for Extended Grid (Lower) and Shorter Grid (Upper)
Fig D. 3 Contours of Static Pressure Normalized with Inlet Total Pressure, at the Admission Channel, for Extended Grid (Lower) and Shorter Grid (Upper)