A DESIGN STRATEGY FOR A 6:1 SUPersonic Mixed-Flow COMPRESSOR STAGE AND ITS VIScOUS FLOW-BASED PERFORMANCE ANALYSIS

A Thesis in
Aerospace Engineering
by
Aravinth Sadagopan

© 2019 Aravinth Sadagopan

Submitted in Partial Fulfillment of the Requirements for the Degree of

Master of Science

May 2019
The thesis of Aravinth Sadagopan was reviewed and approved* by the following:

Cengiz Camci  
Professor of Aerospace Engineering  
Thesis Advisor

Dennis K. McLaughlin  
Professor Emeritus of Aerospace Engineering

Amy R. Pritchett  
Professor and Head of the Department, Aerospace Engineering

*Signatures are on file in the Graduate School
ABSTRACT

A current surge in the small jet engine market requires compact and robust high-performance compressors. This thesis presents the design of a single-stage high-pressure ratio mixed-flow compressor with a prescribed maximum diameter in the (1-10) kg.s⁻¹ mass flow segment. Its compactness and reliability demonstrate its suitability in replacing a multistage axial compressor design in the small aero-engine segment with a high-performance envelope.

A comprehensive background of mixed-flow stage design is provided based on historical developments since the 1940s. A high-pressure ratio demand necessitates supersonic rotor exit flow. The high-pressure ratio and small diameter requirements push the compressor toward a "highly-loaded" supersonic “shock-in rotor” design with a supersonic stator/diffuser. Hence, tandem stator configurations with two blade rows were investigated in the past to reduce blade loading for efficient diffusion. Even so, most of the previous stage designs were inefficient, due to the inability of stators to efficiently diffuse supersonic flow. Thus, this thesis implements a tandem design based on Quishi et al. [16].

A unique mean-line design procedure is presented based on the isentropic equations defined for a mixed-flow stage. Mass flow rate, stage total pressure ratio, and maximum diameter were chosen as the main design constraints, and a geometry construction technique based on Bezier curves was used. The advancement of multi-dimensional and viscous computational tools has improved accessibility to and reduced overall effort in the thorough analysis of complex turbomachinery designs. Therefore, the aim of this thesis is to include all three dimensionality effects of the stage, viscous flow, and compressibility including the shock wave systems.

The computational model employed was thoroughly assessed for its ability to predict compressor performance, as compared to existing well-established experimental data. The results from a RANS-based computational fluid dynamics model were compared to the experimental
results of NASA Rotor 37 [20] and the RWTH Aachen [10] supersonic tandem diffuser. The computational approach shed light upon the mixed-rotor and supersonic-stator 3D shock structures, as well as the viscous/secondary flow. Furthermore, a rotor design evaluation study was conducted for a 3.5 kg/s mass flow based on the current mean-line code and additional computational analysis. A relatively high single-stage pressure ratio of 6.0 was targeted.

The performance map for the mixed-flow stage was obtained to better understanding the viscous flow details and shock systems of this high-pressure ratio mixed-flow compressor. Areas of potential design optimization were highlighted to further improve the stage’s performance.

The in-house mean-line design code predicted a pressure ratio and efficiency of $\Pi_{TT} = 6.0$ and 75.5%, respectively for a mass flow rate of 3.5 kg/s. The mean-line design code obviously lacked the ability to fully capture three-dimensionality, viscous flow, and compressible flow effects due to its inherent over-simplifying assumptions. The inclusion of the RANS-based computations improved the fidelity of the mixed-flow compressor design performance calculations significantly. Comprehensive computational analysis in the current stage showed that the design goal was met with a stage total pressure ratio of $\Pi_{TT} = 5.83$ and an efficiency of $\eta_{IS} = 77\%$ for a mass flow rate of $\dot{m} = 3.03$ kg/s. A total pressure ratio of 6.12 was achieved at a slightly higher rotational speed of $\Omega/\Omega_c = 1.035$ for an efficiency of 75.5%.
TABLE OF CONTENTS

List of Figures .................................................................................................................. vii
List of Tables ....................................................................................................................... vi
List of Symbols ..................................................................................................................... v
Acknowledgements .............................................................................................................. vii
Subscripts ............................................................................................................................ xiii

Chapter 1 Introduction ......................................................................................................... 1
  1.1 Necessity for a mixed-flow compressor ................................................................. 1
  1.2 Historical developments of mixed-flow compressor ............................................. 4
  1.3 Objective and scope of the thesis ........................................................................ 11

Chapter 2 Mean line Stage Design and Geometry ............................................................. 13
  2.1 Rotor 1D Design ..................................................................................................... 13
  2.2 Stator 1D Design .................................................................................................... 16
  2.3 Rotor Geometry ..................................................................................................... 19
  2.4 Tandem stator strategy and geometry definition ................................................... 23

Chapter 3 CFD solver validation and Mesh dependence study ........................................... 28
  3.1 Computational model ............................................................................................ 28
  3.2 Assessment of the computational model using existing compressor rotor and
diffuser data ....................................................................................................................... 31
    Verification 1: NASA Rotor 37 Experiment ............................................................... 31
    Verification 2: RWTH, Aachen Supersonic Tandem Stator Experiment ................... 37
  3.3 Mesh dependency study ........................................................................................ 40

Chapter 4 Stage design evaluation ..................................................................................... 45
  4.1 Rotor design consideration .................................................................................... 45

Chapter 5 Aerodynamic Analysis of the compressor stage ............................................... 57
  5.1 Rotor Flow feature description ............................................................................. 57
  5.2 Stator Flow feature description ............................................................................ 64
  5.3 Stage Performance analysis .................................................................................. 70

Chapter 6 Conclusions ...................................................................................................... 75
  6.1 Future scope of improvements ............................................................................. 79
Appendix............................................................................................................................................. 80

1. Inlet parameters................................................................................................................................ 80
2. Mixed-flow rotor 3D design input parameters.................................................................................... 81
3. Mixed-flow rotor non-dimensionalized airfoils.................................................................................... 82
4. Tandem stator 3D design input parameters.......................................................................................... 83

Bibliography ............................................................................................................................................. 85
LIST OF FIGURES

Figure 1-1 Types of compressors: a) Centrifugal compressor [42], b) Axial-flow compressor [41], c) Mixed-flow compressor. .................................................. 3

Figure 1-2 Mixed-flow compressor stage, Left: meridional and Right: frontal view [1]. ...... 4

Figure 1-3 Transonic mixed-flow compressor as an inlet fan stage by Dodge et al [7]. ........ 4

Figure 1-4 A 3:1 mixed-flow compressor stage featuring tandem stator blades by Musgrave and Plehn [8]. ................................................................. 5

Figure 1-5 A 5.5:1 mixed-flow compressor stage designed by Eisenlohr and Benfer [9] ...... 6

Figure 1-6 A 5:1 supersonic mixed-flow compressor designed by Elmendorf et al. [10] ...... 6

Figure 1-7 Meridional view summary of the previous mixed-flow compressor stage designs [10] ........................................................................................................ 7

Figure 1-8 Subsonic mixed-flow compressor stage by Youssef and Weir [11]................. 7

Figure 1-9 A 5:1 pressure ratio mixed-flow compressor stage design by Giri et al [13]. ...... 8

Figure 1-10 Past designs points of mixed-flow compressors (green diamond is the current design). ................................................................. 9

Figure 1-11 Pressure ratio vs efficiency of mixed-flow compressors at design point, numbers [1-13] denote work cited in bibliography ........................................ 10

Figure 2-1 Meridional view of the stage showing φ, L and r definitions......................... 13

Figure 2-2 Rotor inlet velocity triangle at mean radius $r_{1m}$ showing the benefits of swirl..... 14

Figure 2-3 Rotor exit velocity triangle at mean radius, $r_{2m}$ .............................................. 16

Figure 2-4 Rotor mean camber lines defining the hub, mean and tip sections using a Bezier curve. Corresponding control points: a, b, c, d................................. 19

Figure 2-5 Control points ‘a’ and ‘b’ with the inlet velocity triangle at ‘a.’ ...................... 20

Figure 2-6 Final 3D solid model of the mixed-flow rotor after the airfoil addition to the camber sheet. ......................................................................................... 23

Figure 2-7 2D stator construction using the control points, inlet and exit flow angles......... 25

Figure 2-8 Polynomial function ‘P’ distributing airfoil thickness. ................................... 26

Figure 2-9 Final 3D solid model of a supersonic tandem stator ...................................... 27
Figure 3-1 Stage fluid domain ................................................................. 30

Figure 3-2 Geometry of NASA transonic rotor 37 experiments and computational domain .............................................................................................................. 32

Figure 3-3 NASA rotor 37 mesh description: (a) leading edge resolution, (b) 70% span ...... 32

Figure 3-4 (a) and (b) Total pressure ratio distribution in the span-wise direction at station 4 for \( m/m_{\text{choke}} = 0.98 \) (experiments are from Suder [20]) ..33

Figure 3-5 Total temperature ratio in the spanwise direction at station 4 for \( m/m_{\text{choke}} = 0.98 \) (experiments are from Suder [20]). ............................................. 36

Figure 3-6 CFD vs experimental results for the RWTH Aachen tandem stator cascade. ....39

Figure 3-7 Polyhedral mesh detail at the moderate level with boundary layer cells:............ 40

Figure 3-8 \( C_p \) distribution at 65% rotor blade span for the three mesh densities ..........42

Figure 3-9 Rotor inlet Mach number, \( M_1 \), for coarse, moderate, and fine meshes (circumferentially mass averaged). .................................................................................. 43

Figure 3-10 Stagnation pressure, \( \Pi_{TT} = P_o2/P_o1 \), for coarse, moderate, and fine meshes at the rotor exit, 150 mm (circumferentially averaged). ................................................. 43

Figure 3-11 Isentropic efficiency at rotor exit for coarse, moderate, and fine meshes at the rotor exit, 150 mm (circumferentially averaged). ............................................. 44

Figure 4-1 Rotor mean line evaluation study plots .......................................................... 46

Figure 4-2 Rotor flow exit angle variation with \( M_2 \) ........................................................................ 48

Figure 4-3 Rotor blade exit angle variation with \( M_2 \) ............................................................ 49

Figure 4-4 Rotor meridional exit angle, \( \Phi_2 \), variation in meridional view with \( \Phi_1 = 0^\circ \) ....51

Figure 4-5 Relative parametric analysis for meridional exit angle study. ......................... 51

Figure 4-6 Pressure ratio in the meridional view of six cases (a-f) ...................................... 52

Figure 4-7 Relative Mach number distribution for cases a-f in the meridional view .......... 53

Figure 4-8 Rotor axial length, \( L_R \), variation in meridional view ....................................... 54

Figure 4-9 Relative parametric study for rotor axial length ............................................. 54

Figure 4-10 Relative parametric variation for the rotor pitch study. .............................. 55

Figure 5-1 Meridional view of relative Mach number distribution showing the flow features at the design point: POS - passage oblique shock, TS - terminating
shock, Cs - casing BL separation, PS - passage shock, HLV - hub leading-edge vortex........................................................................................................58

Figure 5-2 Shock stability analysis using eqn. 5.1 in the rotor passage and the
corresponding computational relative Mach number distribution. .....................59

Figure 5-3 Relative Mach number distribution at a) 90%, b) 65%, c) 32.5%, d) 10% span
at the design point ........................................................................................................60

Figure 5-4 Pressure distribution meridional view at the rotor design point: .................61

Figure 5-5 Rotor blade suction side vortex visualization using Q-criterion. .....................62

Figure 5-6 Rotor blade pressure side vortex visualization using Q-criterion...............63

Figure 5-7 Blade loading distribution along blade meridional chord for 10, 65 and 90%
rotor span at the design point ........................................................................................................64

Figure 5-8 Static pressure rise comparison between mid-span 3D stator in stage and 2D
stator alone simulation for the same inlet boundary condition and similar
passage shock structure ..............................................................................................................66

Figure 5-9 Current tandem stator blade loading and corresponding Mach number contour...67

Figure 5-10 Entropy generation across the 2D tandem stator based on computational
effort ........................................................................................................................................69

Figure 5-11 Stage total pressure ratio and efficiency versus normalized mass flow rate, .....70

Figure 5-12 Stagnation pressure and static pressure distribution across the stage for
different back pressures ............................................................................................................73

Figure 5-13 Velocity component distribution across the stage at design speeds ..........74

Figure A-1 Non-dimensionalized NACA 65 series airfoil ...............................................82
List of Tables

Table 1-1 Comparative assessment of the three types of compressor stages .................. 2

Table 3-1 Stage boundary conditions from the mean line analysis .................................. 29

Table 3-2 Comparative performance assessment of rotor alone simulation for different turbulence closure models ........................................................................................................... 31

Table 3-3 RWTH Aachen tandem diffuser boundary conditions ........................................ 37

Table 3-4 A comparative assessment of mesh dependency ................................................. 42

Table 3-5 Rotor CFD simulation boundary conditions ....................................................... 44

Table 3-6 Mesh input data comparison between coarse, moderate and fine cases ............. 44

Table 4-1 Rotor mean-line parametric study variables ...................................................... 45

Table 4-2 Mixed-flow rotor computational study cases .................................................... 49

Table 5-1 Current tandem stator 2D CFD boundary conditions ........................................ 66

Table 5-2 Performance comparison between 2D stator alone and 3D stator in stage simulations ................................................................................................................................. 66

Table 5-3 Design point vs CFD comparison at maximum back pressure for the mixed-flow compressor stage ..................................................................................................................... 74

Table A-1 Compressor inlet parameters ........................................................................... 80

Table A-2 Rotor inlet and exit geometric conditions .......................................................... 81

Table A-3 Input parameters for rotor cases a-f .................................................................. 81

Table A-4 Stator hub and casing Bezier curve definitions ............................................... 83

Table A-5 Custom polynomial airfoil blade row 1 parameters ......................................... 83

Table A-6 Custom polynomial airfoil blade row 2 parameters ......................................... 84
# List of Symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A$</td>
<td>Area, [m$^2$]</td>
</tr>
<tr>
<td>$A'$</td>
<td>Area normal to rotational axis, [m$^2$]</td>
</tr>
<tr>
<td>$b$</td>
<td>Vane-less space depth, [m]</td>
</tr>
<tr>
<td>$BL$</td>
<td>Boundary layer</td>
</tr>
<tr>
<td>$C_a$</td>
<td>Axial absolute velocity, [m/s]</td>
</tr>
<tr>
<td>$C_p$</td>
<td>Specific heat at constant pressure, [J.(kg .K)$^{-1}$]</td>
</tr>
<tr>
<td>$C_p$</td>
<td>Pressure coefficient, [P-P$<em>1$] / [P$</em>{o1}$-P$_1$]</td>
</tr>
<tr>
<td>$CFD$</td>
<td>Computational fluid dynamics</td>
</tr>
<tr>
<td>$CFL$</td>
<td>Courant number</td>
</tr>
<tr>
<td>$Cs$</td>
<td>Casing separation</td>
</tr>
<tr>
<td>$DP$</td>
<td>Design point</td>
</tr>
<tr>
<td>$I_m$</td>
<td>Rothalpy, [J/ Kg$^{-1}$], $C_p . T + 0.5(W^2 - U^2)$</td>
</tr>
<tr>
<td>$HLV$</td>
<td>Hub leading-edge vortices</td>
</tr>
<tr>
<td>$Kn$</td>
<td>Knudsen number</td>
</tr>
<tr>
<td>$L$</td>
<td>Axial length, [m]</td>
</tr>
<tr>
<td>$\dot{m}$</td>
<td>Mass flow rate, [Kg/s]</td>
</tr>
<tr>
<td>$\dot{m}/\dot{m}_{DP}$</td>
<td>Normalized mass flow rate</td>
</tr>
<tr>
<td>$M$</td>
<td>Mach number</td>
</tr>
<tr>
<td>$M_{rel}$</td>
<td>Relative Mach number</td>
</tr>
<tr>
<td>$N$</td>
<td>Number of blades</td>
</tr>
<tr>
<td>$NACA$</td>
<td>National Advisory Committee for Aeronautics, former NASA</td>
</tr>
<tr>
<td>$OS$</td>
<td>Oblique shock</td>
</tr>
<tr>
<td>$P$</td>
<td>Static pressure, [Pa]</td>
</tr>
<tr>
<td>$P_o$</td>
<td>Total pressure, [Pa]</td>
</tr>
<tr>
<td>$PS$</td>
<td>Passage shock</td>
</tr>
<tr>
<td>$PS$</td>
<td>Pressure side</td>
</tr>
</tbody>
</table>
\[ r = \text{Radius, [m]} \]
\[ R = \text{Gas constant, [Air \ R=287 \text{ J/Kg .K}]} \]
\[ RANS = \text{Reynolds-averaged Navier-Stokes} \]
\[ RPM = \text{Revolutions per minute} \]
\[ S = \text{Blade pitch} \]
\[ SS = \text{Suction side} \]
\[ t = \text{Pitch-wise distance between B1 & B2} \]
\[ temp = \text{Temporary variable} \]
\[ T = \text{Static temperature, [K]} \]
\[ T_o = \text{Total temperature, [K]} \]
\[ TS = \text{Terminating shock} \]
\[ u = \text{Absolute velocity, [m/s]} \]
\[ U = \text{Tangential blade velocity, [m/s]} \]
\[ UAV = \text{Uninhabited air vehicle} \]
\[ W = \text{Relative velocity, [m/s]} \]
\[ X = \text{Axial location, [m]} \]
\[ Y+ = \text{Wall coordinate based on y and friction velocity} \]
\[ \nu_T = \text{Turbulent-to-molecular viscosity ratio} \]
\[ \Delta = \text{Control point} \]
\[ \phi = \text{Flow coefficient} \]
\[ \gamma = \text{Specific heat ratio} \]
\[ \rho = \text{Density, [kg/m^3]} \]
\[ \eta_{IS} = \text{Isentropic efficiency (total-to-total)} \]
\[ \Pi_{TT} = \text{Pressure ratio (total-to-total)} \]
\[ \Omega = \text{Angular rotational speed, [rad./s], design value = 28,500 RPM} \]
\[ \Phi = \text{Meridional angle, [degrees]} \]
\[ \beta = \text{Blade angle, [degrees]} \]
\[ \alpha = \text{Flow angle, [degrees]} \]
\[ \lambda \] \quad = \quad \text{Wedge angle, [degrees]} 

**Subscripts**

\[ t \] \quad = \quad \text{Tip} \\
\[ h \] \quad = \quad \text{Hub} \\
\[ m \] \quad = \quad \text{Mid} \\
\[ R \] \quad = \quad \text{Rotor} \\
\[ \text{rot} \] \quad = \quad \text{Rotating frame} \\
\[ \text{VS} \] \quad = \quad \text{Vane-less space} \\
\[ S \] \quad = \quad \text{Stator} \\
\[ 1 \] \quad = \quad \text{Rotor inlet station} \\
\[ 2 \] \quad = \quad \text{Rotor exit station} \\
\[ 3 \] \quad = \quad \text{Stator inlet station} \\
\[ 4 \] \quad = \quad \text{Stator exit station}
ACKNOWLEDGEMENTS

I would first and foremost like to express my sincere gratitude in thanking my advisor, Dr. Cengiz Camci. His words of encouragement, patience, and knowledge immensely guided and propelled me to complete my thesis. Throughout my master’s thesis, his confidence in me, his advice, and truly visionary ideas pushed me to accomplish this amazing task.

I owe special thanks to Shankar Ram who shared my difficulties, provided emotional support, and kept me motivated. I also want to thank my friends Tomas Opazo, Andrea Opazo, Bolor Erdene-Zolbayar for being guiding lights throughout this journey.

I would like to thank my lab mates Mitansh Doshi, Amrat Ranka, Gohar Khokar, and Veerendara Andichamy for providing me with their constructive criticism and support in this effort.

My brother, Kaushik Narayanan and parents, M. Sadagopan and K. Tamilarasi, deserve my highest respect and deepest regard. It would not have been possible for me to achieve my aspirations without their continuous support, guidance, and blessings.
Chapter 1

Introduction

The small aero-engine market has seen enormous growth over the past decade with a wide variety of applications in UAVs. Piston engines have dominated this segment because of the lack of high-performance and cost-effective small jet engines. But, a highly loaded compact jet engine with high altitude operational capability could bridge this gap. Improvisation on conventional jet engine designs with reduced mechanical component volume and weight will lead to higher reliability, reduced component manufacturing costs, and improved engine life-cycle. Since engine performance and efficiency cycle depends highly on the compressor, further development is needed for compact high-performance compression systems.

1.1 Need for a mixed-flow compressor

Previously, compact compression centrifugal compressor designs with high stage pressure ratios were developed, but the main drawback was an inherent large frontal diameter, due to the radial diffuser. This relatively large diameter limited use in aircraft propulsion applications. Conversely, an axial flow compressor requires a longer axial length with multiple stages to achieve the same overall pressure ratio. A mixed-flow single-stage design with a smaller frontal area and a higher pressure ratio is an effective solution to these problems. It provides the robustness and work output of a centrifugal compressor, but in a much shorter length. The reduced number of stages in the mixed-flow design for the same compression leads to significant cost reduction in manufacturing, possible reliability improvements, and reduced maintenance time/cost. The mixed-flow compressor can also better respond to foreign object damage and flow distortion than an axial design. The shorter system length and reduced weight is also highly beneficial. These three types
of compressor systems are presented in Figure 1-1. A comparative property description of the compressors has been given in Table 1-1.

**Table 1-1** Comparative assessment of the three types of compressor stages.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Centrifugal</th>
<th>Axial-flow</th>
<th>Mixed-flow</th>
</tr>
</thead>
<tbody>
<tr>
<td>Frontal diameter</td>
<td>High</td>
<td>Low</td>
<td>Low</td>
</tr>
<tr>
<td>Axial length</td>
<td>Low</td>
<td>High</td>
<td>Medium</td>
</tr>
<tr>
<td>Multi-stage necessity</td>
<td>No</td>
<td>Yes</td>
<td>No</td>
</tr>
<tr>
<td>Robustness and high work output (in a single stage)</td>
<td>Yes</td>
<td>No</td>
<td>Yes</td>
</tr>
<tr>
<td>High efficiency</td>
<td>No</td>
<td>Yes</td>
<td>Yes</td>
</tr>
<tr>
<td>Low manufacturing cost</td>
<td>Yes</td>
<td>No</td>
<td>Yes</td>
</tr>
<tr>
<td>Foreign object resistance</td>
<td>Yes</td>
<td>No</td>
<td>Yes</td>
</tr>
<tr>
<td>Flow distortion sensitivity</td>
<td>No</td>
<td>Yes</td>
<td>No</td>
</tr>
<tr>
<td>Single stage weight</td>
<td>High</td>
<td>Low</td>
<td>Medium</td>
</tr>
</tbody>
</table>

From a utility perspective, combined mixed, axial, or centrifugal stages can provide total pressure ratios up to 10-15, which would be quite advantageous for a turbo-shaft engine or a turbo-fan arrangement with a reduced frontal diameter. This increases the engine’s performance (thrust/weight ratio) by a significant margin. Looking to the future, these systems could also be advantageously incorporated into the upcoming geared-turbo-fan (GTF) core compressor designs [18] and the NASA distributed propulsion vehicle concept [19]. A mixed-flow compressor stage, when installed in the compressor core, could easily replace a multi-stage axial system. The mixed-flow approach has great potential to shorten the core compressor length. This hybrid design would involve lesser moving components, hence, reducing weight and system complexity, thus increasing the reliability factor. Another benefit is the possible reduction in manufacturing and maintenance costs. If incorporated in the distributed propulsion systems, a mixed-flow compressor could provide additional drag reduction benefits and variable duct-type compact compressor applications based on the location of the engine core.
Figure 1-1 Types of compressors: a) Centrifugal compressor [42], b) Axial-flow compressor [41], c) Mixed-flow compressor.
1.2 Historical developments of the mixed-flow compressor

The first mixed-flow design to be experimentally evaluated was by King and Glodeck (1942) [1], as shown in Figure 1-2. This 23-bladed rotor did not feature the typical centrifugal rotor inducer. The design was very sensitive to frontal tip clearance. The reasons for poor stage efficiency were excessive losses in the stator due to a smaller expansion ratio, the large collector area causing sudden expansion, and incidence effects at the stator inlet.

![Figure 1-2 Mixed-flow compressor stage, Left: meridional and Right: frontal view [1].](image)

A series of experiments were then conducted by the NACA during the 1950s and 1960s. Wilcox and Robbins (1951) [4] used a supersonic diffuser with bleed air to improve stage performance. It suffered from significant total pressure losses, due to poor flow distribution and mixing. Blade wakes and shock losses equally contributed to the inefficiency of this diffuser.

![Figure 1-3 Transonic mixed-flow compressor as an inlet fan stage by Dodge et al [7].](image)
Pump, turbocharger and UAV applications slowly propelled mixed-flow design developments for the next three decades. During the 1980s, Dodge (1987) [7] patented a mixed-flow impeller as an inlet transonic fan (shown in Figure 1-3). A three-stage axial flow compressor was found to be replaceable by the single stage 3:1 mixed-flow design by Musgrave and Plehn (1987) [8]; seen in Figure 1-4. It features a subsonic tandem stator built on the philosophy that the low solidity first blade row being lightly loaded could accommodate significant incidence variations resulting from changes in mass flow or entry profile. The second blade row being highly loaded could function over a wide variety of operating conditions with a constant incidence.

Another design provided an impeller total pressure ratio of $\Pi_{TT} = 7.5:1$ at 91% efficiency with a high $r_{2m}/r_{1m}$ ratio, but the splitter blade suffered from overall stage loss with a single-vaned diffuser. This stage was designed by Eisenlohr and Benfer [9], shown in Figure 1-5, and was able to reach a peak efficiency of merely 72% with $\Pi_{TT} = 5.5:1$. The stator diffusion system failed, due to the extreme blade loading in a single-vane row configuration. Eisenlohr and Benfer instead recommended a tandem-type diffusing system to achieve higher stage performance.

Figure 1-4 A 3:1 mixed-flow compressor stage featuring tandem stator blades by Musgrave and Plehn [8].
Elmendorf et al. [10] conducted an extensive study on the prospect of a mixed-flow design applied to small jet engines. A 5:1 shock-in-rotor configuration was developed with a shock stabilizing technique, which was combined with a supersonic tandem-type diffuser. This stage, as shown in Figure 1-6, resulted in reduced overall stage performance, due to the diffuser. The impeller exit Mach number proved to be crucial in defining an efficient diffuser design.

The potential of a single-stage mixed-flow design to be used as an aircraft compressor is realizable, but all these designs suffered from shock losses in the diffuser heavily penalizing the stage efficiency. To overcome this limitation, Youssef [11] resorted to a subsonic mixed-flow
compressor stage. The patented design had a two-stage mixed-flow/centrifugal compressor claiming to achieve an overall compression, $\Pi_{TT}$, of 10-13 by featuring an intermediate duct. The rotor exit flow from the mixed-flow stage had a subsonic absolute velocity. The intermediate duct then reduced the inlet diameter of the centrifugal stage by directing the airflow radially inward as shown in Figure 1-8, leading to a reduction in swirl and an increment in diffusion.

**Figure 1-7** Meridional view summary of the previous mixed-flow compressor stage designs [10]

**Figure 1-8** Subsonic mixed-flow compressor stage by Youssef and Weir [11]
More recently, an impeller tip clearance study conducted by S. Ramamurthy et al. [15] described early surging with higher tip clearance due to the unsteady interaction of the main flow with the leakage flow. The study stated that constant tip clearance provided better performance over variable tip clearance. It also stated that typical jet wake flow and recirculation in the inlet of radial machines were absent in the mixed-flow configuration. The stage design yielded Π_{TT} of 4.55 with η_{IS} of 80% for \( \dot{m} \) of 3.3-3.36 kg/s. A robust design optimization study by Cevik et al. [37] determined that the mixed-flow impeller performed better for a small jet engine than a small radial compressor. The study minimized the cost function based on specific thrust and TSFC (Thrust specific fuel consumption).

A critical study on mixed-flow impeller blade loading distribution was conducted by Xuanyu et al. [14]. An increased blade loading at the hub’s posterior area was desired for better performance, due to control of hub flow separation and it being more adaptable to the changing flow passage. Similarly, increased frontal tip section loading would achieve a higher total pressure ratio in surging conditions. With a mass flow of 18 kg/s, the stage design achieved a Π_{TT} of 2.72.

![Figure 1-9](image)

**Figure 1-9** A 5:1 pressure ratio mixed-flow compressor stage design by Giri et al [13].

Giri et al. [13] designed a mixed-flow impeller and diffuser followed by an axial stator, see Figure 1-9. The impeller inlet and exit end wall contour design minimized losses. The splitter blade length was extended to obtain a higher Π_{TT}. Providing pre-swirl at the impeller inlet improved the
efficiency by reducing the inlet Mach number and it attenuated losses that occurred due to high inlet velocities. The design achieved a $\Pi_{TT}$ of 5.2 with $\eta_{IS}=79.1\%$ for an $\dot{m}$ of 1.74 kg/s.

![Figure 1-10](image)

Figure 1-10 Past designs points of mixed-flow compressors (green diamond is the current design).

Many of the past mixed-flow designs including the current design are presented in Figure 1-10. Figure 1-11 shows the pressure ratio versus stage efficiency for all the designs given in Figure 1-10. Figure 1-11 shows that obtaining a high-pressure ratio and efficiency simultaneously for a single stage was quite challenging. There were only two mixed-flow designs over a pressure ratio of $\Pi_{TT} = 5$. The present design has the highest pressure ratio of 6 at an elevated engine mass flow rate of 3.5 kg/s. There were only four mixed-flow designs with an efficiency $\eta_{IS}$, over 80%. However, all four designs exhibited a pressure ratio of less than 3.7. The current design generated a pressure ratio of 6 at an engine mass flow rate of 3.5 kg/s with a total-to-total efficiency of $\eta_{IS} = 75.5\%$. 
Since most of the earlier designs suffered from poor stator performance, the main challenge is the recovery of stagnation pressure with a high isentropic efficiency in the presence of a supersonic inlet and a large turning angle in the stator. To reduce the blade loading in a single-stator row, many studies have utilized tandem designs for both subsonic and supersonic cases. However, past performances of the supersonic tandem stators were not satisfactory. Recently, Quishi et al. [16] proposed and investigated a highly loaded supersonic tandem stator with two cascade rows based on an aspirated-fan concept. The first row of the cascade had a supersonic airfoil that reduced the flow to be subsonic and the second row provided large flow turning in a subsonic domain. Losses were minimized when the leading edge of the second row was close to the pressure side of the first row at a 20% pitch-to-pitch distance with no axial spacing or overlapping. Incorporation of this
design feature into a tandem-type stator design for a mixed-flow compressor could provide a relatively high stage pressure-ratio and efficiency.

1.3 Objective and scope of the thesis

An aircraft flying at an altitude of 38000 ft has an engine inlet Mach number of 0.85, which is isentropically reduced to 0.7 before entering the compressor using an inlet diffuser. The prime objective is to achieve a relatively high $\Pi_{TT}$ (near 6) in a single-stage compressor for an $\dot{m}$ of 3.5 kg/s with a 400 mm maximum outer diameter for this engine.

Most of the available commercial design codes contain undisclosed procedures, which are heavily dependent on the code developer’s intuition and experience. The codes do not provide adequate transparency to the user. Many mean-line design illustrative procedures are available for centrifugal and axial compressor stages, but very few attempts have been made on mixed-flow stage designs. Chapter 2 provides strategic guidance to design a high-pressure ratio and high-performance mixed-flow compressor. A simple mean-line procedure was defined to relate mixed-flow compressor geometric and aero-thermal parameters to specific design requirements. The rotor mean-line design procedure was followed by a unique tandem stator design. It is followed by a design geometry generation procedure, primarily based on Bezier curves.

Chapter 3 describes the multi-dimensional viscous/turbulent computational model utilized in this study. It was assessed against two well-publicized test cases, which emulate a transonic rotor and a supersonic diffuser. The experimental data sets from NASA rotor 37 [20] and the RWTH supersonic tandem stator [10] were compared to the current computational results using RANS-based viscous flow simulations. The computational results sufficiently predicted the exit profile distribution for rotor 37 and the $C_P$ distribution on the RWTH supersonic tandem stator. After
validating the computational model, an analysis was performed on the current \( \Pi_{TT} = 6 \) class mixed-flow supersonic compressor and is discussed in detail.

Chapter 4 presents a detailed design evaluation on the mixed-flow stage. The rotor design was evaluated based on the mean-line equations for rotor exit radius, blade and flow exit angle, and exit absolute and relative Mach number. Other crucial parameters, such as rotor meridional exit angle, axial length, blade solidity, blade loading, and tip leakage were studied using multidimensional viscous computational analysis.

Chapter 5 summarizes with performance results of the developed mixed-flow design, including viscous flow features, compressibility issues, shock waves, and resulting aerodynamic losses. The aerothermal characteristics of a \( \Pi_{TT} = 6 \) class mixed-flow compressor stage were studied based on a computational RANS analysis. Stage performance charts of this design are presented to highlight the supersonic diffusion-related flow features, including the shock wave system. A comparative study was presented between the mean-line design input data and the final CFD-based performance assessments that were multi-dimensional, viscous, turbulent, and rotational.
Chapter 2

Mean line stage design and geometry

A simple one-dimensional procedure to design a supersonic compressor stage using the design point value is described below. The fundamental concepts involved in designing a turbomachinery device are illustrated.

2.1 Rotor 1D design

![Figure 2-1 Meridional view of the stage showing φ, L and r definitions](image)

The meridional view of a mixed-flow stage is described in Figure 2-1. Inlet geometric parameters at station ‘1’ are defined using the design point specifications. Operational altitude and compressor inlet Mach number (M₁) provide the free stream density (ρ₁) and inlet velocity (C₁). Design mass flow rate (\(\dot{m}\)) fixes the inlet area (A₁) using the continuity equation. A higher A₁ is required with altitude to accommodate the same \(\dot{m}\) through the engine. Inlet meridional angle (Φ₁) can be varied depending upon the compressor stage application.
Eqn. (2.1) relates hub radius, \( r_{1h} \), and tip radius, \( r_{1t} \), for a given \( \dot{m} \). Having a lower \( r_{1h} \) is beneficial, because it increases the \( r_{2m} \)/\( r_{1m} \) ratio, hence increasing the rotor \( \Pi_{TT} \). However, \( r_{1h} \) is limited based on the structural/design constraints of the shaft for the chosen RPM and \( r_{1h} \)/\( r_{1t} \) ratio for the blade structure. The mean radius, \( r_{1m} \), splits the inlet area \( A_1 \) based on \( \dot{m} \).

\[
r_{1t} = \sqrt{\left( \frac{\pi \dot{m}}{4 C_1 \rho_1} \right) + r_{1h}^2}
\]  

(2.1)

RPM is determined by limiting the inlet tip relative Mach number (\( M_{1rel} \)). Based on a literature survey in [43] for a transonic compressor rotor, an \( M_{1rel} \) of 1.4 - 1.5 is typically chosen to minimize losses due to passage shock. Eqn. (2.2-2.4), developed from the inlet velocity triangle, are used to obtain the shaft rotational rate, \( \Omega \).

\[
\beta_{1t} = \cos^{-1} \left( \frac{C_{1t} \alpha_{1t}}{W_{1t}} \right)
\]  

(2.2)

\[
U_{1t} = C_{1t} \tan(\beta_{1t})
\]  

(2.3)

\[
\Omega = \frac{U_{1t} \cdot 60}{2 \cdot r_{1t} \cdot \pi}
\]  

(2.4)

**Figure 2-2** Rotor inlet velocity triangle at mean radius \( r_{1m} \) showing the benefits of swirl.

\( C'_{1m} = C_{1m} \); \( U_{1m} \) is the same; \( W'_{1m} < W_{1m} \).

Flow swirling at the rotor inlet reduces \( M_{1rel} \) and increases the rotor efficiency by curtailing passage shock losses. This effect is seen in **Figure 2-2**, showing the inlet velocity triangle at \( r_{1m} \).
The challenge with this configuration is designing a transonic airfoil with high total pressure recovery and flow turning capability. At this point, all the inlet rotor blade parameters, including inlet blade angles (β₁), have been characterized at the hub, mean and tip radius locations. *Appendix 1* provides the inlet values used in this design effort.

Rotor exit parameters ‘2’ are obtained along the mean-line. Total pressure and temperature at the rotor exit for a design point pressure ratio Πₜₜ and an approximated ηᵢₛ are found using the definition of isentropic total-to-total efficiency; see *eqn. (2.5)*.

\[
T_{o2} = T_{o0} + (\Pi_{TT}^{(\gamma-1)/\gamma} - 1) \cdot (T_{o0}/\eta_{iS})
\]  

*eqn. (2.5)*

The exit Mach number, \(M₂\), is then approximated in the range of 1 to 1.4 in order to obtain the exit static parameters using *eqn. (2.6-2.8)*.

\[
T₂ = T_{o2} / (1 + 0.2 \, M₂^2)
\]  

*eqn. (2.6)*

\[
P₂ = P_{o2} / (T_{o2}/T₂)^{(γ/(γ-1))}
\]  

*eqn. (2.7)*

\[
ρ₂ = P_{o2} / (R \cdot T_{o2} \cdot (1 + 0.2 \, M₂^2)^{(1/(γ-1))})
\]  

*eqn. (2.8)*

\[
I_{m1} = C_p \cdot T₁ + 0.5 \cdot (W₁m^2 - U₁m^2)
\]  

*eqn. (2.9)*

\[
I_{m2} = I_{m1}
\]  

*eqn. (2.10)*

\[
U₂m = Ω \cdot r₂m
\]  

*eqn. (2.11)*

\[
W₂m = \sqrt{I_{2m} - C_p \cdot T₂ + U₂m^2}
\]  

*eqn. (2.12)*

The rotalpy conservation principle between the rotor inlet and exit along the mean-line, as in *eqn. (2.9-2.10)*, is used to connect rotor inlet and exit stations. \(C₂m\) is obtained using the approximated \(M₂\) and \(a₂\). \(W₂m\) and \(U₂m\) are functions of \(r₂m\); see *eqn. (2.11-2.12)*. \(M₂em\) is approximated to determine \(r₂m\). Since this is an iterative procedure, detailed parametric analysis is discussed in *Chapter 4*. A chosen \(r₂m\) value is used to obtain \(α₂m\) and \(β₂m\) and the exit velocity triangle at the mean radial location is derived as shown in *Figure 2-3*. 
Figure 2-3 Rotor exit velocity triangle at mean radius, $r_{2m}$.

A positive $\beta_{2m}$ is desired, since back sweep provides more rotor stability [38]. The meridional exit angle ($\Phi_2$) and the rotor axial length ($L_R$) are chosen as described in Chapter 4. $r_{2t}$ and $r_{2h}$ are evaluated again using the continuity equation and the meridional exit angle ($\Phi_2$), as in eqn. (2.13-2.17). A similar procedure is followed to characterize the exit hub and tip velocity triangles. Appendix 2 provides the values corresponding to the design of this rotor.

\[ C_{a2m} = C_{2m} \cos (\alpha_{2m}) \]  
(2.13)

\[ A'_2 = \frac{\dot{m}}{\rho_2 \cdot C_{a2m}} \]  
(2.14)

\[ r_{2h} = \sqrt{r_{2m}^2 - \frac{A'_2 \cos(\Phi_2)}{2 \pi}} \]  
(2.15)

\[ r_{2t} = \sqrt{2 \cdot r_{2m}^2 - r_{2h}^2} \]  
(2.16)

\[ A_2 = \frac{\pi}{\cos(\Phi_2)} \cdot (r_{2t}^2 - r_{2h}^2) \]  
(2.17)

2.2 Stator 1D Design

Exit Mach number, $M_2$, from the rotor exit is supersonic, but it has subsonic radial, tangential, and axial components for the current design; eqn. (2.18-2.20).
Stagnation enthalpy is assumed to be conserved in the vane-less space (VS) along the mean-line, hence $T_{o3} = T_{o2}$. According to the initial isentropic flow approximation, $P_{o3} = P_{o2}$. The hub and casing radius in the VS are designed to reduce the cross-sectional area in the axial direction to diffuse the supersonic flow. The curvature is designed to turn the radial velocity component to the axial direction. The axial length ($L_{VS}$) in this region determines the rotor-stator interaction. A length exceeding 15 mm generates a normal shock in the VS. Since this region is attempting to diffuse a supersonic flow, any increment in length will correspond to shock instability. A very low $L_{VS}$ value is acoustically undesirable.

The radial vane-less diffusion follows the conservation of mass and momentum equations. The flow angle depends on the density and passage depth, as in Japikse and Baines [39]. Since the current design incorporates varying VS depth ($b$) and $\rho$ (compressibility effect), $\alpha_{3m}$ will vary with $\alpha_{2m}$, as seen in eqn. (2.21-2.22).

$$r_m C_\theta = \text{const.} \quad (2.21)$$

$$\tan(\alpha) = \frac{C_0}{C_m} = (\text{const.} \rho b)/\dot{m} \quad (2.22)$$

The area ratio ($A_3/A_2$) is obtained using the isentropic area eqn. (2.23) by specifying an $M_3$ value. The values of $\Phi_3$ and $r_{3m}$ are specified based on the rotor exit meridional angle, $\Phi_2$, and mean radius, $r_{2m}$, $r_{3h}$ and $r_{3t}$ are then extracted using the continuity equation; see eqn. (2.24-2.25). In this case, the value of $r_{3m}$ is higher than $r_{2m}$. The choice of $\Phi_3$ depends on how the VS is intended to perform. A positive meridional angle, $\Phi_3$, will lead to enhanced diffusion due to an increasing mean radius with the same $L_{VS}$, but $\Phi_3$ is closer to $0^\circ$ due to the outer diameter constraints. This leads to a two-dimensional inlet for the vaned stator with $M_{3\theta}$ and $M_{3x}$ components obtained from eqn. (2.26-2.27).
\[
\frac{A_3}{A_2} = \frac{M_2}{M_3} \left[ \frac{1 + \left( \frac{y-1}{2} \right) \left( \frac{M_2^2}{y} \right)^{\frac{y+1}{2(y-1)}}}{1 + \left( \frac{y-1}{2} \right) \left( \frac{M_3^2}{y} \right)^{\frac{y+1}{2(y-1)}}} \right], \text{ where, } \frac{A_3}{A_2} < 1
\] (2.23)

\[
r_{3h} = \sqrt{r_{3m}^2 - \frac{A_3 \cdot \cos(\Phi_3)}{2\pi}}
\] (2.24)

\[
r_{3t} = \sqrt{r_{3m}^2 + \frac{A_3 \cdot \cos(\Phi_3)}{2\pi}}
\] (2.25)

\[
M_{3h} = M_3 \cdot \cos(\alpha_{3m})
\] (2.26)

\[
M_{3b} = M_3 \cdot \sin(\alpha_{3m})
\] (2.27)

The objective for a stator in this design approach is to diffuse the incoming supersonic flow, to turn it to the axial direction, and to recover the maximum stagnation pressure within the external diameter and axial length constraints. After analyzing a number of possible supersonic diffuser configurations, a tandem stator configuration based on an aspirated-fan design was selected [40].

The tandem diffuser uses two-blade rows with the first row diffusing supersonic flow and the second row turning and diffusing subsonic flow. Stator inlet parameters are assessed from the VS exit mean-line values. The crucial quantities to be determined in order to design this component are the inlet-exit area ratio \(A_4/A_3\) and the axial length \(L_S\). Stator vanes are required to turn the tangential velocity component to the axial direction and diffuse it. \(M_4\) was chosen between 0.3 and 0.5 and the exit area, \(A_4\), was evaluated based on eqn. (2.23) using station 3 values. \(r_{4m}\) was chosen to be equal to \(r_{3m}\). \(r_{4m}\) could be higher than \(r_{3m}\), which in turn would aid radial diffusion if stage external diameter was not a constraint. This is a design choice to be based on the application. However, \(\alpha_{4m}\) was chosen as 0°. \(r_{4h}\) and \(r_{4t}\) were obtained from \(r_{4m}\) and \(A_4\) values using eqn. (2.24-2.25). \(L_S\) was chosen as 150mm for this case, based on the tandem stator aspect ratio definition in [16]. Stagnation enthalpy was assumed to be conserved across stations 3 and 4. The static terms at station 4 were then determined using the isentropic equations.
2.3 Rotor Geometry

The rotor camber line at the hub, mean, and tip sections was constructed using a Bezier curve. Blade parameters characterized in 2.1 were used as the input to define this curve. A 3rd order curve was chosen due to the availability of four definitive points for each section. X, Y and Z definitions for each point were geometrically derived using the mean-line parameters. H, M, and T, eqn. (2.41-2.43), each represented the individual 3x4 matrices of the XYZ points for the corresponding Bezier curve. A 3rd order Bernstein matrix was then multiplied with each H, M, and T matrix to obtain the three curves generated between 0 and 1 to obtain the camber line shown in Figure 2-4.

Figure 2-4 Rotor mean camber lines defining the hub, mean and tip sections using a Bezier curve. Corresponding control points: a, b, c, d.
Point ‘a’ at H, M, and T is defined by the inlet r value and δa, using eqn. (2.28), where \( \Delta_1 \) is the hub X location of point ‘b.’ The \( \Delta_1 \) location is within 20-30% of the total rotor axial length, \( L_R \). T and M ‘X’ locations are given by eqn. (2.29-2.30). \( \Delta_3 \) is a control factor used to adjust the point ‘b’ location, and it is usually in the range of 1 to 3. The second row of the H, M, and T matrices shows the X, Y, and Z definitions for point ‘b.’ Figure 2-5 shows the geometric definitions for points ‘a’ and ‘b.’

![Diagram of point a and b](image)

**Figure 2-5** Control points ‘a’ and ‘b’ with the inlet velocity triangle at ‘a.’

\[
\delta_a = \tan^{-1} \left( \frac{\Delta_1 \tan(\beta_{1h})}{r_{1h}} \right) \tag{2.28}
\]

\[
X_{bm} = r_{1m} \cdot \frac{\tan \delta_a}{\tan \beta_{1m}} \tag{2.29}
\]

\[
X_{bt} = r_{1t} \cdot \frac{\tan \delta_a}{\tan \beta_{1t}} \tag{2.30}
\]

\[
temp_{0i} = \sqrt{(r_{1i} \cdot \tan \delta_a)^2 - (r_{1i} \cdot \tan \delta_a)^2 + r_{1i} \cdot \cos \delta_a + \Delta_1 \cdot \cos \beta_{1h} \cdot \tan \Phi_{1i}}, \text{ where } i=h, m \text{ & } t \tag{2.31}
\]
\[ \delta_d = \tan^{-1} \left( \frac{(L_R - \Delta_2) \tan(\beta_{2h})}{r_{2h}} \right) \]  
\[ X_{ch} = L_R - \Delta_2 \]  
\[ X_{cm} = r_{2m} \cdot \frac{\tan \delta_d}{\tan \beta_{2m}} \]  
\[ X_{ct} = r_{2t} \cdot \frac{\tan \delta_d}{\tan \beta_{2t}} \]  
\[ \text{temp}_{1i} = \sqrt{(r_{2i} \cdot \tan \delta_d)^2 - (r_{2i} \cdot \sin \delta_d)^2} + r_{2i} \cdot \cos \delta_d \]  
\[ \text{temp}_{2i} = \tan \Phi_{2i} \cdot \sqrt{(X_{ci})^2 + (r_{2i} \cdot \sin \delta_d)^2 + (r_{2i} \cdot \cos \delta_d - \text{temp}_{1i})^2} \]  
\[ \text{temp}_{3i} = \frac{r_{2i} \cdot \cos \delta_d - \text{temp}_{1i}}{r_{2i} \cdot \sin \delta_d} \]  
\[ Y_{ci} = \sqrt{\frac{(\text{temp}_{2i} \cdot \text{temp}_{3i})^2}{1 + \text{temp}_{3i}^2}} \]  
\[ Z_{ci} = \text{temp}_{1i} + \frac{Y_{ci}}{\text{temp}_{3i}} \]  
\[ H = \begin{bmatrix} 0 & -r_{1h} \cdot \sin \delta_a & r_{1h} \cdot \cos \delta_a \\ \frac{\Delta_1}{\Delta_3} & -r_{1h} \cdot \frac{\sin \delta_a}{\Delta_3} & \frac{r_{1h} \cdot \cos \delta_a}{\Delta_3} \\ X_{ch} & Y_{ch} & Z_{ch} \\ L_R \cdot r_{2h} \cdot \sin \delta_d & r_{2h} \cdot \cos \delta_d \end{bmatrix} \]  
\[ M = \begin{bmatrix} 0 & -r_{1m} \cdot \sin \delta_a & r_{1m} \cdot \cos \delta_a \\ \frac{X_{cm}}{\Delta_3} & -r_{1m} \cdot \frac{\sin \delta_a}{\Delta_3} & \frac{r_{1m} \cdot \cos \delta_a}{\Delta_3} \\ L_R \cdot X_{cm} & Y_{cm} & Z_{cm} \\ L_R \cdot (r_{2m} - r_{2h}) \cdot \tan(\Phi_2) & r_{2m} \cdot \sin \delta_d & r_{2m} \cdot \cos \delta_d \end{bmatrix} \]  
\[ T = \begin{bmatrix} 0 & -r_{1t} \cdot \sin \delta_a & r_{1t} \cdot \cos \delta_a \\ \frac{X_{ct}}{\Delta_3} & -r_{1t} \cdot \frac{\sin \delta_a}{\Delta_3} & \frac{r_{1t} \cdot \cos \delta_a}{\Delta_3} \\ L_R \cdot X_{ct} & Y_{ct} & Z_{ct} \\ L_R \cdot (r_{2t} - r_{2h}) \cdot \tan(\Phi_2) & r_{2t} \cdot \sin \delta_d & r_{2t} \cdot \cos \delta_d \end{bmatrix} \]
Similarly, point ‘d’ uses eqn. (2.32) defined by \( \delta \) and \( r_2 \) values. \( \Delta_2 \) is the hub X location of point ‘c’. It is 80 - 90% of \( L_R \). The X locations for ‘c’ are obtained from eqn. (2.33-2.35). Because the exit is oriented at a meridional angle, the definitions of Y and Z location for ‘c’ are given in eqn. (2.36-2.40). The two intermediate point locations are controlled using \( \Delta_1 \) and \( \Delta_2 \) to adjust the blade curvature. A low camber variation in the tip section helps to avoid flow separation. \( \Phi_{2i} \) are the meridional angles at the hub, mean and tip according to the rotor exit and vane-less spacing interaction. Using these point definitions, three camber lines for H, M, and T were generated. The chosen airfoil thickness was distributed along the Bezier curve camber perpendicular to the local camber in order to obtain the rotor hub, mean and tip airfoil sections. The slope variation of the camber curve was obtained from the Bezier X, Y, and Z coordinates. The airfoil thickness and axial locations were non-dimensionalized and then converted to the rotor dimensions based on \( L_R \). A NACA 65-206 series airfoil was chosen for this mixed-flow rotor, as given Appendix 3. Due to extreme stretching, the original behavior of the airfoil was difficult to preserve. In this case, the same airfoil with thickness percentages of 75 (hub), 50 (mean) and 35% (tip) were used to obtain the three-dimensional mixed-flow rotor. Appendix 2 provides the values corresponding to this rotor design. Figure 2-6 describes the 3D mixed-flow rotor generated with the given values.
Figure 2-6 Final 3D solid model of the mixed-flow rotor after the airfoil addition to the camber sheet.

2.4 Tandem stator strategy and geometry definition

The preliminary level of design involved tandem stator row definition in a two-dimensional plane. $\alpha_{3m}$ was considered as the inlet flow angle. Exit flow was oriented in the axial direction. A 3rd order Bezier Curve using the inlet ($\alpha_1$) and exit ($\alpha_2$) stator angles defined the camber line. The percentage axial length of each blade row was chosen to determine the axial length of blade row 1 ($L_{B1}$). $X_{LB1}$ is the axial location of the leading edge of $B_1$. Three location control points, $\Delta_1$, $\Delta_2$, and $\Delta_3$, were used to manually define the mean camber. Matrix $P_{b1}$ using eqn. (2.44) contains the Bezier curve defining the points shown in Figure 2-7. It is important to remember that the first blade row’s main purpose is to diffuse the supersonic flow using oblique and passage shocks effectively. Thus, the blade turning angle ($\alpha_1 - \alpha_2$) was limited to 7 to 10° to avoid excessive shock boundary layer separation losses.
\[
P_{B1} = \begin{bmatrix}
X_{LB1} & 0 & 0 \\
X_{LB1} + \Delta_1 & -\tan(\alpha_1) \Delta_4 & 0 \\
X_{LB1} + L_{B1} - \Delta_2 & -\Delta_3 + \Delta_2 \tan(\alpha_2) & 0 \\
X_{LB1} + L_{B1} & -\Delta_3 & 0
\end{bmatrix}
\]  
(2.44)

\[C_{B1} = B \cdot P_{B1}\]  
where B is the 3rd order Bernstein polynomial.

A 3rd order polynomial function, eqn. (2.45), defined the airfoil thickness. The inlet \((\lambda_i)\) and exit \((\lambda_e)\) wedge angles had to be specified based on supersonic inlet Mach number \((M_3)\) and oblique shock positioning. \(\Delta_1\) and \(\Delta_2\) are the two control points that adjust the thickness distribution based on the design. \(P\) is the airfoil thickness distribution function shown in Figure 2-8, which is wrapped on the camber line to obtain the stator blade airfoils. Coefficients are defined by eqn. (2.46-2.48). Thickness \((P)\) was distributed perpendicular to the camber curve’s slope. Alternative thickness distributions, \(\tau_1\) and \(\tau_2\), were used on the pressure and suction side depending on the shock structure location.
\[ P = U_1 r^3 + U_2 r^2 + U_3 r + U_4, \quad \text{where} \quad r = 0:100 \]  

\[ Y \begin{bmatrix} 0 \\ X_2 \\ X_3 \\ X_4 \end{bmatrix} = \begin{bmatrix} Y_1 \\ Y_2 \\ Y_3 \\ Y_4 \end{bmatrix}, \quad X_4 = 100; \quad Y_4 = 0 \]  

\[ F = \begin{bmatrix} 0 & 0 & 0 & 1 \\ X_2 & X_2 & X_2 & 1 \\ X_3 & X_3 & X_3 & 1 \\ X_4 & X_4 & X_4 & 1 \end{bmatrix} \]  

\[ U_1 = F^{-1} [Y_A]^T \quad \text{where} \quad Y_A = [Y_1 \ Y_2 \ Y_3 \ Y_4] \]  

Based on a CFD analysis that is described in Chapter 5, flow was well guided by the first blade row, hence, the leading edge of the second blade row was defined as a wedge-shaped profile. For the subsonic profile, the trailing edge radius was defined in the polynomial thickness distribution for blade 2. A similar procedure was then repeated for blade row 2 camber line construction. The thickness distribution for the suction and pressure side were separately determined by a 3rd order polynomial. ‘S’ is the blade pitch and ‘N’ is the blade count for each row. The leading edge was
shifted by a distance ‘t’ in the pitch-wise direction away from the pressure side of blade 1. The ‘t/S’ ratio was maintained close to 20% based on [16]. Since this blade row guides a subsonic profile, it is associated with a large flow turning angle ($\alpha_3 - \alpha_4$) of 37 - 40°. Throat and exit area signify the amount of diffusion by this blade row. $\alpha_3$ is equal to $\alpha_2$, $\alpha_4$ is 0°, as the exit flow is oriented with the axial direction. The mean camber line is defined by a 4th order polynomial with three control points: $\Delta_1$, $\Delta_2$, and $\Delta_3$. $\Delta_1$ was defined based on the tangent $\alpha_3$ and t. $\Delta_3$ has the same y-value based on the trailing edge location to maintain $\alpha_4$ as 0°. The axial location of the points $\Delta_1$, $\Delta_3$, and $\Delta_2$ are fixed based on the desired camber line curvature.

![Figure 2-8 Polynomial function ‘P’ distributing airfoil thickness.](image)

The stator hub curvature is defined in the meridional plane using a 4th order Bezier curve starting from the rotor exit. Therefore, $\Phi_{3h} = \Phi_{2h}$, the rotor hub exit meridional angle and $\Phi_{4h} = 0°$. The casing inlet and outlet meridional angles ($\Phi_{3t}$ and $\Phi_{4t}$) are defined in the same way. Five control points were used to define each of the hub and casing Bezier curves. $\Delta_{1h}$ was obtained from the rotor hub exit meridional point $r_{2h}$. The axial location of $\Delta_{3h}$ is fixed based on the 2D cascade design
blade 2 exit location, given by $L_S$. The Y location of $\Delta_5h$ is based on the $r_{4h}$ value. The Y location for $\Delta_{2h}$ was obtained from $r_{3h}$. Its X location is equal to $L_{VS}$. The Y value of $\Delta_{4h}$ is equal to $\Delta_{5h}$. $\Delta_{2h}$ and $\Delta_{4h}$ were defined based on the $\Phi_{3h}$ and $\Phi_{4h}$ angles. The $\Delta_{3h}$ point was iteratively varied to maintain a smooth profile. The purpose of the X and Y values of $\Delta_{2h}$, $\Delta_{3h}$, and $\Delta_{4h}$ is to obtain the desired camber line curvature. It is an iterative procedure. A similar procedure was followed to obtain $\Delta_{1t}^{5t}$ points to define the casing profile.

Finally, **Figure 2-9** defines the three-dimensional stator with the 2D stator design being superimposed on the hub and casing profiles. The Y locations of both blades in the 2D cascade were converted to radial ($r$) and angular ($\theta$) values in the pitch-wise direction with the same X (axial location) values. **Appendix 4** provides the design values corresponding to this stator configuration.

![Final 3D solid model of a supersonic tandem stator](image)
Chapter 3 CFD solver validation and mesh dependence study

This chapter illustrates the numerical solver details and corresponding finite volume discretization analysis for the mixed-flow compressor.

3.1 Computational model

The computational effort utilized a finite volume method to transform the Navier-Stokes equation-based mathematical model into a system of algebraic equations. The algebraic multigrid method was used in the general-purpose fluid dynamics solver STAR-CCM+ to provide an approximate numerical solution. The equations were discretized in time and an implicit time-integration scheme was used to obtain a quasi-steady state solution. The convective fluxes were discretized using a second-order upwind scheme. A steady Reynolds-averaged Navier-Stokes (RANS) approach in three dimensions with the one-equation Spalart-Allmaras turbulence model [21] was chosen. A performance assessment comparison using one and two-equation turbulence models on rotor alone simulation is presented in Table 3-2 with corresponding boundary condition in Table 3-5. On that basis, we proceed with Spalart-Allmaras model in the study as it accurately predicts flow features and is relatively time efficient. Ideal gas conditions with air as the fluid were selected. The assumption of a continuum was valid since the Mach number was less than three and the Knudsen number was less than one throughout the compressor stage. The models described herein were used in the current fluid dynamics solver for high-speed flows [22].

A ‘coupled flow and energy model’ combined the conservation equations for mass, momentum and energy simultaneously using a pseudo-time marching approach. The velocity field was obtained from momentum equations, pressure was obtained from the continuity equation and density was obtained from the equation of state. A time marching technique converted the unsteady derivative terms to a steady approach, where those terms were driven to zero. The time step for an individual cell was evaluated based on stability constraints. This model had a robust nature for
compressible fluid flow and the CPU time scaled with cell count providing good convergence with a refined mesh. A Courant number (CFL) of five was used throughout the simulations. The hybrid ‘all Y+ wall treatment’ either resolved the viscous sublayer in a fine mesh or derived wall functions from equilibrium turbulent boundary layer theory for coarse meshes. A grid sequence initialization condition was used, which computed the domain from an approximate inviscid solution. A series of coarse meshes were generated to interpolate the solution onto the next finer mesh toward convergence. This provides faster and more robust convergence for the flow solution, due to the probability of large CFL numbers.

All of the mixed-flow compressor simulations in this study used a single “stage” passage to minimize the computational cost and time requirement by incorporating periodic boundary conditions into the mid passage surfaces. The inlet was 100 mm upstream of the leading-edge of the rotor blade to study the leading-edge interaction. The outlet was 10 mm downstream of the second stator blade. All of the surfaces in the rotor domain, except the casing, rotate. The fluid domain is described in Figure 3-1.

<table>
<thead>
<tr>
<th>Table 3-1 Stage boundary conditions from the mean line analysis</th>
</tr>
</thead>
<tbody>
<tr>
<td>Reference Pressure($P_{o1}$), Pa</td>
</tr>
<tr>
<td>Inlet $P/P_{o1}$</td>
</tr>
<tr>
<td>Inlet $T_0$, K</td>
</tr>
<tr>
<td>Inlet $v_T$</td>
</tr>
<tr>
<td>Exit $T_s$, K</td>
</tr>
<tr>
<td>Exit $P_{avg}/P_{o1}$</td>
</tr>
<tr>
<td>Exit $v_T$</td>
</tr>
</tbody>
</table>
A rotor stagnation inlet condition was used for all simulations. The incoming flow direction was normal to the inlet surface. A mixing-plane rotor-to-diffuser interface was used. The mixing plane efficiently handles the rotor-stator pitch differences in order to transfer circumferentially averaged quantities of mass, momentum and energy with uniform radial thickness. This approach significantly reduces computational time when compared to a transient simulation for turbomachinery [23]. The mid-passage surfaces utilized periodic boundaries. At the outlet of the “rotor-alone” simulations, the back pressure at the hub surface was imposed by using a radial equilibrium condition. For the stage simulations, average back pressure at the stator outlet was imposed. The values for the design point obtained from the mean-line design code explained in Chapter 1 are given in Table 3-1.
Comparative performance assessment of rotor alone simulation for different turbulence closure models

<table>
<thead>
<tr>
<th>Turbulence model</th>
<th>$\Pi_{TT}$</th>
<th>$\eta_{IS}$, %</th>
<th>$\dot{m}$, Kg/s</th>
</tr>
</thead>
<tbody>
<tr>
<td>Spalart-Allmaras</td>
<td>7.15</td>
<td>86.87</td>
<td>3.00</td>
</tr>
<tr>
<td>K-omega (SST)</td>
<td>7.16</td>
<td>86.80</td>
<td>2.92</td>
</tr>
</tbody>
</table>

3.2 Assessment of the computational model using existing compressor rotor and diffuser data

The computational system used in this study is first evaluated for its ability to reasonably predict the compressor performance using existing well-established experimental data sets. Since, main goal of this paper is to take all three dimensionalities of the stage, viscous flow effects and compressibility including the shock wave systems into account, the authors felt that a comprehensive evaluation against existing experimental data was essential. Thus, results obtained from current steady state RANS based computational fluid dynamics solver [22] are compared to the experimental data for NASA Rotor 37 [20] and RWTH Aachen [10] supersonic tandem diffuser.

Verification 1: NASA rotor 37 experiment

The experimental test case of the NASA Lewis/Glenn transonic rotor 37 from Suder and Reid [20, 24] was computed using the CFD model in this study. Rotor 37 is a low aspect ratio inlet stage for an eight-stage core compressor with a 20:1 total pressure ratio [24], and it is an ideal test case for code verification. The analysis domain is shown in Figure 3-2. Extensive data representing performance and flow field variables can be found in [20], which has been used for CFD validation. Boundary conditions were obtained from Dunham [25] for the 0.98 $\dot{m}/\dot{m}_{\text{choke}}$ condition. A stagnation pressure and temperature inlet condition was used along with an average static pressure
outlet. The mesh discretization is shown in Figure 3-3. A comparison of the total pressure ratio and total temperature ratio profiles are described in Figures 3-4 and 3-5, respectively. Solid circular symbols represent the NASA rotor 37 experiments [20, 24].

Figure 3-2 Geometry of NASA transonic rotor 37 experiments and computational domain.

Figure 3-3 NASA rotor 37 mesh description: (a) leading edge resolution, (b) 70% span
<table>
<thead>
<tr>
<th>Investigators</th>
<th>Analysis type</th>
</tr>
</thead>
<tbody>
<tr>
<td>Suder [20]</td>
<td>Experiment</td>
</tr>
<tr>
<td>Sadagopan and Camci*</td>
<td>RANS, no hub leakage</td>
</tr>
<tr>
<td>Sadagopan and Camci*</td>
<td>RANS, with hub leakage</td>
</tr>
</tbody>
</table>

*Present computation*

**3-4 (a)**

- Hah, C. [26]           : LES
- Ameri, A. [27]          : RANS
- Boretti, A. [28]        : RANS
- Bruna and Turner [29]   : RANS

**3-4 (b)**

- Joo, J. et al. [30]     : RANS, trip
- Joo, J. et al. [30]     : LES₀
- Joo, J. et al. [30]     : LESₙₐ₀
- Seshadri, P. et al. [31]: RANS

**Figure 3-4 (a) and (b)** Total pressure ratio distribution in the span-wise direction at station 4 for 

\[ \frac{\dot{m}}{\dot{m}_{\text{choke}}} = 0.98 \] (experiments are from Suder [20])
Figure 3-4 (a) and (b) compare the present stage exit total pressure computations against the NASA transonic rotor 37 experiment and seven other prediction attempts that were available in the open literature. The current study’s predictions used the Spalart-Allmaras model [21] to account for turbulent flow effects. These predictions, as represented by a star symbol, were in excellent agreement with the measured data, which is denoted by solid circular symbols in the region above 75% span. The agreement was reasonable between 30% span and 75% span when compared to all other predictive efforts. Most of the predictions, including the current computational effort below 30% span, were not as successful as the predictions above the mid-span.

The CFD simulation over-predicted the total pressure deficit occurring at 0-30% span, similar to all other predictive efforts from the literature. Hence, further investigation of a possible hub leakage flow effect was undertaken, based on the experimental work of Shabbir et al [32]. A hub leakage flow of 0.34% of the inlet mass flow rate was computationally examined. The hub leakage flow was assumed to have the same total temperature as that of the free stream, with 1% turbulence intensity. The leakage flow case is shown in Figures 3-4 (a) and (b) by red stars. Current computations with no hub leakage are represented by blue stars. Shabbir et al.’s work was unfortunately not conclusive in terms of accounting for the total pressure over-predictions near the hub region below 30% span. The leakage rate was varied and the influence of the hub leakage fluid did not resolve the predictive anomaly in the hub region. The current effort with hub leakage resulted in a very similar total pressure profile in the span-wise direction when compared to the non-leakage case, as shown in Figures 3-4 (a) and (b). Only a slight difference near the hub was observed between the non-leakage and leakage cases.

The authors also investigated rotational effects of the entire hub. However, based on those results two reasons were inferred to negate this possibility. Firstly, results did not vary compared to the rotating blade section results in terms of over-prediction of total pressure below the 30% span. Secondly, rotating the entire hub would induce more energy into the flow through shearing
effects which will add on to total pressure over-prediction. Moreover, it is important to note that the original experiment did not have entire hub rotating.

The most effective simulation of the NASA transonic rotor 37 was performed by Bruna and Turner [29] in the near hub region. CFD simulations were obtained using an isothermal thermal boundary condition. All past simulations in Figures 3-4 and 3-5 used an “adiabatic wall” boundary condition. Bruna and Turner also repeated the simulation [24] using an adiabatic boundary condition. The red solid line for the isothermal wall boundary condition in Figure 3-5 matched the experimental data remarkably well. This is the best prediction of the total pressure distribution for the rotor 37 in the literature. Although total pressure, temperature, and stage efficiency matched experimental data very well, Bruna and Turner [29] observed that the real compressor rig experiments would not be isothermal at the casing. A proper analysis would require a conjugate heat transfer approach. In the simulations, the entire hub was rotating. The cavities in the hub region also needed to be modeled for better prediction. Bruna and Turner [29] reported that the computed efficiency data using an isothermal boundary condition matched the experimental data much better than the adiabatic simulation. The efficiency difference between the isothermal and adiabatic solutions was 1%. Figure 3-5 shows Bruna and Turner’s total temperature prediction at the stage exit. The profiles of total temperature with the isothermal boundary condition matched the data near the casing very well. All past adiabatic simulations had an overshoot that has been consistently part of any CFD result compared with this data set. The total temperature predictions from the current work reasonably compared to the experimental data.

The present computational verification effort for the CFD system used in this study resulted in stage exit total pressure and temperature predictions that were better or very similar to most of the past rotor 37 computations. The near hub region below 30% span was still an area of uncertainty, in terms of the thermal boundary conditions used. The current effort proceeded with an adiabatic
thermal boundary condition. The implementation of a conjugate heat transfer model is likely to be a topic of future investigation. It is the current authors’ observation that the present computational approach can be effectively utilized in the prediction of transonic/supersonic compressor rotor flow fields in the development of better mixed-flow compressors.

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Investigators</th>
<th>Analysis type</th>
</tr>
</thead>
<tbody>
<tr>
<td>●</td>
<td>Suder [20]</td>
<td>Experiment</td>
</tr>
<tr>
<td>★</td>
<td>Sadagopan and Camci*</td>
<td>RANS, no hub leakage</td>
</tr>
<tr>
<td>★★</td>
<td>Sadagopan and Camci*</td>
<td>RANS, with hub leakage</td>
</tr>
<tr>
<td><em>Present computation</em></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Investigators</th>
<th>Analysis type</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Ameri, A. [27]</td>
<td>RANS</td>
</tr>
<tr>
<td></td>
<td>Bruna and Turner [29]</td>
<td>RANS, adiabatic</td>
</tr>
<tr>
<td></td>
<td>Bruna and Turner [29]</td>
<td>RANS, isothermal</td>
</tr>
</tbody>
</table>

**Figure 3-5** Total temperature ratio in the spanwise direction at station 4 for $\frac{m}{m_{choke}} = 0.98$ (experiments are from Suder [20]).
Verification 2: RWTH Aachen supersonic tandem stator experiment

The current computational method’s ability to resolve the aerothermal characteristics of a supersonic diffuser containing shock wave systems was assessed using experimental data from the RWTH Aachen supersonic tandem diffuser [10]. This is a two-dimensional simulation. The supersonic tandem diffuser operating conditions and the boundary conditions from Elmendorf et al. [10] are given in Table 3-3. Figure 3-6 shows the geometry of the tandem diffuser in a linear cascade arrangement. The blade ‘1’ inlet region had a straight contour to avoid excessive flow acceleration and was followed by a locally divergent region to position the strong shock stabilized by back pressure. Blade ‘2’ was introduced to keep the subsonic flow diffusion at a moderate level.

Table 3-3 RWTH Aachen tandem diffuser boundary conditions

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Reference Pressure ($P_{o1}$), Pa</td>
<td>126300.0</td>
</tr>
<tr>
<td>Inlet Mach number</td>
<td>1.478</td>
</tr>
<tr>
<td>Inlet $P/P_{o1}$</td>
<td>1.836</td>
</tr>
<tr>
<td>Inlet $T_o$, K</td>
<td>559.12</td>
</tr>
<tr>
<td>Exit Mach number</td>
<td>0.368</td>
</tr>
<tr>
<td>Exit $P/P_{o1}$</td>
<td>4.30</td>
</tr>
<tr>
<td>$\eta_{IS}$ (from present computation)</td>
<td>72.09%</td>
</tr>
</tbody>
</table>

The current computations compared the computed blade ‘1’ airfoil surface static pressure distribution ($P/P_{o1}$) to the RWTH experiments in Figure 3-6. Multiple static pressure jumps are visible in the figure, which correspond to shocks. The inlet supersonic flow generated a weak oblique shock at the leading-edge of blade ‘1.’ The flow slightly accelerated on the pressure and suction side of blade ‘1.’ The pressure side leg of the oblique shock impinged on the suction side at ‘B.’ A convergent passage led to a strong shock from ‘A’ to ‘C,’ which had a higher strength on the pressure side. A complex shock structure was formed near the leading-edge of blade ‘2’. The convex suction side of blade ‘2’ generated extreme acceleration of the flow leading to another
strong shock extending into the concave pressure side of blade ‘1,’ as shown in location ‘D.’ There was a slight discrepancy in the shock location and magnitude of ‘D’ when compared to the experimental data. This was due to a subtle geometrical variation in blade ‘2,’ which could not be captured with high enough resolution using the current grid system from the published information in [10]. The strong passage shock at ‘D’ increased the pressure tremendously. The current computations predicted the isentropic efficiency of this supersonic diffuser as 72.09%. There was very good agreement between the measured data and current computations between $X/L_{B1} = 0.4$ and 1.0 in Blade ‘1.’ Even though this region was located just downstream of a very complex set of shock waves, the current predictions effectively captured the measured diffuser pressure distribution. There was no experimental pressure data available for blade ‘2’ after $X/L_{B1}$, as shown in Figure 3-6. Although there were a limited number of wall static pressure measurements before $X/L_{B1} = 0.4$, the current predictions followed the measured wall static pressure points closely. This area was dominated by a weak oblique shock and a stronger passage shock between ‘A’ and ‘C.’ The complex shock wave interaction prediction with the boundary layers and other flow features followed the experimental trends very well as shown in Figure 3-6. It should be noted that the experimental data will have a certain amount of uncertainty that was not documented in [10].

The computational assessments presented in this chapter showcased the nature of CFD-based performance predictions against two well-established high-speed flow cases existing in the open literature. The current computational model was able to capture the general features of the three-dimensional, viscous, and turbulent flow effects in a compressible environment very similar to the mixed-flow compressor. Thus, this computational model was used to analyze the performance of the current mixed-flow compressor stage.
Figure 3-6 CFD vs experimental results for the RWTH Aachen tandem stator cascade.
3.3 Mesh dependency study

The mesh of a finite volume-based RANS computation influences the quality of viscous flow predictions; therefore, it is essential to quantify the mesh dependency of the computations. An automated meshing tool built into the current fluid dynamics solver [22] was used to generate unstructured polyhedral cells to discretize the fluid domain for the current mixed-flow compressor design analysis. The base cell size on the fluid domain surfaces was specified to construct the polyhedral mesh. The cells were further discretized into orthogonal prismatic cells next to the wall surfaces to improve flow resolution inside boundary layers. Typical sectional views of representative grid zones are shown in Figure 3-7 and the mesh property inputs are presented in Table 3-6. A rotor fluid-domain simulation was used for undertaking the mesh dependence study. Three different meshes (coarse, moderate, and fine) were compared. The same set of boundary conditions was used for all three computations, as shown in Table 3-5. All meshes, except the coarse mesh, had prismatic cells that were sufficiently small to maintain $Y^+ < 1$ in all cells adjacent to the wall. The base cell size on the casing, hub and blade walls were reduced to increase passage resolution.

![Figure 3-7 Polyhedral mesh detail at the moderate level with boundary layer cells:](image)

a. Rotor leading-edge,  b. Stator mid-chord sectional plane.
Rotor blade static pressure ($C_P$) distribution at 65% span plotted in Figure 3-8 clearly shows the influence of the mesh size on providing passage shock resolution. The leading-edge acceleration on both the pressure and the suction side is depicted by a $C_P$ drop near $X/L_R=0$. Expansion fans generated on the suction side (SS) were observable for all three mesh resolutions between $X/L_R=0$ and 0.05. Further downstream on the SS, a weak shock influence is shown by the gradual increase in $C_P$ observed before $X/L_R = 0.10$. This shock on the SS generated a separation bubble due to the shock-boundary layer interaction. The rotor passage shock observed in the coarse mesh was wider, as it was spread over relatively larger cells.

The coarse mesh computation showed that the passage shock ended near $X/L_R=0.37$. However, such a wide area of shock influence was only because of the coarse mesh. This could easily be attributed to the absence of boundary layer resolution in the coarse mesh configuration. This apparent transition in shock location was observed directly as a mass flow rate increment, as shown in Table 3-4. The inlet Mach number shown in Figure 3-9 also supports this case. The inlet Mach number for the coarse grid was very influenced by the grid structure not resolving the boundary layers properly. No precaution was taken to resolve the boundary layers in the coarse mesh representation of the rotor passage. Beyond $X/L_R = 0.37$ on the suction side, the $C_P$ trend on the rotor airfoil was similar for all three mesh resolutions. On the pressure side, the shock influence due to the coarse grid disappeared after $X/L_R = 0.15$. The sudden drop in $C_P$ on the SS beyond $X/L_R = 0.9$ was due to the casing region’s separated flow interaction near the rotor exit.

The stagnation pressures at the outlet, shown in Figure 3-10, were within $\Pi = \pm 0.15$ for most span-wise locations of the moderate and fine meshes. The coarse mesh contained shock-influenced areas that were unrealistically spread over larger zones. The predicted efficiency for the coarse mesh was 3% higher than the other two cases, as seen in Table 3-4 and Figure 3-11.
Table 3-4 clearly specifies that performance evaluation of both moderate and fine mesh is very close to each other. The former case being computationally time efficient, authors infer to use moderate mesh throughout the mixed-flow compressor computational effort.

Table 3-4 A comparative assessment of mesh dependency

<table>
<thead>
<tr>
<th>Mesh</th>
<th>No. of cells</th>
<th>BL</th>
<th>( \dot{m}, \text{kg/s} )</th>
<th>( \eta_{IS}, % )</th>
<th>( \Pi_{TT} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse</td>
<td>0.1 million</td>
<td>No</td>
<td>3.129</td>
<td>89.27</td>
<td>7.13</td>
</tr>
<tr>
<td>Moderate</td>
<td>3.1 million</td>
<td>Yes</td>
<td>3.000</td>
<td>86.87</td>
<td>7.15</td>
</tr>
<tr>
<td>Fine</td>
<td>12.0 million</td>
<td>Yes</td>
<td>2.960</td>
<td>86.49</td>
<td>7.15</td>
</tr>
</tbody>
</table>

Figure 3-8 \( C_p \) distribution at 65% rotor blade span for the three mesh densities
Figure 3-9 Rotor inlet Mach number, $M_1$, for coarse, moderate, and fine meshes (circumferentially mass averaged).

Figure 3-10 Stagnation pressure, $\Pi_{TT} = P_{o2}/P_{o1}$, for coarse, moderate, and fine meshes at the rotor exit, 150 mm (circumferentially averaged).
Figure 3-11 Isentropic efficiency at rotor exit for coarse, moderate, and fine meshes at the rotor exit, 150 mm (circumferentially averaged).

Table 3-5 Rotor CFD simulation boundary conditions

<table>
<thead>
<tr>
<th>Boundary Condition</th>
<th>Coarse</th>
<th>Moderate</th>
<th>Fine</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet stagnation pressure (Pa)</td>
<td>33113.0</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Inlet total temperature (K)</td>
<td>247.7842</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Inlet turbulence viscosity ratio, $v_T$</td>
<td>10.0</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Exit hub static pressure (Pa)</td>
<td>81930.0</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Exit static temperature (K)</td>
<td>320.98</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Exit turbulence viscosity ratio, $v_T$</td>
<td>10.0</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 3-6 Mesh input data comparison between coarse, moderate and fine cases

<table>
<thead>
<tr>
<th>Mesh Property definition</th>
<th>Coarse</th>
<th>Moderate</th>
<th>Fine</th>
</tr>
</thead>
<tbody>
<tr>
<td>Polyhedral mesh</td>
<td>Yes</td>
<td>Yes</td>
<td>Yes</td>
</tr>
<tr>
<td>Prism Layer (Boundary layer mesh)</td>
<td>No</td>
<td>Yes</td>
<td>Yes</td>
</tr>
<tr>
<td>Base cell size - rest of the domain (mm)</td>
<td>3</td>
<td>2.1</td>
<td>0</td>
</tr>
<tr>
<td>Base cell size - wall (mm)</td>
<td>0.99</td>
<td>0.99</td>
<td>0.3-0.6</td>
</tr>
<tr>
<td>No. of prism layers</td>
<td>N/A</td>
<td>25</td>
<td>25</td>
</tr>
<tr>
<td>Total prism layer thickness (mm)</td>
<td>N/A</td>
<td>0.99</td>
<td>0.99</td>
</tr>
<tr>
<td>Stretching factor (Geometric Progression)</td>
<td>N/A</td>
<td>1.2</td>
<td>1.2</td>
</tr>
<tr>
<td>Wall proximity distance (mm)</td>
<td>N/A</td>
<td>0.001</td>
<td>0.001</td>
</tr>
<tr>
<td>Total no. of cells (in millions)</td>
<td>0.1</td>
<td>3.1</td>
<td>12</td>
</tr>
</tbody>
</table>
Chapter 4

Stage design evaluation

4.1 Rotor design consideration

The one-dimensional mean-line design approach from Chapter 2 was used for this parametric study. The design variables were varied to provide better visualization and clarity for the designer. The primary goal was to obtain a very high-pressure ratio of greater than 6.0 within an exit external radius ($r_2$) of 180 mm. Based on the literature survey and additional computational efforts, the majority of the rotor losses are due to the casing shock-boundary layer interaction and the blade leading edge tip region shocks. Secondary flow feature vortices and blade viscous losses were the other two loss-generating components considered in this mixed-flow rotor. To account for all losses, the rotor isentropic efficiency was initially chosen to be 85% for a relative inlet tip Mach number of 1.4. Parameters that were analyzed throughout this exercise are given in Table 4-1. This was only an initial evaluation effort based on an assumed rotor isentropic efficiency of 85%.

Table 4-1 Rotor mean-line parametric study variables

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\eta_{IS}$</td>
<td>70-100%</td>
</tr>
<tr>
<td>$M_2$</td>
<td>0.5-1.5</td>
</tr>
<tr>
<td>$M_{2rel}$</td>
<td>0.7-1.2</td>
</tr>
<tr>
<td>$r_{2m}$</td>
<td>120-220 mm</td>
</tr>
<tr>
<td>$m$</td>
<td>3.5 Kg/s</td>
</tr>
<tr>
<td>$\Omega$</td>
<td>28500 RPM</td>
</tr>
<tr>
<td>$M_1$</td>
<td>0.7</td>
</tr>
</tbody>
</table>
Figure 4-1 Rotor mean line evaluation study plots
Rotor inlet Mach no. \( \text{M}_1 \) is 0.7 and inlet relative Mach no. increases from hub to tip with a value of \( \text{M}_{1\text{rel}} \) as 1.4 at the tip section. Based on Figure 4-1(a) if the rotor exit mean radius (\( r_{2m} \)) and the exit Mach number increase (\( \text{M}_2 \)), the rotor pressure ratio (\( \Pi_{TT} \)) increases. A higher \( \text{M}_2 \) for the same exit radius, efficiency and passage diffusion corresponds to higher pressure ratio. Since the stator inlet Mach number (\( \text{M}_3 \)) is limited to be in the range of 1.1–1.3 for a good stator efficiency \( \text{M}_2 \) is chosen to be 1.3. \( r_{2m} \) is chosen as 175mm due to the stage external diameter constraint of 200mm to provide additional radial gap for the stator section.

Lower \( \text{M}_{2\text{rel}} \) denotes more diffusion in the rotor passage. Higher the diffusion in the passage denotes higher amount of work done by the rotor on the fluid which is shown by the high rotor pressure ratio. From Figure 4-1(b) for the same \( \text{M}_2 \) and efficiency value for pressure ratio of 6, \( r_{2m} \) and \( \text{M}_{2\text{rel}} \) need to decrease. However it’s not feasible to reduce \( \text{M}_{2\text{rel}} \) due to shock losses and corresponding boundary layer growth due to that adverse pressure gradient.

For the same pressure ratio 6 and blade tangential velocity, a lower \( \text{M}_{2\text{rel}} \) corresponds to a lower \( \text{M}_2 \) as seen in Figure 4-1(c). A higher exit Mach number \( \text{M}_2 \) leads to increase in \( \text{M}_{2\text{rel}} \) and hence increasing efficiency as seen in Figure 4-1(d). Obtaining a lower \( \text{M}_{2\text{rel}} \) at the same \( \text{M}_2 \) decreases the rotor efficiency as seen in Figure 4-1(d) and it is very difficult to design a passage which can effectively generate high pressure ratio and perform this diffusion as perceived in Figure 4-1(b).

In conclusion, a higher rotor exit Mach number, higher exit mean radius and low rotor exit relative Mach number is desired to obtain a high-pressure ratio. To increase the isentropic efficiency for the same pressure ratio \( \text{M}_2 \) and \( \text{M}_{2\text{rel}} \) should increase. Rotor back pressure increase corresponds to rotor pressure ratio increment.

The variation of exit flow angle (\( \alpha_{2m} \)) with respect to the axial direction is shown in Figure 4-2. The minimum \( \alpha_{2m} \) is desired, as it reduces the necessity of the stator to turn the flow back to the axial direction. A lower \( \text{M}_2 \) corresponds to a higher \( \alpha_{2m} \) which is not beneficial, as it generates a smaller rotor pressure ratio. The minima for \( \alpha_{2m} \) is found within the \( \text{M}_{2m} \) range of 1.1.
to 1.3 with $\Pi_{TT}$ from 4.5 to 7. A higher $M_{2rel}$ provides a lower $\alpha_{2m}$ and a correspondingly lower $\Pi_{TT}$.

Blade exit angle ($\beta_{2m}$) variation with $M_{2m}$ is shown in Figure 4-3. $\beta_{2m}$ monotonically decreases with increasing $M_{2m}$. A lower $\beta_{2m}$ is beneficial, as less turning in the rotor blade would avoid flow separation. An inflection point was observed at $M_{2m} = 1$ where higher subsonic diffusion in the rotor relative frame required lower $\beta_{2m}$ for $M_{2m} > 1$, and vice versa. As concluded in the previous inferences, higher diffusion in the rotor frame corresponds to an increase in $\Pi_{TT}$.

**Figure 4-2** Rotor flow exit angle variation with $M_{2}$
Figure 4-3 Rotor blade exit angle variation with $M_2$

The mean-line code predicted many of the rotor parameters, but additional performance parameters needed to be analyzed to gather the comprehensive knowledge required for the mixed-flow rotor design process. Hence, a computational analysis with a 3D RANS code was conducted to evaluate the rotor exit meridional angle, rotor axial length, number of blades, and other parameters as shown in Table 4-2 that provide the optimum rotor performance for the chosen design point. Details of this CFD analysis were given in Chapter 3.

Table 4-2 Mixed-flow rotor computational study cases

<table>
<thead>
<tr>
<th>Case</th>
<th>$\Phi_2$ (degrees)</th>
<th>$L_R$ (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>a</td>
<td>10</td>
<td>130</td>
</tr>
<tr>
<td>b</td>
<td>10</td>
<td>150</td>
</tr>
<tr>
<td>c</td>
<td>10</td>
<td>170</td>
</tr>
<tr>
<td>d</td>
<td>30</td>
<td>140</td>
</tr>
<tr>
<td>e</td>
<td>60</td>
<td>118</td>
</tr>
<tr>
<td>f</td>
<td>90</td>
<td>100</td>
</tr>
</tbody>
</table>
The **rotor meridional angle study** was conducted for a wide range of $\Phi_2$, from 0 - 90°. The same mean exit radius condition and $\Omega$ were maintained for cases b, d, e, and f to understand the changes in pressure ratio, efficiency, etc. It is well-established that a centrifugal compressor generates a higher pressure ratio compared to an axial design with some compromise on the efficiency. This study was primarily conducted to understand performance changes when a mixed-flow design was derived from a centrifugal rotor variant. The rotor geometrical variation is shown in **Figure 4-4**.

The axial length of the rotor increased with a reduction in the $\Phi_{2m}$ angle in order to instill the same amount of work into the fluid. Relative parametric variation in each of the configurations is shown in **Figure 4-5**. $\Pi_{TT}$ variation was within a 10% margin for all cases. Efficiency varied within a 3% margin. The result shows that it is possible to derive a mixed-flow rotor from a centrifugal variant, preserving the efficiency and pressure ratio. The increment in axial length for reducing $\Phi_{2m}$ imparts extra work on the fluid when compared to a higher $\Phi_{2m}$ with a lower axial length case. Additional axial length would add extra weight to the stage, but a reduced $\Phi_{2m}$ would lead to a smaller frontal area, hence a reduction in drag force. The pressure ratio and relative Mach number meridional contour plot of all four cases are described in **Figures 4-6 and 4-7**.
Figure 4-4 Rotor meridional exit angle, $\Phi_2$, variation in meridional view with $\Phi_1 = 0^\circ$.

<table>
<thead>
<tr>
<th>$\Phi_2$</th>
<th>$L_\beta$ (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>b</td>
<td>10\degree</td>
</tr>
<tr>
<td>d</td>
<td>30\degree</td>
</tr>
<tr>
<td>e</td>
<td>60\degree</td>
</tr>
<tr>
<td>f</td>
<td>90\degree</td>
</tr>
</tbody>
</table>

| 150      |
| 140      |
| 118      |
| 100      |

Figure 4-5 Relative parametric analysis for meridional exit angle study.

A lower back pressure led to a higher $M_2$ at the exit, as evident in Figure 4-5. Inlet Mach number, $M_1$, and mass flow rate decreased for the 90\degree case. Hub and casing contours played a very
important role in the pressure distribution throughout the passage. The low momentum zone was a minimum for the $90^\circ$ case, which is shown in the form of maximum efficiency. Higher pressure ratio growth was observed for the cases with higher passage diffusion. This effect was dominantly observed near the casing surface. The $\Phi_{2m} = 60^\circ$ case had a smaller region of low momentum, due to the casing separation, thus the diffusion was less than the other cases. This effect was visible, as the rotor pressure ratio was lower, but efficiency was higher. $\Phi_{2m} = 60^\circ$ and $90^\circ$ tended to behave in a beneficial manner with regard to casing separation. To obtain the desired goal of minimum frontal area, the $\Phi_{2m} = 10^\circ$ case was chosen. This conclusion matches the inference presented by Eisenlohr and Benfer [9], on mixed-flow rotor designs.

**Figure 4-6** Pressure ratio in the meridional view of six cases (a-f).
A **rotor axial length** \((L_R)\) study was conducted for three cases: (a) 130mm, (b) 150mm, and (c) 170mm, as shown in Figure 4-8. \(r_{zm}\), \(\Omega\), and computational boundary conditions were the same for all three cases. Figure 4-9 shows the non-dimensional parametric variation for the 18-bladed rotor design. \(\Pi_{TT}\) increased with increasing \(L_R\), as more blade surfaces perform more work on the fluid. Nearly a 9% increment in \(\Pi_{TT}\) was observed in the 40mm \(L_R\) increment. \(M_2\) followed the same trend as \(\Pi_{TT}\). Contrarily, this increment in \(\Pi_{TT}\) also added blade weight. The main criteria for selection were rotor weight and \(\eta_{IS}\). A rotor with a 150 mm length had the maximum efficiency out of the three cases. The 170mm case had a 4.5% decrement in efficiency, due to higher viscous losses with the extended blade surface. Rothalpy losses consistently increased in the blade passage, due to the increment in length. The 150 mm case was chosen for its weight and efficiency benefits. The pressure ratio and relative Mach number meridional contour plot of the three cases are described in Figure 4-6 and 4-7.
Figure 4-8 Rotor axial length, $L_R$, variation in meridional view,
where a) $L_R = 130$ mm, b) $L_R = 150$ mm, c) $L_R = 170$ mm with $\Phi_2 = 10^\circ$ and $\Phi_1 = 0^\circ$.

Figure 4-9 Relative parametric study for rotor axial length.
A brief discussion on rotor blade loading along the span is provided based on Xuanyu et al. [14]. For the hub line near the leading edge, centrifugal force cannot generate a high pressure ratio, as the meridional passage height does not increase abruptly. The hub line height increased on the posterior side, so any loading increment in this zone enhanced pressure-rising capability. Higher loading in the hub frontal area provided over-bending of the blade and led to flow separation on the suction side, causing a reduction in efficiency. Conversely, for the tip region, higher loading near the frontal area increased the rotor’s pressure-rising ability. Decreased loading in the tip posterior area controlled the inevitable suction side separation of the tip region. Therefore, increasing hub loading achieved higher performance with lower tip leakage loss and uniform flow at the exit. Increasing tip loading resulted in a high impeller pressure ratio.

![Figure 4-10](image)

**Figure 4-10** Relative parametric variation for the rotor pitch study.
The **Rotor pitch variation study** was conducted by varying the number of blades on the hub surface, as shown in Figure 4-10. An 18-bladed configuration imparted the highest energy into the fluid, as evident from the $M_2$ and $\Pi_{TT}$ values. All the parameters varied in a very narrow margin of 5 - 6%. Rotor mass consistently increased by 1.6% for each additional blade. A 10-bladed configuration delivered a 5% decrement in performance for a 16% reduction in weight. Engine performance would significantly improve, because of this dramatic reduction in weight.

Based on findings of *RajaKumar et al.* [17], for a mixed-flow rotor with subsonic rotor passage flow, the tip leakage flow interacts with the main passage flow contributing significantly to losses. Choked mass flow increased with higher inlet area, due to the increment in tip clearance. Surge occurred earlier, due to the separation tendency of the interaction between the cross flow and the main flow in the clearance space. Pressure ratio and performance decreased linearly with the impeller shroud gap. The pressure drop at the tip section was due to the leakage and main flow interaction. Mid-plane blade loading had no impact on tip clearance variation. The static pressure rise across the rotor passage reduced with increasing tip clearance. A constant tip clearance performed better than a variable tip clearance [15]. A classical jet wake flow observable in a centrifugal rotor was absent in a mixed-flow rotor. Mixed and centrifugal impellers had similar flow structures, but the mixed-flow design performed better than the centrifugal design, due to lower tip leakage loss with a more uniform flow field at the rotor outlet [14]. In this effort, all of the designed rotors had a constant tip clearance of 1mm. Tip leakage occurring from pressure to suction side in the initial tip section leading edge merged with casing separated low momentum fluid. In the mid chord section, certain streamlines travelled all the way from the pressure side hub region to suction side tip region through the gap, eventually mixing with the low momentum fluid.
Chapter 5 Aerodynamic analysis of the compressor stage

5.1 Rotor flow feature description

Rotor flow features at the design point were analyzed using the high-resolution 12 million cell mesh in a rotor simulation. The fluid-domain had a stagnation inlet 100 mm upstream of the leading edge and the average pressure outlet was 10 mm downstream of the rotor trailing edge. Boundary conditions are shown in Table 3-4. Dominant viscous flow and shock interaction-related structures triggering aerodynamic losses are highlighted in this description. Some light was also shed on the typical mixed supersonic rotor flow configurations.

The casing shock boundary layer separation ‘Cs’ was the most severe loss creating a low momentum zone which extended for at least 60% of the passage length, as seen in Figure 5-1. This flow separation was fully corroborated by the shock stabilization condition in eqn. (5.1) derived from the conservation of mass and energy [34]. The flow was stable in the zones where the area derivative (left-hand side of equation 5.1) exceeded the speed line derivative (right hand side of equation 5.1). This condition must be satisfied to obtain shock structures that are stable across the passage for different rotational rates, as given in Figure 5-2. The speed line derivative of equation 1 (dashed lines) exceeded the area derivative (solid line) in the X/L_R = 0.1 – 0.18 range. Meridional Mach number distribution clearly showed the separation at the X/L_R = 0.1 location. Further downstream, another unstable zone occurred at X/L_R = 0.70. Flow accelerated in the relative frame beyond the X/L_R = 0.70 mark.

\[
\frac{\partial}{\partial x} \left( \frac{A}{A_1} \right) \geq \frac{\partial}{\partial x} \left[ \frac{2C_p{T_{rot}}^2 + 1}{u_1^2} \right]^{\frac{\gamma + 1}{2(\gamma - 1)}} \left( \frac{u}{u_1} \right)^2 
\]  

(5.1)
Figure 5-1 Meridional view of relative Mach number distribution showing the flow features at the design point: POS - passage oblique shock, TS - terminating shock, Cs - casing BL separation, PS - passage shock, HLV - hub leading-edge vortex.

The relative Mach number contours in the rotor passage at 90% span are shown in Figure 5-3 (a). $M_{rel}$ in the 90 - 100% span, near the tip region, was supersonic at 1.3 - 1.4. The wedge shape of the leading-edge generated attached oblique shocks. Inlet flow encountered a series of compression and expansion zones, due to the oblique shock waves of neighboring blade passages before reaching the leading-edge, shown by ‘POS’ in Figure 5-4. The suction side leg of the attached oblique shock ‘a’ extended into the inlet stream. The pressure side leg formed the passage shock ‘b’. The suction side flow accelerated, due to expansion fans ‘I’ and ‘II,’ because the viscous boundary layer was growing over a slightly convex surface. These fans reflected back from the sonic line as compression waves. As the shock was weak in nature, it reattached and then accelerated. The third set of expansion fans ‘III’ formed a stronger shock, which extended to the pressure side of another blade as a passage shock ‘b’. The boundary layer on the suction side was
detached, due to the stronger nature of this shock wave. Another set of expansion fans ‘IV’ were formed, due to a convex boundary layer. It led to a set of weaker reattached flow zones. Finally, it merged into the low momentum zone ‘c,’ which exists due to the casing shock boundary layer separation marked as ‘Cs’ in Figure 5-3 (a). The blade loading diagram in Figure 5-7 clearly shows these features at the 90% span. The gradual decrease and increase in $C_p$ signifies the expansion and compression fans, respectively, whereas the sudden increments correspond to the shocks.

**Figure 5-2** Shock stability analysis using eqn. 5.1 in the rotor passage and the corresponding computational relative Mach number distribution.
Figure 5-3 Relative Mach number distribution at a) 90%, b) 65%, c) 32.5%, d) 10% span at the design point.
Figure 5-4 Pressure distribution meridional view at the rotor design point:

POS – passage oblique shock, Cs – casing BL shock separation, HLV – hub leading-edge vortex.

$M_{1rel}$ was approximately 1.0 at 60% span, as seen in Figure 5-3 (b). The rotor leading edge behaved like that of a transonic airfoil at the 60 - 90% span. The rotor leading edge at this span had a finite radius, which generated a stagnation region, due to the deceleration caused by terminating shock waves ‘TS.’ As observed at the 90% span, inlet flow passes through a series of compression and expansion zones, due to the suction side terminating shock waves ‘d.’ As the flow decelerates to subsonic behind the shock, a stagnation zone ‘e’ prevails, observed as ‘TS’ in Figure 5-1. Suction side acceleration generated an expansion and compression region ‘V’ in Figure 5-3 (b), which was terminated by the shock wave ‘d.’ Further downstream of the shock, prior to merging into the low momentum zone of the casing separation, the flow behaved similar to that within a subsonic compressor passage.
Figure 5-5 Rotor blade suction side vortex visualization using Q-criterion.

The boundary layer growth on the hub surface generated the typical leading-edge horseshoe vortices ‘HLV’ on the blade, as shown in Figure 5-5 and 5-6. This recirculatory flow feature extended on both the suction and pressure sides near the leading edge hub junction. Both the suction and pressure side hub corner vortices ‘HPS CV’ and ‘HSS CV’ were visible, which extended throughout the passage; see Figure 5-5. The subsonic inlet flow at the 0 - 30% span decelerated due to an increment in pressure because of ‘HLV.’ This effect was observed in Figure 5-4. The incidence angle of the relative inlet flow increased, shifting the stagnation point and causing a leading edge separation on the suction side, as observed in Figure 5-3 (d). The rotor leading edge region for the 30 - 60% span behaved like a high subsonic airfoil, seen in Figure 5-3 (c). Incidence effects caused flow separation on the suction side. Leading edge separation in the 0 - 60% span manifested as a rotating spike structure [35], which traveled through the entire passage near the suction side to join the low momentum zone, as seen in Figure 5-3 (a). The tip leakage
vortex was dominant near the leading edge of the rotor blade. It mixed with the low momentum zone. On the pressure side, the ‘HLV’ of PS climbed on the blade towards the tip region and mixed with the low momentum zone, passing over the tip as a leakage flow.

![Figure 5-6](image)

**Figure 5-6** Rotor blade pressure side vortex visualization using Q-criterion

Blade loading distribution at 10 (hub), 65 (mid) and 90% (tip) span for the rotor is shown in **Figure 5-7**. Higher loading in the hub frontal area provided over-bending of the blade and led to flow separation on the suction side causing a reduction in efficiency. For the hub line near the leading edge, the centrifugal force could not generate a high pressure ratio, as the meridional passage height did not increase abruptly. As the hub line height increased on the posterior side, any loading increment in this zone enhanced pressure rising capability. Conversely, for the tip region, higher loading near the frontal area increased the rotor’s pressure-rising ability and decreased loading in the tip posterior area, which controlled the inevitable suction side separation of the tip region. Based on Xuanyu *et al.* [14], increasing hub loading resulted in the impeller achieving
higher performance with lower tip leakage loss and uniform flow at the exit. Increasing tip loading achieved a higher-pressure ratio.

**Figure 5-7** Blade loading distribution along blade meridional chord for 10, 65 and 90% rotor span at the design point.

### 5.2 Stator flow feature description

A two-dimensional computational simulation of the tandem stator configuration is shown in **Figure 5-9.** **Table 5-1** shows the boundary conditions for this computation. The aim was to visualize the shock wave system for this tandem diffuser with a constant passage height from inlet to exit. Inlet boundary condition is obtained from mixed-flow rotor exit data. Current predictions revealed that the stator inlet flow was nearly two-dimensional in nature, as shown in **Figure 5-13.**
An oblique shock was created on the leading-edge of blade ‘1.’ The pressure side leg of the shock impacted the suction side, whereas the suction side leg diffused into the flow. Suction side acceleration was obstructed, due to the impinging passage shock. Separated flow zones were generated, due to this shock on PS and SS locations from shock-boundary layer interaction. Flow turning was limited to 7 - 10° for blade ‘1’ to avoid excessive separation losses, which are the most dominant mechanism. The leading edge SS of blade ‘2’ was placed near the trailing edge PS of blade ‘1.’ The low momentum zone of the separated flow diffused with the suction side acceleration of blade ‘2,’ thus diffusing the shock-BL wake. The subsonic blade ‘1’ exit flow was perfectly aligned with the second blade. The second blade row provided subsonic diffusion along with high flow turning in the 40 - 50° range. These effects are evident in the $C_P$ distribution.

Although the initial flow visualization was performed as a two-dimensional computation in the stator, the final performance computations for the mixed-flow compressor stage were performed using a three-dimensional stator. The present stage design uses a constant diffuser height from the inlet to the exit of the supersonic diffuser, as shown in Figure 5-12. Performance comparison of 2D stator alone and 3D stator in stage simulations with same inlet boundary condition and shock structures are presented in Table 5-2. Figure 5-8 describes the corresponding $P/P_{o1}$ comparison between the mid-span 3D and 2D stator computations. It is inferred that static pressure rise in 3D stator is lower than 2D stator. The authors conclude that a 2D stator alone simulation is highly efficient and effective in predicting constant radius 3D stator’s shock structure and performance. It is concluded that the current design performs better than previous supersonic tandem stator designs [10], [9] and [13] when the inlet Mach number is in 1.1-1.4 range.
Table 5-1 Current tandem stator 2D CFD boundary conditions

| Reference Pressure ($P_{ref.}$), Pa | 88844.5 |
| Inlet Mach number | 1.2315 |
| Inlet $P_o/P_{ref.}$ | 2.528 |
| Inlet $T_o$, K | 456.64 |
| Exit Mach number | 0.4407 |
| Exit $P/P_{ref.}$ | 2.031 |
| $\eta_{IS}$ | 90.1% |

Table 5-2 Performance comparison between 2D stator alone and 3D stator in stage simulations

<table>
<thead>
<tr>
<th></th>
<th>$M_3$</th>
<th>$M_4$</th>
<th>Total Pressure</th>
<th>$\eta_{IS}$</th>
<th>$P_4/P_3$</th>
<th>Diffusion factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>2D (alone)</td>
<td>1.313</td>
<td>0.42</td>
<td>0.92</td>
<td>90.76</td>
<td>2.287</td>
<td>0.73</td>
</tr>
<tr>
<td>3D (stage)</td>
<td>1.308</td>
<td>0.457</td>
<td>0.85</td>
<td>90.12</td>
<td>2.184</td>
<td>0.71</td>
</tr>
</tbody>
</table>

Figure 5-8 Static pressure rise comparison between mid-span 3D stator in stage and 2D stator alone simulation for the same inlet boundary condition and similar passage shock structure.
Figure 5-9 Current tandem stator blade loading and corresponding Mach number contour
Figure 5-10 represents entropy generation of the comparative assessment 2D stator alone simulation. ‘A’ highlights the viscous losses occurring due to boundary layer growth. ‘B’ represents the weak passage shock. ‘C’ defines the shock-boundary layer interaction creating a flow separation. This phenomenon generates the highest entropy as inferred from the computational assessment. Downstream of the separation point, this low momentum flow infuses with high momentum blade row ‘1’ passage flow and aspirated flow between blade ‘1’ and ‘2’. The generated wake diffuses, and this behavior is clearly seen at point ‘D’ as entropy values reduce. Finally, excessive flow turning in blade row 2 suction side leads to mild flow separation at point ‘E’.
Figure 5-10 Entropy generation across the 2D tandem stator based on computational effort
5.3 Stage performance analysis

Performance charts of the supersonic mixed-flow compressor stage are shown in Figure 5-11. The design goal of $\Pi_{TT} = 6.0$ is marked as the dashed horizontal line in Figure 5-11. At the rotor design speed, this compressor stage reached a maximum $\Pi_{TT}$ of 5.83 with 77% efficiency at 86.5% of the intended design mass flow rate. The slight difference against the intended mass flow rate appears since the blocking effect of the rotor blade was not accounted for during design in the mean-line code. A total pressure ratio, $\Pi_{TT} = 6.12$ was achieved at a slightly higher rotational speed of $\Omega/\Omega_o = 1.035$ for an efficiency of 75.5%.

![Figure 5-11](image_url)

**Figure 5-11** Stage total pressure ratio and efficiency versus normalized mass flow rate,
The $\Pi_{TT}$ and $\eta_{IS}$ curves had almost constant mass flow rates, depicting a choking-like condition throughout the $\Omega$ range. This was related to the presence of a shock in the stator passage. As the total pressure ratio of the stage increased with back pressure, as seen in Figure 5-12, the increment in the static pressure was not sensed upstream of the stator passage shock. This effect was clearly visible, as rotor performance was constant for all back-pressure cases, explaining the constant mass flow rate. The static pressure rise pattern for different $P_{o3}/P_{o1}$ cases showed identical results up to the passage shock location in the stator. Flow acceleration on the SS and PS in the stator for low back pressure scenarios pushed the shock downstream in the passage. This increased the total pressure loss, due to the stronger nature of a passage shock reducing the stage total pressure ratio, as seen in the $P_{o3}/P_{o1}$ curves. Conversely, at the maximum back pressure condition, the passage shock in stator blade row 1 was located near the pressure side leading edge. Any further increment would make the simulation unstable. The same behavior was observed in all cases above 80% of the design speed when there was a passage shock in the stator. At a lower $\Omega$, the performance curve varied over a range of mass flow rates when the back-pressure increment traveled upstream into the rotor.

The Mach number variation, obtained as the local speed of sound across the stage, was measured on sections perpendicular to the axis, as shown in Figure 5-13. A slight increase in the radial component was observed, due to the HLV stagnation zone within the subsonic regime. Work induced by the mixed-flow rotor onto the fluid is seen as radial and tangential velocity increase. The axial component had a decrease in magnitude in the $X = 50$ to 100 mm range, due to the low momentum zone generated by the casing boundary layer separation. An absolute Mach number of 1.3, as desired from the preliminary mean-line design study, occurred at the rotor exit. Across the vane-less space, the magnitude of the radial component reduced to nearly zero, due to the hub and casing contouring. This made the stator inlet flow almost two-dimensional in nature. A sudden reduction in the velocity magnitude was observed, due to the passage shock. After the shock, blade
row ‘2’ turned the flow resulting in a reduction of the tangential velocity component. Despite blade ‘2’ exit angle’s alignment with the axial direction, the flow separation on the SS of blade ‘2’ meant a tangential velocity component still existed.

The computational “mixing plane” in between the stator and the rotor fluid domains transferred circumferentially averaged quantities. It contributed to a slight increment in $P_o$ and $P$ quantities. This artificial increment corresponds to a $\Pi_{TT}$ of 0.1, as seen in Figure 5-12. It was also seen as a sudden decrement in velocity magnitude (i.e. $V_a$) in Figure 5-13. Thus, the computational solution immediately upstream and downstream of the mixing plane should be discounted for absolute flow evaluation in transonic cases [23].

The specified design point and CFD results for the stage are compared in Table 5-3. The CFD data was based on mass-averaged quantities. The discrepancy in $\dot{m}$ was due to the rotor blade blocking effect not being considered. This decrement in $\dot{m}$ manifested as an $M_1$ decrement. An $M_2$ of 1.29 was very close to the desired value of 1.3. $M_3$ depended on the supersonic diffusion in the vane-less space, which was not attained in this design. Flow separation on the SS of B2 accelerated the subsonic flow, leading to a higher $M_1$ value than desired. Rotor efficiency was closer to the desired 85%.

The stage total pressure ratio at the design point could be improved by increasing $r_{2m}$ and designing a more efficient 3D diffusion system in the stator.
Figure 5-12 Stagnation pressure and static pressure distribution across the stage for different back pressures.
Figure 5-13 Velocity component distribution across the stage at design speeds.

Table 5-3 Design point vs CFD comparison at maximum back pressure for the mixed-flow compressor stage

<table>
<thead>
<tr>
<th>Property</th>
<th>Design point</th>
<th>CFD ($\Omega/\Omega_0 = 1.0$)</th>
<th>CFD ($\Omega/\Omega_0 = 1.035$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\dot{m}$ (kg/s)</td>
<td>3.5</td>
<td>3.037</td>
<td>3.132</td>
</tr>
<tr>
<td>$M_1$</td>
<td>0.7</td>
<td>0.5326</td>
<td>0.558</td>
</tr>
<tr>
<td>$M_2$</td>
<td>1.3</td>
<td>1.29</td>
<td>1.347</td>
</tr>
<tr>
<td>$M_3$</td>
<td>1.2</td>
<td>1.25</td>
<td>1.31</td>
</tr>
<tr>
<td>$M_4$</td>
<td>0.4</td>
<td>0.468</td>
<td>0.453</td>
</tr>
<tr>
<td>$\Pi_{TR}$</td>
<td>6</td>
<td>5.83</td>
<td>6.12</td>
</tr>
<tr>
<td>$\eta_{IS}$ (%)</td>
<td>75.5</td>
<td>76.93</td>
<td>75.55</td>
</tr>
<tr>
<td>$\eta_R$ (%)</td>
<td>85</td>
<td>84.41</td>
<td>82.85</td>
</tr>
<tr>
<td>$\pi_R$</td>
<td>6.7</td>
<td>6.643</td>
<td>6.98</td>
</tr>
</tbody>
</table>
Chapter 6

Conclusions

The main objective of this thesis was to provide strategic design guidance for a single-stage mixed-flow compressor with a high pressure ratio of 6:1 in the (1-10) kg.s\(^{-1}\) mass flow segment within a 400 mm diameter. A brief historical background of the previous mixed-flow designs, including the pros and cons of the current design, were compared to the centrifugal and axial stages in the small aero-engine segment. As noted, most of the earliest designs suffered from inefficient stator performance, which could not accommodate the high blade loading requirement. Tandem stator designs had been attempted to distribute the high blade loading, but none could provide an efficiency above 80%.

A simple mean-line mixed-flow stage design procedure has been provided based on first principles. As it does not involve any empirical formulas to predict losses, it could be used as an initiating point for designers to comprehensively grasp the fundamentals of turbomachinery design. Along with the definition of geometric parameters, this step is furthered by a design technique based on Bezier curves. For the stator, a 2D cascade design was generated for the tandem configuration, and was further developed into a 3D design based on hub and casing definitions.

The rotor design was then evaluated based on the mean-line equations, which stated that a high pressure ratio demands a supersonic rotor exit flow with a high flow exit angle of 47 - 60° from the axial direction. The rotor efficiency values increased with high passage diffusion. A higher rotor exit Mach number, higher exit mean radius, and low rotor exit relative Mach number were desired to obtain a high pressure ratio. To obtain an increment in the isentropic efficiency for the same pressure ratio, \(M_2\) had to be reduced and \(M_{2rel}\) had to increase. The optimum \(\alpha_{2m}\) was in the \(M_{2m}\)
range of 1.1 - 1.3, generating a rotor $\Pi_{RT}$ of up to 4.5 - 7. A higher $M_{2rel}$ provided a lower $\alpha_{2m}$ and a correspondingly lower $\Pi_{RT}$. In the blade exit angle, $\beta_{2m}$ plot, an inflection point was observed at $M_{2m} = 1$, where higher subsonic diffusion in the rotor relative frame required a lower $\beta_{2m}$ for $M_{2m} > 1$ and vice versa.

Though most of the design parameters could be analyzed in a 1D system, there were other crucial parameters, such as rotor meridional exit angle, axial length, blade solidity, blade loading, and tip leakage that required multidimensional and viscous computational analysis. Higher hub loading allowed the impeller to achieve higher performance with lower tip leakage loss and uniform flow at the outlet. Increasing tip loading achieved a higher pressure ratio. The results showed that it was possible to derive a mixed-flow rotor from a centrifugal variant, reasonably preserving the efficiency and pressure ratio.

The challenging objective of supersonic diffusion and high flow turning with high efficiency and stagnation pressure recovery was achieved with a tandem stator design, based on Quishi et al. [16]. Acceleration of the low momentum flow zones of the first blade by the suction side of the second blade increased the diffusion efficiency. This tandem stator design provided significant performance benefits when compared to the previous mixed-flow stage designs. The design strategy for the supersonic tandem diffuser explained in this thesis is straight-forward to implement in a modern single-stage mixed-flow compressor design effort.

The computational viscous flow features and the performance of a 6:1 total pressure ratio compressor with a 75.5% efficiency designed by the in-house code were elaborately explained. The new tandem-design supersonic diffuser performed better than the previous supersonic diffusion configurations by a margin of 20% in terms of efficiency. This is the key component that enables this design to achieve a very high pressure ratio with much improved energy efficiency.
In order to qualify the current stage performance predictions, NASA rotor 37 and RWTH Aachen supersonic diffuser computations were benchmarked against experimental data to showcase the improved CFD prediction capabilities. The present computational verification effort for the CFD system used in this study resulted in stage exit total pressure and temperature predictions that were better or very similar to the past rotor 37 computations. The present computational approach can be effectively utilized in the prediction of transonic/supersonic compressor rotor flow fields in the development of mixed-flow compressors.

The current computational method’s ability to resolve the aerothermal characteristics of a supersonic diffuser with shock wave systems was assessed using experimental data from RWTH Aachen. This assessment was particularly important because the over-simplified mean-line system was not able to account for compressible flow effects, shock waves, three-dimensionality and other viscous effects, including turbulence. The current computations predicted an isentropic efficiency of this supersonic diffuser at 72.09%.

All except the coarse mesh for the present mixed-flow compressor had prismatic cells sufficient to maintain a $Y^+ < 1$ at all cells adjacent to the wall. Although the coarse mesh proved to be a great tool for iterating designs quickly, the rotor passage shock observed in the coarse mesh was wider and spread over a larger area. Based on the current mesh dependency study, the moderate mesh was extensively used, since it generated a stage performance that was very close to the fine mesh results.

Rotor casing boundary layer separation occurring due to shock instability was the most severe loss creating mechanism. A shock stability analysis was presented to further validate this claim. Increased hub loading achieved higher efficiency with lower tip leakage loss and uniform flow
outlet at the exit. Increased tip loading also achieved a higher pressure ratio. The new stator design performed better than the previous supersonic diffuser attempts.

The total pressure ratio chart of this supersonic compressor had a constant mass flow rate curve, depicting a choking-like condition. At $\Omega > 80\%$, any increment in back pressure shifted the shock location upstream in the stator, but the effect did not travel upstream into the rotor. Mach number distribution across the stage showed that the stator inlet flow was two-dimensional in nature.

The mean-line procedure supported the design of a mixed-flow compressor. The mean-line design code obviously lacked the ability to fully represent the three-dimensionality, viscous flow, and compressible flow effects, due to its inherent over-simplifying assumptions. In spite of the design code’s limitations, a mixed-flow stage design with a pressure ratio of 5.83 and a 75.5% efficiency was achieved from this effort, as observed and quantified from the computational RANS simulations. A pressure ratio of 6.12 was also achievable with only a 3.5% rotational speed increase, at $\Omega / \Omega_o = 1.035$. The current approach presents a complete mixed-flow compressor design system. The method results in higher performance compressors when compared to existing mixed-flow designs in the open literature. The design was also much shorter than corresponding axial designs, resulting in additional system weight and maintenance benefits.
6.1 Future scope of improvements

As a result of the observations from the current computational effort, the following modifications are suggested for possible improvements to the stage performance. Rotor airfoil design could be improved to maintain stable shock positioning throughout the rotor passage. This unstable shock causes casing boundary layer separation, which affects rotor efficiency. The rotor leading edge at the 0 - 30% span needs to be twisted to suppress the leading-edge rotating spike flow feature. A hub surface near the rotor leading edge featuring a trough would improve the loss due to HLV [36]. If ‘Cs’ separation was controlled, $\eta_{IS}$ would increase and $\Pi_{TT}$ would decrease, due to a reduction in passage diffusion.

Splitter rotor blade benefits should be analyzed to reduce the rotor weight. Meridional contours in the vane-less space can be optimized to enhance supersonic 3D diffusion without a normal shock. Stator airfoils, hub, and casing contours may also benefit from further optimization. These changes would aid in improving the stage performance and operational range.

The current stage design evaluation process uses a steady simulation with mixing plane at the rotor-stator interface. An artificial increment in total pressure was noticed across the interface, due to circumferential averaging. Thus, computationally efficient harmonic balance or sliding mesh techniques could be used to study these interactions and assess the stage performance parameters more accurately.
Appendix

1. Inlet parameters

The values provided in Table A-1 define the compressor inlet aero-thermal boundary conditions used in section 2.1. The air properties correspond to an altitude of 38000 ft.

Table A-1 Compressor inlet parameters

<table>
<thead>
<tr>
<th>Property</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Static pressure (Pa)</td>
<td>20646.17</td>
</tr>
<tr>
<td>Static temperature (K)</td>
<td>216.5</td>
</tr>
<tr>
<td>$\gamma$</td>
<td>1.4</td>
</tr>
<tr>
<td>Gas constant, R</td>
<td>287.04</td>
</tr>
<tr>
<td>$C_p$</td>
<td>1005</td>
</tr>
<tr>
<td>$\dot{m}_i$ (kg/s)</td>
<td>3.5</td>
</tr>
<tr>
<td>$M_0$ (engine inlet Mach no.)</td>
<td>0.85</td>
</tr>
<tr>
<td>$M_1$ (compressor inlet Mach no.)</td>
<td>0.7</td>
</tr>
<tr>
<td>Inlet swirl angle</td>
<td>0°</td>
</tr>
</tbody>
</table>
2. Mixed-flow rotor 3D design input parameters

The corresponding set of values were used to define the rotor geometry in section 2.1 and 2.3.

Inlet conditions remained the same for all six rotor cases (a-f).

**Table A-2 Rotor inlet and exit geometric conditions**

<table>
<thead>
<tr>
<th>Inlet conditions</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>$r_{1h}$ (mm)</td>
<td>25</td>
</tr>
<tr>
<td>$r_{1t}$ (mm)</td>
<td>122.3</td>
</tr>
<tr>
<td>$r_{1m}$ (mm)</td>
<td>88.268</td>
</tr>
<tr>
<td>$\beta_{1h}$ (deg)</td>
<td>19.49°</td>
</tr>
<tr>
<td>$\beta_{1m}$ (deg)</td>
<td>51.336°</td>
</tr>
<tr>
<td>$\beta_{1t}$ (deg)</td>
<td>60°</td>
</tr>
<tr>
<td>$\Omega$, RPM</td>
<td>28500</td>
</tr>
<tr>
<td>$M_{trel}$</td>
<td>1.4</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Exit conditions</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>$\Pi_{TT}$</td>
<td>6</td>
</tr>
<tr>
<td>$\eta_{IS}$</td>
<td>100%</td>
</tr>
<tr>
<td>$M_2$</td>
<td>1.2</td>
</tr>
<tr>
<td>$M_{2rel}$</td>
<td>1</td>
</tr>
<tr>
<td>$\alpha_{2m}$ (deg)</td>
<td>47.31°</td>
</tr>
<tr>
<td>$r_{2m}$, (mm)</td>
<td>176.1</td>
</tr>
<tr>
<td>$\beta_{2h}$ (deg)</td>
<td>33.03°</td>
</tr>
<tr>
<td>$\beta_{2m}$ (deg)</td>
<td>35.55°</td>
</tr>
<tr>
<td>$\beta_{2t}$ (deg)</td>
<td>37.72°</td>
</tr>
<tr>
<td>$\Delta_3$</td>
<td>2</td>
</tr>
</tbody>
</table>

**Table A-3 Input parameters for rotor cases (a-f)**

<table>
<thead>
<tr>
<th>CASE</th>
<th>a</th>
<th>b</th>
<th>c</th>
<th>d</th>
<th>e</th>
<th>f</th>
</tr>
</thead>
<tbody>
<tr>
<td>$L_{R}$ (mm)</td>
<td>130</td>
<td>150</td>
<td>170</td>
<td>140</td>
<td>118</td>
<td>100</td>
</tr>
<tr>
<td>$\Phi_2$</td>
<td>10</td>
<td>10</td>
<td>10</td>
<td>30</td>
<td>60</td>
<td>90</td>
</tr>
<tr>
<td>$\Phi_{2h}$</td>
<td>33</td>
<td>33</td>
<td>33</td>
<td>40</td>
<td>53</td>
<td>80</td>
</tr>
<tr>
<td>$\Phi_{2m}$</td>
<td>25</td>
<td>25</td>
<td>25</td>
<td>28</td>
<td>45</td>
<td>75</td>
</tr>
<tr>
<td>$\Phi_{2t}$</td>
<td>16</td>
<td>16</td>
<td>16</td>
<td>20</td>
<td>37</td>
<td>65</td>
</tr>
<tr>
<td>$r_{2h}$ (mm)</td>
<td>170</td>
<td>170</td>
<td>170</td>
<td>170.747</td>
<td>173.03</td>
<td>176.09</td>
</tr>
<tr>
<td>$r_{2m}$ (mm)</td>
<td>181.99</td>
<td>181.99</td>
<td>181.99</td>
<td>181.29</td>
<td>179.117</td>
<td>176.11</td>
</tr>
<tr>
<td>$\Delta_1$</td>
<td>100</td>
<td>100</td>
<td>100</td>
<td>100</td>
<td>100</td>
<td>80</td>
</tr>
<tr>
<td>$\Delta_2$</td>
<td>100</td>
<td>100</td>
<td>100</td>
<td>100</td>
<td>80</td>
<td>80</td>
</tr>
</tbody>
</table>
3. Mixed-flow rotor non-dimensionalized airfoils

The NACA 65 – 206 series airfoil shown in Figure A-1 was used to define the hub, mean, and tip sections of the mixed-flow rotor.

![Figure A-1 Non-dimensionalized NACA 65 series airfoil](image-url)
4. Tandem stator 3D design input parameters

1. Stator hub and casing contour description

Table A-4 Stator hub and casing Bezier curve definitions

<table>
<thead>
<tr>
<th>Φ_{3h}</th>
<th>Φ_{4h}</th>
<th>Φ_{5h}</th>
</tr>
</thead>
<tbody>
<tr>
<td>25°</td>
<td>0°</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Control Points</th>
<th>X (mm)</th>
<th>Y (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Δ_{1h}</td>
<td>150</td>
<td>168.5</td>
</tr>
<tr>
<td>Δ_{2h}</td>
<td>165</td>
<td>\tan(Φ_{3h}, (X Δ_{2h} - X Δ_{1h}))</td>
</tr>
<tr>
<td>Δ_{3h}</td>
<td>250</td>
<td>176</td>
</tr>
<tr>
<td>Δ_{4h}</td>
<td>300</td>
<td>176</td>
</tr>
<tr>
<td>Δ_{5h}</td>
<td>310</td>
<td>171</td>
</tr>
<tr>
<td>Δ_{1t}</td>
<td>148.12</td>
<td>182</td>
</tr>
<tr>
<td>Δ_{2t}</td>
<td>190</td>
<td>192</td>
</tr>
<tr>
<td>Δ_{3t}</td>
<td>250</td>
<td>187</td>
</tr>
<tr>
<td>Δ_{4t}</td>
<td>300</td>
<td>187</td>
</tr>
<tr>
<td>Δ_{5t}</td>
<td>310</td>
<td>185</td>
</tr>
</tbody>
</table>

2. Stator blade row 1 airfoil and mean camber line definition

Table A-5 Custom polynomial airfoil blade row 1 parameters

<table>
<thead>
<tr>
<th>Suction side</th>
<th>Pressure side</th>
</tr>
</thead>
<tbody>
<tr>
<td>λ_{s}</td>
<td>λ_{e}</td>
</tr>
<tr>
<td>2°</td>
<td>2°</td>
</tr>
<tr>
<td>C_{1}</td>
<td>C_{2}</td>
</tr>
<tr>
<td>20</td>
<td>80</td>
</tr>
<tr>
<td>40</td>
<td>70</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>L_{B1}</th>
<th>64 (40% of the Stator chord)</th>
</tr>
</thead>
<tbody>
<tr>
<td>X_{L_{B1}}</td>
<td>L_{R} + L_{VS}</td>
</tr>
<tr>
<td>α_{1}</td>
<td>47.31° (*VS correction required)</td>
</tr>
<tr>
<td>α_{2}</td>
<td>40°</td>
</tr>
<tr>
<td>Δ_{1}</td>
<td>15</td>
</tr>
<tr>
<td>Δ_{2}</td>
<td>15</td>
</tr>
<tr>
<td>Δ_{3}</td>
<td>45</td>
</tr>
</tbody>
</table>
3. Stator blade row 2 airfoil and mean camber line definition

Matrix $P_{B2}$ defines the 4th order Bezier curve to design the mean camber line of blade row 2.

$$P_{B2} = \begin{bmatrix} X_{LB2} & -36 & 0 \\ X_{LB2} + 12 & -\tan(\alpha_3) \cdot 12 - 36 & 0 \\ 255 & -65 & 0 \\ X_{LB2} + 65 & -74.5 + 31 \cdot \tan(\alpha_4) & 0 \\ X_{LB2} + L_{B2} & 74.5 & 0 \end{bmatrix}$$

### Table A-6 Custom polynomial airfoil blade row 2 parameters

<table>
<thead>
<tr>
<th></th>
<th>Suction side</th>
<th>Pressure side</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\lambda_i$</td>
<td>$2^\circ$</td>
<td>$2^\circ$</td>
</tr>
<tr>
<td>$\lambda_e$</td>
<td>$3^\circ$</td>
<td>$3^\circ$</td>
</tr>
<tr>
<td>$C_1$</td>
<td>20</td>
<td>10</td>
</tr>
<tr>
<td>$C_2$</td>
<td>80</td>
<td>90</td>
</tr>
</tbody>
</table>

$\begin{align*}
L_{B2} & \quad 96 \text{ (60\% of the Stator chord)} \\
X_{LB2} & \quad X_{LB1} + L_{B1} \\
\alpha_3 & \quad 40^\circ \\
\alpha_4 & \quad 0^\circ
\end{align*}$
Bibliography


