A Simple Approach to Generating Body Force Models of Jet Engine Fans and its Application to Inlet-Fan Coupling Interaction

Quentin J. Minaker

University of Windsor

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

Recommended Citation

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

by

Quentin J. Minaker

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

Windsor, Ontario, Canada

2019

©2019 Quentin J. Minaker
A Simple Approach to Generating Body Force Models of Jet Engine Fans and its Application to Inlet-Fan Coupling Interaction

by

Quentin J. Minaker

APPROVED BY:

________________________
R. Bowers
Department of Mechanical, Automotive & Materials Engineering

________________________
V. Stoilov
Department of Mechanical, Automotive & Materials Engineering

________________________
J. Defoe, Advisor
Department of Mechanical, Automotive & Materials Engineering

September 12, 2019
Declaration of Co-Authorship / Previous Publication

I hereby declare that this thesis incorporates material that is the result of joint research, as follows: The thesis was authored by Quentin J. Minaker under the supervision of professor Dr. J. Defoe. In all cases, the key ideas, primary contributions, experimental designs, data analysis, interpretation, and writing were performed by the author; Dr. J. Defoe provided feedback on refinement of ideas and editing of the manuscript.

I am aware of the University of Windsor Senate Policy on Authorship and I certify that I have properly acknowledged the contribution of other researchers to my thesis, and have obtained written permission from each of the co-author(s) to include the above material(s) in my thesis.

I certify that, with the above qualification, this thesis, and the research to which it refers, is the product of my own work.

This thesis includes 2 original papers that have been previously published/submitted for publication in peer reviewed journals, as follows:
<table>
<thead>
<tr>
<th>Thesis Chapter</th>
<th>Publication title/full citation</th>
<th>Publication status</th>
</tr>
</thead>
</table>

I certify that I have obtained a written permission from the copyright owner(s) to include the above published material(s) in my thesis. I certify that the above material describes work completed during my registration as a graduate student at the University of Windsor.

I declare that, to the best of my knowledge, my thesis does not infringe upon anyone’s copyright nor violate any proprietary rights and that any ideas, techniques, quotations, or any other material from the work of other people included in my thesis, published or otherwise, are fully acknowledged in accordance with the standard referencing practices. Furthermore, to the extent that I have included copyrighted material that surpasses the bounds of fair dealing within the meaning of the Canada Copyright Act, I certify that I have obtained a written permission from the copyright owner(s) to include such material(s) in my thesis.

I declare that this is a true copy of my thesis, including any final revisions, as
approved by my thesis committee and the Graduate Studies office, and that this thesis has not been submitted for a higher degree to any other University or Institution.
Abstract

Modern aircraft design is seeing an increase in inflow distortions entering the engines as a consequence of modifying the size, shape, and placement of the engine and/or nacelle casing to increase propulsive efficiency and reduce weight and drag. This could take the form of increasing the fan diameter, which generally leads to a decrease in intake length to maintain lower nacelle weight, or fuselage-embedded engines. It is important to be able to predict how these changes will affect the external flow-fan interaction. High computational costs as well as a limited access to detailed fan geometry has impaired the ability of airframers to investigate these interactions.

In this thesis, the objective is to present a process, which is used to create a simplified numerical model, known as a body force model, and which produces, within the framework of a fluid flow simulation, a desired fan performance without the need for detailed geometry. This body force approach uses volumetric source terms and a compressibility correction to model the blade rows. The main advantage of using this approach is that it allows for steady calculations to capture distortion effects; compared to traditional bladed unsteady calculations it reduces the computational cost by two orders of magnitude. The process determines the requirements for the fluid simulations using both a 1D analysis through the fan stage, as well as simplified blade camber shapes, and is enabled by making a series of simplifying assumptions. An example fan stage representative of one seen in modern large bypass ratio engines was created using this process, and was found to produce the desired performance to within 1%. The process is also used to create a stage which mimics the performance of NASA Stage 67. This newly created stage, as well as NASA Stage 67 are inserted into a nacelle and used to predict flow separation at varying crosswind speeds. The simplified stage was capable of reproducing the overall trends well; it over predicted the separation velocity by approximately 6% compared to NASA Stage 67.
Acknowledgments

The work presented in this thesis would not have been possible if not for the support of those close to me. Firstly, I am grateful for the support and guidance of my supervisor, Dr. Jeff Defoe. His expertise in the field of turbomachinery, dedication to his students, and passion towards teaching has greatly influenced me on my path to becoming an engineer. Additionally, I would like to thank my committee members Dr. Randy Bowers and Dr. Vesselin Stoilov for both their time in reviewing my thesis and thoughtful insight on improvements.

To the past and current members of the Turbomachinery and Unsteady Flows Research Group, I would like to thank you for your friendship and support that aided me during our time together. In particular, I would like to acknowledge David Jarrod Hill, Matheson West, Syamak Pazireh, Majed Etemadi, and Hanieh Khalili Param.

A final big thank you is extended to my family: my fiancée, Miranda, my parents, Bruce and Beth, and my brothers Eoin and Keiran for their endless support and encouragement. I would not be at this point in my life without each and every one of you.
Contents

Declaration of Co-Authorship / Previous Publication

Abstract

Acknowledgments

List of Figures

List of Tables

Nomenclature

1 Introduction

1.1 Objective and High-Level Approach

1.2 Major Findings and Conclusions

1.3 Thesis Outline

1.4 Bibliography


2.1 Introduction

2.2 Body Force Formulation

3.1 Introduction ............................................. 42
3.2 Simplified Stage Creation .............................. 45
    3.2.1 Comparison of Stages at Design Condition ....... 48
3.3 Numerical Setup of Full Wheel Crosswind Simulations .... 51
3.4 Results and Discussion of Crosswind Simulations ......... 55
3.5 Summary and Conclusions .............................. 65
3.6 Bibliography ........................................... 67

4 Summary, Contributions, and Future Work .................. 69
4.1 Summary ............................................... 69
List of Figures

2-1 Comparison of the physical blades (left) and to the source term model (right) ................................................. 10
2-2 Computational domains of (a) the single passage bladed RANS simulations and (b) the body force simulations. .............................. 14
2-3 Work coefficient vs. meridional distance through the rotor at: (a) 20% span (b) 50% span (c) 80% span and (d) rotor trailing edge at 70% corrected speed, and $\phi = 0.48$. .......................................................... 15
2-4 Work coefficient vs. meridional distance through the rotor at: (a) 20% span (b) 50% span (c) 80% span and (d) rotor trailing edge at 90% corrected speed, and $\phi = 0.48$. .......................................................... 15
2-5 Meridional profile displaying the geometric parameters required for gas path generation and station numbering. .......................... 18
2-6 Example of blade camber line. .......................................................... 23
2-7 Rotor chordwise work coefficient at 80% span comparison between a blade with and without added incidence from the CFD simulations of the example design shown later$^3$. ........................................... 24
2-8 Velocity triangle at the rotor trailing edge ........................................ 25
2-9 Camber lines for the example design shown later$^1$ produced by running Stagen. Hub, 50% Span, and Tip camber lines are supplied to Stagen 26
2-10 Work coefficient vs. meridional distance through the Stage 67 rotor at:
(a) 20% span (b) 50% span (c) 80% span and (d) rotor trailing edge for comparison of chordwise loading between a real machine and the simplified stage for 90% corrected speed and \( \phi = 0.48 \).

2-11 Computational domain created for internal flow simulations.

2-12 Work coefficient vs. meridional distance through the rotor at: (a) hub (b) 50% span (c) tip and (d) rotor trailing edge at design corrected speed and \( \phi = 0.64 \).

2-13 Chordwise flow coefficient through the rotor at design corrected speed and \( \phi = 0.64 \).

2-14 Deviation through the rotor tip.

2-15 Work coefficient at rotor trailing edge during off-design (takeoff) comparison.

3-1 Comparison between NASA Stage 67 and simplified stage gas paths and meridional blade profiles.

3-2 Work coefficient vs. meridional distance through the rotor at: (a) 20% span (b) 50% span (c) 80% span and (d) rotor trailing edge at design speed.

3-3 Mass flux along rotor trailing edge.

3-4 Computational domain for crosswind simulations. (a) Side view, (b) front view, and (c) zoomed in view of nacelle and fan stage.

3-5 Nacelle casing used within full wheel simulations, where the dashed line represent the casing extended axially downstream and the dashed-dotted line is the rotation axis.

3-6 Separation size and FPR as a function of grid points used to show grid dependence.
3-7 Areas of separated flow within the nacelle at the (a) fully attached condition, (b) separation point, and (c) highly separated condition. 56
3-8 Mach number in plane tangent to crosswind velocity with (a) attached flow and (b) highly separated flow. 57
3-9 Stagnation pressure ratio at rotor leading edge with (a) attached flow and (b) highly separated flow. 58
3-10 Mass flux distribution at the fan face for the (a) fully attached condition and (b) highly separated condition. 59
3-11 Absolute flow angles at the fan leading edge for the (a) fully attached condition and (b) highly separated condition. 61
3-12 Meridional velocity normalised by the blade rotation speed at 50% span at the fan leading edge for the (a) fully attached condition and (b) highly separated condition. 62
3-13 Change in incidence angles from design at the fan leading edge for the (a) fully attached condition and (b) highly separated condition. 62
3-14 DC60 comparison between NASA Stage 67 and the simplified stage as crosswind velocity varies. 64
## List of Tables

2.1 Important characteristics of NASA Stage 67 rotor at 90% speed [11]. 12

2.2 Summary of the bladed simulations grid independence study performed by Hill and Defoe (2018) [2]. 13

2.3 Summary of the body force grid independence study performed by Hill and Defoe (2018) [2]. 13

2.4 Work coefficient RMS errors for NASA Stage 67 at 70% corrected speed and $\phi = 0.48$. 16

2.5 Work coefficient RMS errors for NASA Stage 67 at 90% corrected speed and $\phi = 0.48$. 16

2.6 Summary of the grid independence study for the example design in the next section. 32

2.7 Key design parameters. 33

3.1 Comparison of NASA Stage 67 and the simplified stage. 50

3.2 Summary of the DC60 values. 63
Nomenclature

Symbols

$A$ Area
$a$ Location of maximum camber
$B$ Number of blades in a row
$b$ Blade span
$c$ Blade chord
$c_p$ Pressure coefficient
$e$ Energy
$F$ Thrust
$f$ Force
$h$ Enthalpy
$i$ Incidence angle
$K$ Compressibility correction factor
$L$ Length
$M$ Mach number
$\dot{m}$ Mass flow rate
$\dot{n}$ Blade camber surface normal vectors
$p$ Pressure
$R$ Gas constant
$r$ Radius
s Spacing between blades
T Temperature
U Blade speed
U* Velocity ratio
\tilde{V} Velocity
\tilde{W} Relative velocity
\alpha Absolute flow angle
\beta Relative flow angle
\gamma Specific heat ratio
\delta Deviation
\eta Efficiency
\kappa Blade metal angle
\xi Change in blade angle
\rho Density
\tau Shear stress
\Upsilon Tip radius change parameter
\phi Flow coefficient
\psi_{pr} Stagnation pressure rise coefficient
\psi Work coefficient
\omega Angular velocity

Subscripts

13 Fan trailing edge quantity
19 Nozzle exit quantity
2 Fan face quantity
67 NASA Stage 67 quantity
abs Absolute quantity
BF Body force quantity
BS Bladed simulation quantity
c Compressible
corr Corrected value
hub Hub span location
i Incompressible
inlet Inlet boundary value
is Isentropic
LE Blade leading edge
m Meridional component
mid Mid span location
n Normal component
n\% Location percent
R Rotor
ref Reference quantity
rel Relative quantity
S Stator
slice Slice quantity
Spin Spinner nose
t Stagnation quantity
tip Tip span location
TE Blade trailing edge
x Axial component
y Crosswind quantity
δ Deviation
θ Tangential component
∞ Freestream quantity

Superscripts

\(A\) Area-averaged quantity
\(M\) Mass-averaged quantity

Abbreviations

BMA  Blade Metal Angle
CFD  Computational Fluid Dynamics
FPR  Fan Stagnation Pressure Ratio
RANS  Reynold-Averaged Navier-Stokes
RMS  Root Mean Squared
Chapter 1

Introduction

In an effort to increase propulsive efficiencies in, and thereby reduce fuel consumption from commercial aircraft, manufacturers are using lower fan stagnation pressure ratios (FPR) and thus increasingly larger fan diameters in turbofan engines. Early-generation geared turbofan engines with FPR of 1.4 and bypass ratios of 12 have been shown to reduce fuel burn by up to 16% when compared to prior engines with the same thrust range [1]. This reduction is expected to grow in the future as decreasing FPR are used [2]. This design trend requires ever-larger engine diameters, which leads to increased weight; typically this increase is offset by reducing the length of the nacelle casing. However, shorter nacelle inlets lead to an increased chance of inlet distortion effects, which can negatively affect fan stage performance [3]. It is critical for airframers to have the ability to assess the changes in external flow-fan stage interaction caused by these changes in nacelle design.

To view this interaction, the airframer must be capable of modelling the fan stage. It is an issue as typically the airframer would not have detailed fan stage geometry because they are well protected by the engine manufacturer - this work serves to resolve this problem. Through the use of 1D analysis to determine necessary stage quantities, simplified camber shapes to create blade representations, and simplifying
assumptions, a process is defined that generates a body force model intended for use in assessing external flow-fan stage interaction.

1.1 Objective and High-Level Approach

The objective of this thesis is to present a process that allows for the creation of a fan stage body force model without the need for a prior knowledge of detailed fan geometry. The body force model generated is assessed on its ability to produce the desired performance, capture trends that are found in real modern machines, and reproduce the inlet distortion responses seen when detailed geometry is used. The distortions of interest are those created by varying crosswind velocities. Crosswind flow is defined as flow that moves perpendicular to the axis of rotation of the turbo-machinery. The use of a body force fan model is critical for this process as it greatly reduces computational costs, especially for inlet distortion cases. The reduction in computational costs occurs because the body force replaces physical blade rows with volumetric source terms. This replacement allows for the use of steady as opposed to unsteady, computational fluid dynamics, and reduces the computational grid by approximately two orders of magnitude.

1.2 Major Findings and Conclusions

In this thesis, two major topics are investigated. The first is the development of the body force model design process and its ability to create a body force model that produces the correct desired performance. The second topic is the effect of using a model thus produced on inlet distortion response prediction compared to using a detailed stage geometry.

Using the design process, a body force model representative of a high bypass ratio fan stage is created, the overall performance of which meets the requirements spec-
ified at design conditions. Design elements, such as desired chordwise and spanwise loading distributions are examined. The chordwise loadings show excellent agreement with the design intent at lower span fractions. The local root mean squared (RMS) difference is approximately 4% of the maximum chordwise loading; however, agreement decreases as span fraction increases, and this value increases to 15% in the outer span. The spanwise loading distribution matches well with the design intent with a local root mean squared difference of 0.6%.

The process is also used to create a body force model based on the performance of an existing machine to determine the effects of the simplifications used to generate the model. The overall performance at design is well matched with slight deviations from the desired requirements, namely the FPR is 1.14% over the desired value. The effect of simplification on the ability to capture distortion interaction effects on the fan and nacelle performance was investigated. The simplification has little effect on fan performance prediction. The maximum difference in the predicted incidence angle during highly separated flow was 2.4° in the outer span, which is deemed as an appropriate accuracy when considering the level of intended fidelity. The maximum difference in predicted nacelle performance, measured using a metric which quantifies the stagnation pressure loss in a 60° sector, occurred with highly separated flow; the simplified stage over predicted the loss by 9%.

The intended use of this process is not to produce a detailed, or realistic fan stage design, but to produce a model that recreates the effect that a fan stage has on the flow. It is a tool that is intended to be used during preliminary design of nacelles when detailed fan stage geometry is not available.
1.3 Thesis Outline

This thesis is split into two major chapters, each responsible for one of the topics mentioned. In Chapter 2, the body force formulation is described in detail and its validation is demonstrated. The design process is detailed here along with the implementation of this process into a commercial CFD framework. Finally an example application of this process is demonstrated. In Chapter 3, this process is used to create a body force model based on an existing stage as a design reference. The effects of the simplifications employed are investigated, at both design condition and with inlet distortion. Lastly, conclusions are drawn from this work and possible future plans are described in Chapter 4.
1.4 Bibliography


Chapter 2


2.1 Introduction

In the design stage of an airframe, the external flow around all components must be considered. This is certainly important around engine nacelles, where the external flow will be affected by the operation of the fan. This interaction is dependent on both the positioning of the fan stage within the nacelle and its operating condition [1]. The state of the art for engine modeling in full-airframe computations is to use a
simplified model of the propulsion system. This is done to reduce computational costs compared to traditional bladed Reynolds-averaged Navier-Stokes (RANS) methods. These simplified models use steady computational fluid dynamics (CFD) simulations for non-uniform inflow where normally unsteady simulations would be required, as well as reducing the number of grid cells needed by approximately two orders of magnitude within the turbomachinery blade rows [2].

These modeling approaches are discussed in detail in Godard et al. (2017) [3]. One of the approaches commonly used in full airframe simulations involves using actuator disks. Actuator disks work by imposing changes to flow direction and stagnation quantities over a single plane but are limited in their ability to reproduce the effects of the coupling between external flow and the fan. Godard et al. proposed that the main reason for this inability comes from the fact that the actuator disk takes the inflow as is and computes the outflow accordingly but lacks feedback effects [3]. Through-flow or body force methods are another approach that is examined to help capture this coupling effect however this is a higher fidelity approach and therefore requires more information. Body force methods work by applying sources of momentum and energy in the swept volumes where the blades would normally be, and were found to capture the external flow-fan coupling more accurately than actuator disks [3].

Many variations of body force methods exist however they usually require the user to have detailed blade geometry. Gong’s model [4] and its later refinements in Peters et al. (2015) [1], Hill’s model [2] as well as a Lift-Drag model [5] are examples of these; they require calibration based on experiments or more detailed computations which include the blade rows in detail. This calibration therefore relies on detailed blade geometry and entails additional computational cost. Models such as those proposed by Hall et al. (2017) [6] and Pazireh and Defoe (2019) [7] have been shown to work without the need for calibration, however they still require the blade geometry (no
thickness information needed in Hall’s approach) and the gas path\textsuperscript{1} of the fan stage. The modeling approach of Sato et al. (2019) \cite{8} works without the need of fan blade geometries but still requires information on the gas path, and blade leading and trailing edge meridional\textsuperscript{2} profiles.

However, the airframer may not have selected what engines will be used, and even if they have they may not be able to obtain the fan geometry and/or bypass duct gas path from the engine manufacturer. This means that they would be unable to use the body force methods mentioned above. Tools currently exist that allow for creation of highly detailed stage and blade geometry, however they generally require more experience with turbomachinery, as well as detailed information about the stage. MULTALL is an open source turbomachinery design suite which takes basic stage information and will generate 3D blades and gas paths \cite{9}. Although simplified, these inputs still require the user to have information on the blade performance which may be unknown, such as blade rotation speed or stage work coefficient. The airframer is not interested in the level of fidelity of the blades or the stage; all they require is that the body force recreates the external-internal flow interaction. Therefore the airframer desires to create this body force model based on information that they know to some degree of accuracy, such as, the required thrust, limitations on engine size, and estimation of the fan stagnation pressure ratio (FPR) at the design point.

The objective of this paper is to introduce and assess a body force model generation process which enables simulation of powered nacelles and/or full airframes without any prior detailed fan geometry information. This process consists of 1D analysis to determine the required change in flow quantities through the stage, as well as generating simplified blade camber surfaces and a gas path. The full MULTALL suite is not used since it requires too much input information, however certain tools within the suite are utilized as will be described later. A number of assumptions

\textsuperscript{1}Hub and casing radius as a function of axial position
\textsuperscript{2}Projection of the blade leading/trailing edges to the axial-radial plane
and simplifications are used during these steps to determine the required information. The key outcomes are that (1) the process enables creation of a body force model of a fan stage without a priori knowledge of detailed fan geometry or gas path; (2) this body force model, once implemented in a CFD framework, matches the design intent performance at the design point; (3) this body force matches the desired spanwise loading at the rotor trailing edge, which in this process is uniform, and provides a reasonable estimate of the rotor chordwise loading similar to that found in modern machines. With these outcomes met the resulting model can also be used to assess off-design conditions.

In the first section of this paper the body force formulation is described in detail and its validation is demonstrated. Next the design process is explained and the selected camber shape is presented and validated. The implementation of this process into a commercial CFD framework is discussed, and finally an example application of this process is demonstrated.

2.2 Body Force Formulation

The concept of body force modeling involves replacing the physical rotor and stator blades with momentum and energy sources. These sources are added across a circumferential region covering the radial and axial extent of the physical blades. These sources are used to generate the flow turning, as well as the pressure and temperature changes which occur in the real machine. These sources can be thought as a local blade force smeared across the blade pitch. This concept is visualized in Figure 2-1.
The momentum and energy equations are,

\[ \frac{\partial \rho \vec{V}}{\partial t} + \nabla (\rho \vec{V} \vec{V}^T) + \nabla p - \nabla \cdot \tau = \rho \vec{f} \]  \hspace{1cm} (2.1)

\[ \frac{\partial \rho e_t}{\partial t} + \nabla (\rho h_t \vec{V}) - \nabla \cdot (\nabla \cdot \tau) = \rho (\vec{r} \times \vec{\omega}) \cdot \vec{f} \]  \hspace{1cm} (2.2)

where \( \rho \) is the fluid density, \( \vec{V} \) is the flow velocity, \( p \) is the static pressure, \( e_t \) is stagnation energy (where \( \rho h_t = \rho e_t + p \)), \( h_t \) is stagnation enthalpy, \( r \) is radius, \( \omega \) is angular rotation speed, and \( \tau \) is the viscous stress. The equations are modified to account for the momentum and energy source terms, which are represented as a body force per unit mass \( \vec{f} \).

The body force method chosen for this process is Hall’s model [6], because it requires no calibration and therefore reduces the amount of stage information needed. The required inputs for this approach are the number of blades per row, \( B \), the blade camber surface normal vectors, \( \hat{n} \), and the relative velocity vector, \( \vec{W} \). This force acts to reduce the local deviation \( \delta \), which is the angle between the relative velocity vector and a vector tangent to the blade surface in the plane shared by \( \hat{n} \) and \( \vec{W} \).
The source term per unit mass, here the incompressible normal force, \( f_{n,i} \), is defined as:

\[
f_{n,i} = \frac{2\pi \delta \left( \frac{1}{2} W^2 / |n_\theta| \right)}{2\pi r/B}
\]  

(2.3)

The blade leading and trailing edge meridional profiles, and the full machine gas path are also needed. The original Hall model was only intended to be used for low speed machines (incompressible flow), so a correction factor is added to account for compressibility since modern commercial engine fans operate at transonic relative Mach numbers. This takes the form of an added compressibility correction, \( K \), where:

\[
f_{n,c} = K f_{n,i}
\]  

(2.4)

as used in Benichou et al. (2019) [10]. This correction uses the Prandtl-Glauert rule in subsonic relative flow, and the Ackeret formula in supersonic relative flow,

\[
K' = \begin{cases} 
\frac{1}{\sqrt{1-M^2}} & M < 1 \\
\frac{2}{\pi \sqrt{M^2-1}} & M > 1 
\end{cases}
\]  

(2.5)

and has an upper limit set to avoid instabilities as the relative Mach number approaches 1 giving,

\[
K = \begin{cases} 
K' & K \leq 3 \\
3 & K' > 3 
\end{cases}
\]  

(2.6)

Body force models exist that have added terms to account for the blockage effects caused by the blades; this can be seen in the model used in Benichou et al. (2019) [10]. Including blockage adds complexity as it requires information on blade thickness. This was neglected in the current approach as the aim is not to generate full blade shapes, and as will be shown later is not required to accurately predict the loading in the body force model. Simple loss models exist which require little or no calibration
however the upstream influence of a fan on incoming flow is not significantly impacted by the viscous losses in blade rows [6] and are therefore neglected in the body force model used in this paper.

### 2.3 Assessment of Body Force Approach

A single passage bladed RANS simulation is compared against a body force model to assess the approach both with and without the compressibility correction at the design flow coefficient,

\[
\phi = \frac{V_x}{U_{mid}}
\]  

of 0.48, where \( U_{mid} \) is the rotor blade speed at mid-span. The machine used is NASA Stage 67 [11]. The important features of this machine are shown in Table 2.1. The overall, spanwise, and chordwise loading is examined for both the 70% and 90% speedlines.

Table 2.1: Important characteristics of NASA Stage 67 rotor at 90% speed [11].

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \omega_{corr} (rad/s) )</td>
<td>1512</td>
<td>B</td>
<td>22</td>
</tr>
<tr>
<td>( M_{rel,tip} )</td>
<td>1.2</td>
<td>( \frac{r_{hub}}{r_{tip}} ) *( \frac{M}{U_{mid}} )</td>
<td>0.5</td>
</tr>
<tr>
<td>FPR</td>
<td>1.48</td>
<td>( \frac{r_{hub}}{r_{tip}} ) _inlet</td>
<td>0.375</td>
</tr>
<tr>
<td>( \dot{m}_{corr} ) (kg/s)</td>
<td>31.1</td>
<td>( \frac{r_{hub}}{r_{tip}} ) _outlet</td>
<td>0.478</td>
</tr>
<tr>
<td>True Chord Aspect Ratio</td>
<td>1.56</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

The simulations were run using Ansys CFX 16 [12]. The grids and computational approach are the same as those used in Hill and Defoe (2018) [2]. The bladed Stage 67 simulation uses a single passage containing \( 3.58 \times 10^6 \) cells. Two grids were used to check grid independence; Table 2.2 shows the results of the independence study. Less
than 1% change is seen between the stagnation pressure ratio and isentropic efficiency and it was therefore determined that the medium grid is sufficient. The simulations are steady state, and use the shear-stress-transport turbulence model. The stagnation quantities are set at the inlet and a mass flow rate boundary condition is used at the outlet. In this paper, blades with zero shear stress surfaces were used so that a direct comparison could be made against the body force model which includes only the turning (normal) force in the blade rows. The body force model consisted of a 1/16 annulus slice containing 279,760 cells. In Table 2.3 a summary of the body force grid independence study is shown. The boundary conditions are the same as in the bladed simulations. The computational domains are shown in Figure 2-2. Further information on the computational setup can be found in Hill and Defoe (2018) [2].

Table 2.2: Summary of the bladed simulations grid independence study performed by Hill and Defoe (2018) [2].

<table>
<thead>
<tr>
<th></th>
<th>Medium Grid</th>
<th>Fine Grid</th>
<th>Percent Change</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rotor Cell Count</td>
<td>1.78x10^6</td>
<td>2.45x10^6</td>
<td>37.6</td>
</tr>
<tr>
<td>FPR-1</td>
<td>0.493</td>
<td>0.496</td>
<td>0.71</td>
</tr>
<tr>
<td>Rotor $\eta_{is}$</td>
<td>92.3%</td>
<td>92.3%</td>
<td>0</td>
</tr>
</tbody>
</table>

Table 2.3: Summary of the body force grid independence study performed by Hill and Defoe (2018) [2].

<table>
<thead>
<tr>
<th></th>
<th>Medium Grid</th>
<th>Fine Grid</th>
<th>Percent Change</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cell Count</td>
<td>279,760</td>
<td>609,500</td>
<td>117%</td>
</tr>
<tr>
<td>$\bar{T}<em>{t,2}/T</em>{t,1} - 1$</td>
<td>0.130</td>
<td>0.130</td>
<td>0%</td>
</tr>
</tbody>
</table>
The work coefficient, defined as

$$\psi = \frac{h_t - h_{t,\text{inlet}}}{U_{\text{mid}}^2} \quad (2.8)$$

as a function of chord is shown in Figures 2-3 and 2-4 for 70% and 90% corrected speed,

$$\%\omega_{\text{corr}} = \frac{\omega}{\sqrt{T_1/T_{1,\text{ref}}}} / \frac{\omega_{\text{des}}}{\sqrt{T_2/T_{2,\text{ref}}}} \quad (2.9)$$

respectively. The figures include results from the Hall body force model with and without the compressibility correction as well as the single-passage results, which are circumferentially averaged.
Figure 2-3: Work coefficient vs. meridional distance through the rotor at: (a) 20% span (b) 50% span (c) 80% span and (d) rotor trailing edge at 70% corrected speed, and $\phi = 0.48$.

Figure 2-4: Work coefficient vs. meridional distance through the rotor at: (a) 20% span (b) 50% span (c) 80% span and (d) rotor trailing edge at 90% corrected speed, and $\phi = 0.48$. 
The root mean squared (RMS) difference in local chordwise and spanwise work coefficient as a percent of total change in bladed simulation work coefficient along the chord (mass averaged in the spanwise case) between the body force (BF) models and the bladed simulations (BS), defined as

\[
\%RMS = \sqrt{\int_0^c \frac{(\psi_{BS} - \psi_{BF})^2}{c} \, dx} \times \frac{100}{(\psi_{BS,TE} - \psi_{BS,LE})} \times 100 \quad (2.10)
\]

is shown in Tables 2.4 and 2.5.

Table 2.4: Work coefficient RMS errors for NASA Stage 67 at 70% corrected speed and $\phi = 0.48$.

<table>
<thead>
<tr>
<th>$f_{n,i}$</th>
<th>%RMS</th>
<th>$f_{n,c}$</th>
<th>%RMS</th>
<th>Improvement</th>
</tr>
</thead>
<tbody>
<tr>
<td>20% Span</td>
<td>7.78%</td>
<td>6.62%</td>
<td>1.16%</td>
<td></td>
</tr>
<tr>
<td>50% Span</td>
<td>7.27%</td>
<td>2.74%</td>
<td>4.53%</td>
<td></td>
</tr>
<tr>
<td>80% Span</td>
<td>7.98%</td>
<td>3.75%</td>
<td>4.23%</td>
<td></td>
</tr>
<tr>
<td>Spanwise</td>
<td>4.50%</td>
<td>4.30%</td>
<td>0.20%</td>
<td></td>
</tr>
</tbody>
</table>

Table 2.5: Work coefficient RMS errors for NASA Stage 67 at 90% corrected speed and $\phi = 0.48$.

<table>
<thead>
<tr>
<th>$f_{n,i}$</th>
<th>%RMS</th>
<th>$f_{n,c}$</th>
<th>%RMS</th>
<th>Improvement</th>
</tr>
</thead>
<tbody>
<tr>
<td>Difference</td>
<td>Difference</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>20% Span</td>
<td>9.25%</td>
<td>5.37%</td>
<td>3.88%</td>
<td></td>
</tr>
<tr>
<td>50% Span</td>
<td>10.9%</td>
<td>1.81%</td>
<td>9.09%</td>
<td></td>
</tr>
<tr>
<td>80% Span</td>
<td>12.7%</td>
<td>2.15%</td>
<td>10.5%</td>
<td></td>
</tr>
<tr>
<td>Spanwise</td>
<td>8.34%</td>
<td>3.42%</td>
<td>4.92%</td>
<td></td>
</tr>
</tbody>
</table>

The correction factor improves the accuracy of the body force model in both the
overall and chordwise loadings. The correction factor has a greater influence the larger the relative Mach number becomes; this is seen as the span fraction increases and as the corrected rotational speed is increased. In the 70% corrected speed case the improvement increases by approximately 3% as the span fraction increases, and in the 90% corrected speed case this increases even further to approximately 6%. The reason for this larger improvement in the 90% corrected speed case is due to the fact that the relative Mach numbers are increased throughout the rotor. This is also seen in the fact that the improvement more than doubles throughout in this higher corrected speed case. The results show that the body force model with a compressibility correction is capable of matching the loadings to within 7% which is deemed as an acceptable level of accuracy for this design process.

2.4 Fan Stage Design Approach

We take the fan design point to be cruise, as this is the typical design condition for low fan pressure ratio commercial aircraft engines [13] which are of increasing interest in modern design. The benefit of selecting this typical design condition is that this is usually where the designer will have the most information about required performance. This condition requires the specification of a cruise altitude and flight Mach number. These are quantities an airframer would normally know and provide the information needed to find inlet stagnation quantities. We employ 1D analysis to determine the flow properties through the stage to meet the desired performance at cruise; based on the resulting flow properties as well as a series of assumptions and geometric constraints, the gas path can be defined.

The fan pressure ratio and net thrust are required inputs. The input geometric parameters are the fan blade tip leading edge radius, rotor hub-to-tip ratio \( r_{\text{hub}}/r_{\text{tip}} \) at the leading edge, blade aspect ratios \( b_{R}/c_{R}, b_{S}/c_{S} \) based on the axial chords,
axial distances upstream of, between, and downstream of the blade rows ($L_1/c_R$, $L_2/c_R$, $L_3/c_R$), nozzle contraction length ($L_N/c_R$), hub curvature length ($L_A/c_R$) and fractional tip radius change through the rotor ($\Delta r_{tip}/r_{tip}$). A diffusion factor is specified to determine the number of rotor and stator blades, or these can be directly specified. If an elliptical spinner nose is desired, the axial length of the spinner nose, $L_{Spin}$, is also needed, and is specified as a percent of a linear spinner nose length. The body force model is created to generate a set fan stagnation-to-stagnation pressure ratio at a corrected mass flow which, combined with the gas path geometry, achieves the desired thrust. Figure 2-5 shows the generic meridional profile of the gas path and illustrates the definitions of the geometric parameters.

![Diagram showing geometric parameters](image)

Figure 2-5: Meridional profile displaying the geometric parameters required for gas path generation and station numbering.

The assumptions made are:

1. the axial velocities at the leading and trailing edge of the blade rows are equal and constant along the span,

2. the bypass ratio is high enough that the core flow contribution to thrust generation and the core suction effect on flow in the fan rotor is negligible,

3. the turbomachinery and duct flow are isentropic, and
4. the flow is in the meridional direction at fan inlet.

From assumption (2), we do not include a bifurcation into a core duct in the gas path.

A simple linear scaling is used to set the blade tip relative Mach number. From literature it was found that modern day fans with pressure ratios of 1.6 would be expected to have a tip relative Mach number of approximately 1.4 [13, 14]. We apply this scaling to set our tip relative Mach number based on the design fan pressure ratio ($FPR$):

$$M_{rel,tip} = \frac{1.4}{1.6} FPR \quad (2.11)$$

At design, hub and casing boundary layers are thin and fully attached due to the high Reynolds numbers in practical engine fans and thus we assume no changes in stagnation quantities up to the fan face; these are then set by the flight condition.

### 2.4.1 Stage Performance and Gas Path

Application of control volume analysis to the flow going through the engine yields the standard expression for the thrust

$$F = \dot{m}(V_{19} - V_\infty) + A_{19}(p_{19} - p_\infty) \quad (2.12)$$

where $F$ is thrust, $\dot{m}$ is the mass flow rate, and $A$ is passage area. The thrust, flight velocity, and freestream static pressure are known at the outset, with the other quantities to be determined; this is done using a quasi-1D approach. Two cases can exist, depending on whether the exhaust nozzle is choked or not. The nozzle is choked if

$$FPR \frac{p_{1,\infty}}{p_\infty} \geq 1.893 \quad (2.13)$$
for air with specific heat ratio $\gamma = 1.4$. If the nozzle is choked the nozzle exit static pressure is

$$p_{19} = p_\infty \frac{FPR}{1.893} \left(1 + \left(\frac{\gamma - 1}{2}\right) M_\infty^2\right)^{\frac{\gamma}{\gamma - 1}}$$  \hspace{1cm} (2.14)$$

If the nozzle is not choked the nozzle exit static pressure is equal to the atmospheric static pressure:

$$p_{19} = p_\infty$$  \hspace{1cm} (2.15)$$

The nozzle velocity, assuming isentropic flow, is

$$V_{19} = M_{19} \sqrt{\frac{\gamma R}{1 + \left(\frac{\gamma - 1}{2}\right) M_\infty^2}}$$  \hspace{1cm} (2.16)$$

If the flow is choked, the mass flow and nozzle area are then given by the simultaneous solution of Equations 2.12 and the corrected flow per unit area equation applied at station 19,

$$\dot{m} = \frac{A_{19} p_{19}}{\sqrt{T_{t,19}}} \sqrt{\frac{\gamma}{R}} M_{19} \left(1 + \frac{\gamma - 1}{2} M_{19}^2\right)^{-\frac{\gamma + 1}{2(\gamma - 1)}}.$$  \hspace{1cm} (2.18)$$

In 2.18, the stagnation quantities are the mass-weighted averaged values. To keep the body force model as simple as possible, we design for uniform spanwise work input so that the local values are everywhere equal to the mass-weighted averages.

If the flow is unchoked, the mass flow rate is directly calculated from Equation 2.12 since $p_\infty = p_{19}$. This along with the nozzle exit Mach number is used in Equation
2.18 to determine the nozzle exit area.

The axial Mach number at the fan face (station 2) is found from Equation 2.18 given the fan inlet area (computed from the tip radius and hub-to-tip ratio) and the now-known mass flow rate. This Mach number is then used to determine the static temperature at the fan face.

The assumption of equal leading and trailing edge axial velocities along with the choice of $FPR$ allows the rotor trailing edge area to be calculated. In doing so we neglect the effect of swirl on the required rotor exit area, however, within the design space typically of interest, swirl angles will normally be well under 30° and there is only a minor effect on the passage area [15].

The gas path shape through the rotor is generated using straight line hub and casing curves. This means that the axial velocity will vary within the blade row, but greatly simplifies the generation of the gas path. A parameter, $\Upsilon$, which is a fraction of the fan leading edge span sets the amount of tip radius change through the rotor,

$$\Delta r_{\text{tip}} = \Upsilon(r_{\text{cas,LE}} - r_{\text{hub,LE}})$$  \hspace{1cm} (2.19)

Downstream of the rotor the casing radius is constant.

The slope of the hub through the rotor is set to meet the required decrease in passage area while keeping the leading and trailing edge axial velocities equal.

Downstream of the rotor trailing edge, the hub radius curves back towards axial over some desired fraction of the distance between rotor and stator ($L_A$). The stator span is set to be constant along the chord. In reality the removal of swirl would require a decrease in passage area, but by the same logic applied to the determination of the rotor trailing edge area, this effect is normally small.

The spinner length determines its shape. If the axial length is less than that of a straight line with the rotor hub slope extended to zero radius, then the spinner nose is assumed to be elliptical in shape. It matches the rotor hub slope downstream and
extends to zero radius upstream with the tangent to the ellipse at the nose purely radial. Otherwise, a conical spinner is used with the rotor hub line extended directly down to zero radius.

2.4.2 Blade Performance and Camber

The rotor inlet velocity triangle at the tip, which is determined by the axial Mach found using Equation 2.18 and the relative Mach number found using Equation 2.11 determines the rotation speed of the rotor blades.

A camber surface is needed for the body force model. Camber lines are determined at set span fractions; in the current approach hub, mid, and tip span fractions are used. The camber surface is generated by fitting a 3D surface which passes through these lines.

Chordwise loading distribution has been shown to have an effect on inlet distortions [6], therefore one of the aims is to generate a body force model with a camber surface that produces realistic chordwise loading distributions, while remaining relatively simple. The solution employed is to use camber shapes defined by a combination of a circular arc and a straight line. An example of this camber shape is shown in Figure 2-6. In physical blades the highest loading tends to be in the leading edge region, however in the Hall body force model (Equation 2.3) the loading scales with deviation, which tends to increase towards the trailing edge at design. The intent of pushing all camber curvature forward is to combat this effect. The straight line in the rear section of the chord works to ensure that the required overall flow turning is met as the Hall model acts to reduce the deviation. A range of circular-straight line dividing locations were tested, and it was found that a 50/50 split between circular arc and straight line provides the best combination of guaranteeing the correct flow turning and chordwise loading distribution accuracy as shown later. It should be noted that the model design approach does not produce realistic blade shapes but
increases the accuracy of the loading distribution of the body force model. This is a significant difference compared to the no blade information process used by Sato, Spotts, and Gao (2019) [8] as no real attempt was made to capture realistic chordwise loading in that paper.

In the design velocity triangles the meridional velocity is used as opposed to the axial velocity. This is important because of the significant radial velocities in the rotor, especially near the hub. The consequence is that the velocity triangles and hence camber angles are dependent on the streamsurface inclination since the leading and trailing edge axial velocities are assumed constant.

At the rotor leading edge a small positive incidence of $2^\circ$ is used; this along with the design velocity triangles sets the rotor inlet camber angle. The incidence is added to provide a more realistic chordwise loading. It increases the blade loading and flow deflection in the rotor leading edge region. This also helps ensure that the chordwise loading distributions match predicted trends when the assumption of constant axial velocity is not realized when employing the model within a CFD simulation; if the axial velocity exceeds the assumed value it will cause negative incidence at the leading edge which can result in local work removal. The positive incidence acts to counteract this trend. In Figure 2-7 the chordwise loading is shown at 80% rotor span with and
without the added incidence from the example design described later\textsuperscript{3} to demonstrate the difference in work addition. In the blade with $0^\circ$ incidence the stagnation enthalpy in the first 20% chord drops below the freestream stagnation enthalpy; this could alter the expected distortion interaction behaviour. Adding the incidence eliminates this decrease in the leading edge region. The stator leading edge camber angle is set by assuming zero incidence. Zero incidence was used for the stator leading edge because there is no change in work across the stator which eliminates the need to add incidence to improve the chordwise loading distribution.

Figure 2-7: Rotor chordwise work coefficient at 80% span comparison between a blade with and without added incidence from the CFD simulations of the example design shown later\textsuperscript{3}.

The rotor trailing edge flow angles are set based on the required work input, using the Euler turbine equation,

$$c_p(T_{t,TE} - T_{t,LE}) = \omega (r_{TE}V_{\theta,TE} - r_{LE}V_{\theta,LE}),$$

the design choice to have a constant spanwise work input/pressure rise, and flow deviation. The stator trailing edge is set to remove the swirl from the flow. The

\textsuperscript{3}Section 2.9 of this thesis
trailing edge angles in both blade rows account for deviation. Trailing edge deviation is estimated using a modified form of Carter’s rule [13]:

$$\delta_{TE} = \left(0.23 \left(\frac{2a}{c}\right)^2 + \left(\frac{\alpha}{500}\right)\right) \xi \left(\frac{s}{c}\right)^{0.5} \text{ (degrees).} \quad (2.21)$$

where the maximum camber of the blade is at an axial distance $a$ from the leading edge, $c$ is the axial chord length, $\alpha$ is the exit flow angle ($\beta$ in the rotor), $\xi$ is the overall change in blade angle, and $s$ is the pitch (spacing between blades). Carter’s rule is intended for fully circular blade camber shapes; a modification is made to $a$ to account for the adjustment of the location of maximum camber from the mid point to the new value of 37.5% chord ($a = 0.375$). The relationship between the flow angles ($\alpha$ and $\beta$), blade angle ($\kappa$), and deviation ($\delta$) at a rotor trailing edge are illustrated using the generic velocity triangle shown in Figure 2-8.

![Velocity triangle at the rotor trailing edge](image)

Figure 2-8: Velocity triangle at the rotor trailing edge

The 2D camber line sections are stacked using the open-source turbomachinery design suite MULTALL’s geometry and grid generator Stagen. The information required for Stagen is the camber distribution and the corresponding axial and radial coordinates through the gas path at the set span fractions (0% and 100% are required, however additional span fractions can be supplied to further constrain the design) and the leading and trailing edge meridional coordinates. Stagen creates a single-passage
grid based on the information given; the number of spanwise sections generated is equal to the number of radial grid points requested. The current maximum of 64 points is used here, which has been shown to be an adequate number of radial grid points in body force models [2]. The thickness distribution is set within Stagen to produce blades of negligible thickness such that the maximum thickness is less than 1% of the blade chord. Blade thickness information is not required for the body force model, and therefore this is done so the camber surface extraction provides a good estimate. The blade sections are stacked with their centroids lying along a radial line through the centroid of the hub blade section. Shown in Figure 2-9 are the camber lines that are produced by Stagen. The grid points on the blade surfaces are then extracted and this is used to generate the 3D blade shapes. The camber surface is found by extracting the average of the \( r\theta \) coordinates of the pressure and suction sides of the 3D blade shapes at each axial location. Finally the camber surface normal vectors used in the compressibility-corrected Hall body force model are calculated using the MATLAB [16] built in function “surfnorm”. For more information on how Stagen works refer to Denton (2017) [9].

![Camber lines](image)

Figure 2-9: Camber lines for the example design shown later\(^4\) produced by running Stagen. Hub, 50% Span, and Tip camber lines are supplied to Stagen

\(^4\)Section 2.9 of this thesis
2.5 Assessment of Camber Distribution

The simplified camber shape is assessed by using NASA Stage 67’s gas path and overall performance specifications at $\phi = 0.48$ and 90% corrected speed. The purpose is to determine how accurately a half circular arc, half straight line camber distribution can reproduce the real stage’s chordwise loading. Leading and trailing edge meridional profiles, as well as the gas path are kept the same as the existing NASA Stage 67 to isolate loading changes caused by camber shape. The trailing edge blade angles for the simplified camber distribution are iteratively adjusted to minimize the spanwise work coefficient distribution $\%RMS$ value at the trailing edge between the simplified and the original camber. The loading distributions are shown in Figure 2-10. The spanwise loading (shown in Figure 2-10 (d)) $\%RMS$ difference value is 3.19%; with the outer region having a larger contribution to this difference. The work coefficient in the outer region has a higher sensitivity to adjustments in blade angle which leads to increased computational costs to reduce the $\%RMS$ difference in this region, therefore the accuracy of the trailing edge blade angle was iterated to $\pm 0.05^\circ$. The chordwise loadings display similar overall trends especially at lower span fractions with the $\%RMS$ difference being 6.48% and 7.43% along the 20% and 50% span lines respectively. This slight difference is due to the modified distribution having larger work addition within the first 50% chord, but this is expected as this is where blade turning occurs. As span fraction increases this difference becomes more evident as the $\%RMS$ difference increases to 13.2%; again this stems from the increased sensitivity due to blade angles changes in the outer span region. The $\%RMS$ difference provides a way to quantitatively compare the loading distributions to those of a real machine however it is expected that the loadings distributions will not be exactly the same, as the camber distributions are different. The general trends in the rate of work addition show similarities which is a good indication that although the camber distribution is relatively simple it produces loadings similar to those in real machines.
Figure 2-10: Work coefficient vs. meridional distance through the Stage 67 rotor at: (a) 20% span (b) 50% span (c) 80% span and (d) rotor trailing edge for comparison of chordwise loading between a real machine and the simplified stage for 90% corrected speed and $\phi = 0.48$.

2.6 Estimating Operating Conditions for Off-Design Thrust and Mass Flow

One of the intended uses of the models produced by the design process is to allow airframers to investigate external-fan interactions at a variety of different conditions. To investigate off-design conditions the user must know the fraction of design inlet corrected mass flow,

$$\dot{m}_{corr} = \dot{m}_i \sqrt{\frac{T_i}{T_{i,ref}}} \left( \frac{P_{t,ref}}{P_t} \right) \quad (2.22)$$
as well as the flight and atmospheric conditions. If the off-design operating point yields an unchoked nozzle, inlet corrected mass flow is a function of the fan pressure ratio and thus rotational speed. When investigating the off-design conditions it is assumed that the fan is operating along the working line; in CFD this requires the outlet boundary condition to be set at a constant pressure. The fact that, in general, the flow coefficient is nearly a constant along a fan’s working line is used to assume a linear relationship between the fraction of design inlet corrected mass flow and the fraction of design corrected speed

\[
\frac{\dot{m}_{\text{corr}}}{\dot{m}_{\text{corr,des}}} = \frac{\omega_{\text{corr}}}{\omega_{\text{corr,des}}}
\]  

(2.23)

This yields an initial guess for the rotational speed required to drive a certain mass flow through the machine.

With the mass flow supplied the fan inlet axial Mach number can be found using Equation 2.18 at the fan face and subsequently the static temperature can be found. With the axial Mach number and static temperature, the axial velocity is known. It is again assumed that the axial velocity is equal at the rotor leading and trailing edges. While this assumption is acceptable for the design condition it will be far less accurate off-design, however this is only used as an initial estimate which is then later corrected through an iterative process. Using the new velocity triangles and the blade angles set at design the Euler turbine equation, Equation 2.20, is used to determine the rotor outlet stagnation temperature, and this is then used to determine the rotor outlet stagnation pressure. The stagnation quantities are found at the hub, 50% span, and the casing; a parabolic curve is then fit to these points and that is used to analytically mass average the rotor outlet stagnation conditions. Using the stagnation quantities at the rotor outlet the same steps as before are used to determine the axial velocity, static temperature, and static pressure at the nozzle exit. With the static pressure and temperature known the density at the nozzle exit is found, which allows
for the mass flow rate to be computed. This mass flow rate is compared to the desired
mass flow rate and the process is repeated with the rotational speed altered until the
desired mass flow rate is achieved. This provides an initial guess for the rotational
speed, however CFD simulations must be run and the rotational speed adjusted to
verify that the off-design operating point has been correctly found. An example of
this process is shown later in this paper and the off-design predictions are compared
to those found using CFD and the overall 1D performance prediction is matched to
within 2%.

2.7 Implementation of the Body Force Model Generation Approach

The design process has been implemented as a MATLAB [16] code, but could be im-
plemented in any scientific computing system. It generates the hub and casing curves
as well as the 2D blade camber lines at the hub, 50%, and tip span fractions. The
blade camber surface extraction process is also done within MATLAB. The process
runs on a personal workstation and is computationally inexpensive. Computational
run times for all steps (including Stagen) are typically under two minutes.

2.8 Implementation in 3D CFD

Hub and casing curves, as well as the the blade leading and trailing edge profiles are
imported into grid generation software (here, Pointwise v18 [17]) to generate the gas
path and demarcate blade swept volumes, which must be designated as separate cell
zones.

Shown in Figure 2-11 is a computational domain created using this process. The
upstream and downstream boundaries are placed 1.2, and 1 fan diameters from the
rotor leading edge respectively. These are set far enough away to provide clean inflow and avoid possible interactions with the blade rows. The process creates a constant radius (equal to blade tip radius) casing curve upstream of the rotor blades. It should also be noted that in the design point simulation the outlet nozzle is manually cut slightly before the throat area($A^*$); in the example case this was at $A/A^*$ of 1.08 (nozzle length cut by %10 before the throat area). This was done to avoid having a Mach number equal to one occurring at a boundary condition, which was found to lead to stability issues in some solvers.

![Computational domain created for internal flow simulations.](image)

Four grid levels are used to assess grid independence. A 5-degree slice of the full machine is used with uniform inflow. This saves computational cost as the body force model produces circumferentially uniform flow when the inflow is uniform. Simulations are run at the design operating point for the design detailed in the next section. All grids are fully structured using hexahedral cells with higher mesh density in the bladed areas. A summary of the grids tested is in Table 2.6. Only a 0.9% change in pressure rise coefficient was seen from the second finest grid to the finest grid and therefore the second finest grid is chosen as the final grid level.
Table 2.6: Summary of the grid independence study for the example design in the next section.

<table>
<thead>
<tr>
<th>Overall Cell Count</th>
<th>((x,r,\theta)) Grid Count</th>
<th>Percent Change in Pressure Rise Coefficient</th>
</tr>
</thead>
<tbody>
<tr>
<td>30024</td>
<td>(147,25,10)</td>
<td>N/A</td>
</tr>
<tr>
<td>123480</td>
<td>(288,50,10)</td>
<td>8.1%</td>
</tr>
<tr>
<td>464310</td>
<td>(474,100,11)</td>
<td>1.5%</td>
</tr>
<tr>
<td>914860</td>
<td>(621,150,11)</td>
<td>0.9%</td>
</tr>
</tbody>
</table>

The computations are carried out using Ansys CFX v18.2 [12], using the same boundary condition types described in the NASA Stage 67 body force simulations at the design operating point, however at off-design operating points the outlet static pressure is fixed based on the atmospheric conditions being tested and the rotational speed is adjusted until the desired mass flow is reached. This is done to remain on the fan working line. The hub and casing are set as zero shear stress walls as the model assumes no losses.

### 2.9 Example Application of Process

In this section we present an example application of the model generation process and its implementation into CFD. The main purpose of this example is to show the level of expected accuracy of the desired performance at design and therefore assumes that there is no inlet distortion or separation. In part 2 of this paper\(^5\) the ability to predict these flow phenomena is examined. The example design discussed is based a on high-bypass ratio turbofan engine for a medium-range jet airliner. The geometric values, as well as the desired thrust are based on publicly available information found

\(^5\)Chapter 3 in this thesis
on the Pratt & Whitney 1500G engine [18]. The key design parameters that are used in this example are shown in Table 2.7.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>FPR</td>
<td>1.4</td>
<td>$b_{R/R}$</td>
<td>2.33</td>
</tr>
<tr>
<td>Thrust at cruise</td>
<td>16.75kN</td>
<td>$b_{S/S}$</td>
<td>2.25</td>
</tr>
<tr>
<td>Fan tip diameter</td>
<td>1.85m</td>
<td>$L_1$</td>
<td>8</td>
</tr>
<tr>
<td>Fan hub-to-hip ratio</td>
<td>0.3</td>
<td>$L_2$</td>
<td>0.813</td>
</tr>
<tr>
<td>Cruise Mach number</td>
<td>0.78</td>
<td>$L_3$</td>
<td>2</td>
</tr>
<tr>
<td>Cruise altitude</td>
<td>10668m</td>
<td>$L_N$</td>
<td>2</td>
</tr>
</tbody>
</table>

In this example application the number of rotor and stator blades is supplied based on the Pratt & Whitney 1500G engine, and are 18 and 36 respectively. The spinner nose is set such that the rotor hub slope line is extended to zero radius.

2.9.1 Results at Design Point

These inputs produce a stage with the gas path shown in Figure 2-11. The rotational speed of the rotor (camber lines shown in Figure 2-9) is 374 rad/s and the required mass flow computed is 182 kg/s. The computed increase in the mass averaged fan stagnation pressure ratio across the rotor (FPR-1) was found to be 0.395 which is 1.25% below the design intent and the mass averaged stage work coefficient was found to be 1.9% lower than the desired value. Using the mass averaged Mach number at the outlet boundary it was found that the area which would create choked flow was 0.8% lower than that generated by the design code. This resulting smaller area required is a result of the stage slightly under predicting the FPR and work coefficient. If the flow is isentropically brought to the nozzle area for choke, the thrust generated would be 0.14% lower than the desired thrust. In Figure 2-12 the rotor chordwise
and trailing edge spanwise work coefficients are shown. The spanwise trailing edge work distribution has an RMS difference of 0.6% from the mass averaged overall work coefficient, so the goal of uniform outlet stagnation temperature is largely achieved.

Figure 2-12: Work coefficient vs. meridional distance through the rotor at: (a) hub (b) 50% span (c) tip and (d) rotor trailing edge at design corrected speed and $\phi = 0.64$.

The loading is compared against a 1D prediction generated using the Euler turbine equation and assuming constant axial velocity through the blade, as well as a linear build-up of in deviation along the chord. The 1D prediction has a discontinuity of slope at the transition from circular arc to straight line camber. The chordwise loading is well predicted at lower span fractions with % RMS difference between the 1D prediction and the CFD being 4.12% and 4.09% at the 20% and 50% span fractions respectively, however as span fraction increases to 80% span the accuracy decreases and the % RMS difference increases to 14.5%. The error in the prediction
stems from the difference in axial velocity between the 1D prediction and the CFD simulations. The assumption of equal axial velocity at the blade leading and trailing edge, and along the span is not borne out upon model implementation. In Figure 2-13 the chordwise flow coefficients at set spans within the rotor are compared; equal axial velocities at the leading and trailing edge would result in all the lines starting and ending upon one another at a value of 0.64.

Figure 2-13: Chordwise flow coefficient through the rotor at design corrected speed and $\phi = 0.64$.

The flow coefficient and the work coefficient are inversely related to one another for a given rotational speed. This trend matches what is displayed at the leading and trailing edge regions in the chordwise loading distributions shown in Figure 2-12. This further confirms the need for positive incidence at the rotor leading edge as this variation in velocities causes a change in inlet relative flow angle. This is the result of the lack of consideration of 2D/3D effects in the design code.

The largest difference in flow coefficient was found to be a deficit of 0.22 in the rotor hub leading edge region, which corresponds to an increase in relative flow angle of $11^\circ$. This rather large difference in relative flow angle has minimal impact on the
local work coefficient when compared to the predicted value as shown in Figure 2-12(a). This change is minor because it occurs at a low span fraction as well as the hub leading edge blade angles being relatively small.

The goal is to create a chordwise loading distribution similar to those seen in modern machines. This is a qualitative goal as there is no one correct solution. The lower 80% span fractions seem to be producing acceptable chordwise loading distributions as work is increasing almost linearly along the chord, similar to what is seen in Stage 67. However in the upper 20% span the work coefficient increases to a maximum and then slightly decreases through the last 10-15% chord. This decrease is caused by a combination of two effects. The first is the variation in spanwise axial velocity mentioned prior, and the second is the deviation buildup along the chord. Shown in Figure 2-14 is the deviation through the rotor at the tip. In the prediction code the deviation linearly changes from the $2^\circ$ caused by the incidence to the final predicted value. In the body force simulation deviation decreases in the leading edge region until it approaches the blade camber angle; it then starts to increase as the flow is turned through the circular arc camber section. At mid chord the camber angle stops changing which allows for the body force to rapidly decrease the deviation, however it over turns the flow slightly which then the causes the body force to remove local work as it acts to return the deviation to 0. The buildup of deviation is not known a priori and therefore makes it difficult to predict the severity of this work removal. The overall work addition is very well predicted and shows that keeping the rear 50% camber constant is working as intended to ensure the correct work addition by the rotor trailing edge.
2.9.2 Results Off-Design

The off-design point of interest in this example is the start of takeoff roll condition, with no forward movement of the aircraft. The desired corrected mass flow at take off was chosen to be 114% of design; the initial prediction for corrected rotation speed was found to be 119%. After running detailed CFD it was found to be 116%. The thrust is lower than the thrust found by the 1D mass averaged takeoff prediction code by 1.79%, which stems from the CFD producing a FPR of 0.016 less than the 1D prediction.

In Figure 2-15 the trailing edge work coefficient is shown, and as can be seen the prediction code does not accurately capture the spanwise loading correctly. This inaccuracy is not unexpected as running at off design conditions modifies the velocity triangles to an extent that the simplifying assumptions are no longer valid. A large decrease in the meridional velocity at the tip caused by a large increase in density which is not captured within the prediction code due to the simplifying assumptions is the cause of increased work coefficient.
The prediction code struggles at reproducing the chordwise and spanwise distributions due to the simplifying assumptions however it is still beneficial due to the fact that it provides estimates of the 1D values to within 2% and a initial corrected rotation speed which was found to be within 3% of the desired value, which saves computational time by reducing the amount of iteration required.

### 2.10 Summary and Conclusions

In this paper, a process has been described that generates a body force model which meets the performance requirements specified. This proposed process differs from current models in that it does not require information about blade or passage geometry. The body force model uses Hall’s formulation with an added compressibility correction. This approach was assessed by comparing the results to those found with traditional single passage simulations using the NASA Stage 67 fan geometry. Good agreement is seen with the max %RMS difference in the chordwise and spanwise work
coefficient being 6.62% and 4.40% respectively.

CFD simulations using the models produced match the desired performance well at the design point, where the desired fan stagnation pressure ratio and thrust are matched to within 1.25%. The design objective of constant spanwise work addition is met as the RMS variation is 0.6% from the mass averaged overall work coefficient. The simplified camber distribution is used with NASA Stage 67 information to assess if the chordwise loading produced is realistic. The chordwise loading within the inner 80% span matched closely to those produced by NASA Stage 67 with the %RMS being below 8%, with the remaining outer span showing less agreement with a %RMS of 13.2%. The agreement in the outer span could be improved by adjusting the ratio between the circular arc and straight line camber distribution; in this case increasing the size of the circular arc section would create a loading distribution more similar to that of NASA Stage 67. This suggests that the camber distribution should be a function of span. The rate of work addition through the rotor show similarities, which indicates that the simplified camber distributions are producing realistic loading distributions when considering the intended use and level of fidelity. The intended use of this process is not for fan design but for assessing external flow-fan interactions when limited fan information is available. This process creates the fan stage which would be integrated into a nacelle and run in full wheel simulations. This is useful when airframers wish to investigate coupling between non-uniform inflows caused by off-design operation and the engine fan, such as a takeoff with crosswind.

The current process is aimed to be used in the preliminary design of the propulsion system and airframe integration; in part 2 of this paper an example of this is done by installing a fan gas path into a nacelle. The setup is then run at crosswind and the results are compared to those produced by a real machine with similar performance.

---

6Chapter 3 of this thesis
2.11 Bibliography


Chapter 3


3.1 Introduction

Modern aircraft engine design is moving towards using larger bypass ratios with lower fan stagnation pressure ratios. This improves propulsive efficiency, however it comes with the negative trade off of larger and heavier nacelles. To combat this
negative side effect, manufacturers are using shorter inlets with thinner nacelle lips [1], however these changes increase the chance of flow separation occurring in the inlet. The airframers’ ability to determine the impact of these changes is important during design. To model such situations the airframer must be able to also model the engine fan stage as its operation will greatly impact the intake performance [1]; however two major concerns arise while modeling these flows. The first is that it requires full wheel simulations to capture the non-uniform flow caused by inlet separation; using traditional bladed full wheel simulations to model non-uniform flow is very computationally expensive. These bladed full wheel simulations can contain over 100 million cells for the internal flow alone, usually require 20-30 rotor revolutions to obtain statistically stationary results [2, 3], and can take over two months to complete even with modern computing power. The second issue is that access to detailed fan stage geometry may not be possible and that most airframers lack the expertise or time required to reproduce this geometry.

The solution to the first issue is to use a simplified model of the propulsion system. This reduces the computational cost, as it allows for steady simulations and reduces the number of cells required. Godard et al. (2017) [4] discusses multiple simplified modelling approaches and concludes that a body force approach is needed to capture the coupling effect between the external flow and the fan’s operation. Body force methods work by applying sources of momentum and energy in the swept volumes where the blades would normally be. Another study conducted by Burlot et al. (2018) [5] compares simplifying methods to high fidelity unsteady Reynolds-averaged Navier-Stokes (RANS) simulations of a nacelle casing with non-uniform inflows; it agrees with the findings of Godard et al. (2017) and shows that body force models work better than other simplifying methods at reproducing the results of the unsteady RANS simulations. The body force method captures radial distributions of stagnation pressure and downstream distortion maps as smeared out averages of that seen in the
bladed simulations.

Many variations of the body force method exist however they commonly require calibration based on experimental results or detailed bladed simulations. As described in part 1 of this paper\(^1\), the model produced by Hall et al. (2017) [6] requires no calibration, however it still requires blade geometry and gas path information. This relates to the second issue mentioned earlier as this information would typically not be available. Part 1\(^1\) describes a process which creates a simplified fan stage body force model using Hall’s method with limited stage information, and was shown to produce the desired results at design conditions.

In this process the desired thrust, fan stagnation pressure ratio (FPR), and geometric parameters, all of which would commonly be known by an airframer, are inputs used to generate a body force model. The process consists of using 1D analysis through the fan stage, simplified blade camber shapes, and simplifying assumptions to find the required information needed for the Hall body force approach. How the parameters are found, what assumptions are made, and the steps used to create the body force model are described in part 1\(^1\).

An important non-uniform flow studied is crosswind around a nacelle because it is the most likely scenario to result in inlet separation. In Yeung et al. (2019) [7] different nacelles are tested with crosswind flows and the effects on the separation velocity and the stagnation pressure distributions are described. In Lee et al. (2018) [8] the effect that crosswind has on the operation of the fan stage is analyzed. It was found that this inlet distortion causes a loss in stall margin, even in cases where separation has not occurred. The authors also discuss the suppression effect that the fan applies to the inlet separation. These works further demonstrate the importance for designers to be able to incorporate this external flow-fan stage interaction into the nacelle design process.

\(^1\)Chapter 2 of this thesis
The objective of this paper is to assess the ability of a model produced using the simplified process detailed in part 1\textsuperscript{2} to predict the crosswind separation velocities seen in a real machine operating with crosswind. The model design process is used to create a stage based on a real machine, NASA Stage 67. Both the created stage and the original NASA Stage 67 are run with varying crosswind speeds using Hall’s model and the results are compared. In Hall et al. (2017) [6] the model’s ability to correctly capture upstream flow redistribution and distortion transfer showed good agreement with higher fidelity models. Comparing the results from NASA Stage 67 to those of the simplified stage will allow for a quantification of the accuracy lost by simplifying the stage design. The key outcomes are that the design simplification has minor effects on the rotor and the nacelle performance prediction.

The first section of this paper\textsuperscript{3} will discuss how the simplified stage is created. This simplified stage is run at the design conditions and compared to the original NASA Stage 67. Next the numerical setup of the full wheel crosswind simulation is described. In the results section the full wheel crosswind body force simulations using the original NASA Stage 67 and the simplified stage are compared. This includes the difference in separation velocities and the effect on the fan stage performance.

### 3.2 Simplified Stage Creation

The simplified generation process is used to create a stage based on the performance and geometry of NASA Stage 67. The FPR supplied to the simplified design process was found using the results of NASA Stage 67 running at 70\% corrected speed at the design flow coefficient using Hall’s model [6] with an added compressibility correction, as described in part 1 of this paper\textsuperscript{2}. Normally the process would not require any prior simulations but as the intent is to create a stage as similar to NASA Stage 67

\textsuperscript{2}Chapter 2 of this thesis

\textsuperscript{3}Chapter 3 of this thesis
as possible this is done here. The corrected speed of 70% was used as this provides similar tip Mach numbers to those seen in modern engine fan stages. The details of this simulation, including the setup, validation, and results can be found in part 1\textsuperscript{4}. This was found to produce a FPR of 1.31; this value is then supplied for the simplified design process.

The process described in part 1\textsuperscript{4} uses the desired thrust to determine the required mass flow for the machine, however in this case the mass flow from the NASA Stage 67 simulation is used so a fair comparison can be made between the two models. This means that the process was slightly altered such that mass flow is supplied, instead of a desired thrust. This is done by substituting the mass flow rate as opposed to the thrust in the step where the process uses the standard thrust equation,

\begin{equation}
F = \dot{m}(V_{19} - V_{\infty}) + A_{19}(p_{19} - p_{\infty})
\end{equation}

where the subscripts represent the quantities at the stations shown in Figure 3-1. The thrust then becomes an output of the process, though it is not important in this instance.

The simplified process requires several geometric parameters to be specified to enable the generation of the gas path. In this case all of the parameters required are found using the Stage 67 geometry and applying simplifications as required. Similar to the FPR, these parameters would not normally need detailed geometry a priori however this is done so that a comparison can be made and the effects of simplifying the design can be seen. The parameters were generated as follows:

\begin{itemize}
  \item the inlet rotor tip radius of 0.255\textit{m}, and hub-to-tip ratio of 0.375 are set equal to NASA Stage 67;
  \item the simplified process generates constant axial coordinate leading and trailing
\end{itemize}

\textsuperscript{4}Chapter 2 of this thesis
edge blade profiles. The blade aspect ratios were set such that these axial coordinates were equal to the average axial coordinates of the leading and trailing edge profiles from NASA Stage 67, which gives rotor and stator blade aspect ratios of 2.53 and 2.22 respectively;

- the parameter which determines the amount of casing radius change through the rotor was set so that there is an equal decrease in radius between the two machines and is 0.04;

- the spacing between the blades is set so that the stator leading edge is set at an equal axial coordinate to that in NASA Stage 67 and is 0.706;

- the axial length of the spinner nose is set so that the upstream distance from the rotor leading edge (averaged for NASA Stage 67 case) are equal in both cases and is 94% of the length if the rotor hub slope was continued to zero radius.

A comparison between the original NASA Stage 67 gas path and the gas path generated using the simplified process is shown in Figure 3-1.

![Figure 3-1: Comparison between NASA Stage 67 and simplified stage gas paths and meridional blade profiles.](image)

There is a major difference between the blade shapes and their operation in the NASA Stage 67 compared to the simplified stage. The simplified process generates blade angles at the hub, mid, and tip spans. These are used to create a camber
surface. The change in rotor blade angle from leading to trailing edge found at these
spans are 33.2°, 7.5°, and 2.3° respectively. The change in blade angles at the same
span locations in NASA Stage 67 are 63.2°, 17.8°, and 12.2°. It is seen that the
simplified process generates blades with significantly less turning, however it creates
the required overall flow turning by increasing the rotational speed of the blades. The
corrected rotational speed of the rotor blades in the simplified stage is 1435 rad/s
compared to 1176 rad/s used by NASA Stage 67. This difference in blade shapes
is not unexpected or considered an issue though as the aim is to reproduce stage
performance rather than blade shapes.

CFD simulations are run with the simplified stage at the design point. A 1/8 slice
of the simplified stage is used with uniform inflow, with the same boundary conditions
as the NASA Stage 67 simulation used to determine the supplied FPR. The grid used
contains 559,520 cells and has a distribution similar to what was used by the NASA
Stage 67 simulations and therefore the grid independence study conducted for NASA
Stage 67 in part 1\(^5\) is considered sufficient to treat the new results as grid-independent.
The solver and grid generation tools are the same as those used in part 1\(^5\).

\subsection{Comparison of Stages at Design Condition}

The overall performance of the simplified stage is compared against NASA Stage 67
at 70\% corrected speed. The simplified stage has a mass averaged fan stagnation
pressure ratio (FPR-1) of 0.325, which is an 3.83\% increase from the desired value of
0.313. In Figure 3-2 the chordwise and trailing edge spanwise work coefficient,

\[ \psi = \frac{h_t - h_{t,\infty}}{U_{mid,67}^2} \]  

\(^5\)Chapter 2 in this thesis
is compared between the simplified stage and NASA Stage 67, using the blade velocity in NASA Stage 67 for normalisation. The spanwise mass averaged work coefficient found in the simplified stage is 5.15% higher than the desired value. The root mean squared (RMS) local difference in work coefficient normalised by the total work addition along the chord at the 20%, 50%, and 80% span lines as well as at the rotor trailing edge are shown in Table 3.1.

Figure 3-2: Work coefficient vs. meridional distance through the rotor at: (a) 20% span (b) 50% span (c) 80% span and (d) rotor trailing edge at design speed.
Table 3.1: Comparison of NASA Stage 67 and the simplified stage.

<table>
<thead>
<tr>
<th>Span</th>
<th>RMS Local $\psi$ Difference/$\Delta\psi$</th>
</tr>
</thead>
<tbody>
<tr>
<td>20% Span</td>
<td>0.091</td>
</tr>
<tr>
<td>50% Span</td>
<td>0.147</td>
</tr>
<tr>
<td>80% Span</td>
<td>0.284</td>
</tr>
<tr>
<td>Spanwise</td>
<td>0.068</td>
</tr>
</tbody>
</table>

The spanwise trailing edge work distribution in the simplified stage has an RMS difference of 2.6% from the mass averaged overall work coefficient; the process is set to create a spanwise uniform trailing edge work coefficient, this shows that this has been largely achieved.

A decrease in desired performance accuracy between the simplified stage created here and the example fan stage based on the Pratt & Whitney 1500G engine created in part 1\textsuperscript{6} is seen; for example the accuracy of desired FPR-1 decreases by 2.58%, and the non-uniformity of the spanwise trailing edge work distribution increases by 2%. This decrease in accuracy is caused by the change in radius through the tip section of the rotor. This change causes the axial velocity in the tip region to increase which leads to a radially outward shift in mass flux as shown in Figure 3-3; this shift lowers the axial velocity in the mid-span region, which modifies the velocity triangles and results in an increase in the work coefficient near mid-span.

\textsuperscript{6}Chapter 2 of this thesis
This simplified stage shows comparable performance to NASA Stage 67 at design condition and therefore provides an adequate foundation to continue with the performance comparison of the two stages with non-uniform inflow.

3.3 Numerical Setup of Full Wheel Crosswind Simulations

The simulation is setup to model pure crosswind at sea level with no forward movement, similar to the conditions seen at the start of a takeoff roll; Ansys CFX v18.2 [9] is used as the solver. The simulations are steady state, and use the shear-stress-transport turbulence model [9]. The computational domain has the outer boundaries at 25 engine diameters away from the fan axis as schematically illustrated in Figure 3-4. The cylindrical computational domain comprises four separate boundary conditions on its outer surface, since CFX lacks a true far-field boundary condition. The curved surface is divided into two equally-sized boundaries, one inlet and one
outlet. At the inlet, the static pressure and crosswind velocity are imposed, and the other is an outlet with the same imposed static pressure. The boundaries perpendicular to the fan axis are openings, which in CFX means the boundary can act as either an inlet or an outlet. When the flow leaves the domain across these boundaries, the static pressure is set to the same value as on the inlet and outlet boundaries. Flow entering across this boundary is treated as a total pressure (set as the same value as the static pressure) from which the static pressure is calculated. In these simulations the static pressure was taken as sea level standard. A mass flow outlet boundary condition is set at the outlet within the fan stage. The hub and casing curves are set as no slip walls everywhere except downstream of the rotor leading edge where they switch to zero shear stress walls. This is done as the process described in part 17 makes the assumption of zero losses within the turbomachinery. The spinner nose is a no slip rotating wall with the same rotation speed of the rotor blades. The crosswind velocity is varied.

![Computational domain for crosswind simulations](image)

Figure 3-4: Computational domain for crosswind simulations. (a) Side view, (b) front view, and (c) zoomed in view of nacelle and fan stage.

The nacelle geometry was generated by Bombardier Aerospace for this analysis and is similar in design to those seen on modern wide-body aircraft. Figure 3-5

---

7Chapter 2 of this thesis
shows the meridional profile of the nacelle used. To avoid sharp corners that have been known to cause issues within numerical methods [9] the outside of the nacelle is extended axially outwards to the downstream boundary.

![Diagram](image)

Figure 3-5: Nacelle casing used within full wheel simulations, where the dashed line represent the casing extended axially downstream and the dashed-dotted line is the rotation axis

A structured mesh containing $13.5 \times 10^6$ nodes is used for the full domain and was generated using Pointwise v18.0 [10]. A cell growth rate of 1.1 is used at all wall boundaries. The grid used downstream of the fan leading edge is similar to that used in the uniform inflow slice from the prior NASA Stage 67 simulations described in part 1\textsuperscript{8}. The automatic wall treatment option within Ansys CFX v18.2 is used; this treatment is $y^+$ insensitive, however it is recommended that at least ten cells exist in the boundary layer if this option is used therefore all results are checked to ensure this condition is met.

These full wheel simulations are relatively computationally expensive due to larger cell counts. This causes two main issues, the first being that the grid independence study itself can become expensive, and the second being that to find the separation point many iterations on the crosswind velocity may be necessary. To maintain a lower computational cost a method presented in Roache [11] which is based on the use of Richardson extrapolation can be used to determine the expected error based on the grid used. This method involves performing two or more simulations with successively

\footnote{Chapter 2 of this thesis}
finer grids, and assumes that the change in the results should asymptotically approach zero as grid cell counts increase to infinity.

Three grid levels where tested with \( U^* = 0.232 \), where \( U^* \) is a ratio between the crosswind velocity and mass averaged fan leading edge axial velocity, and is described in more detail later. This value was selected as it was large enough to generate flow separation, however low enough that the flow reattaches before the rotor leading edge. The variables of interest in the independence study are the separation size (defined by the percent area where the shear stress is less than zero in the axial direction of the total area between the nacelle lip and the rotor leading edge), and FPR. Figure 3-6 shows the FPR and separation size at the three grid levels tested, along with the value predicted using Richardson extrapolation with a cell count of infinity. This method predicts a separation size of 3.96% with a possible error of ±0.06% and a FPR of 1.308 with a possible error of ±0.0004 when the cell count tends to infinity. To maintain lower computational cost the medium grid level is selected. The method predicts a difference in separation size and FPR between this grid and the infinite grid of 0.91% ± 0.06% and 0.0004 ± 0.0004 respectively.

![Figure 3-6: Separation size and FPR as a function of grid points used to show grid dependence](image)
3.4 Results and Discussion of Crosswind Simulations

The crosswind velocity was increased until separation was seen. We define the non-dimensional crosswind velocity to be

\[ U^* = \frac{V_y}{\overline{V}_{x,2}^M} \]  \hspace{1cm} (3.3)

where \( V_y \) is the crosswind velocity and \( \overline{V}_{x,2}^M \) is the mass averaged axial velocity at the fan face. Since discrete crosswind velocities are required in the CFD, identifying the precise velocity required for separation is challenging. Instead, we aim to find the value of \( U^* \) to within ±0.005 at which separation first appears. With the conditions tested the mass averaged axial velocity at the fan face is approximately constant which translates to a precision of roughly 0.5m/s for the crosswind velocity.

Flow separation is identified by a region where the flow is locally travelling upstream; this can be seen by a negative value of the wall shear stress in the axial direction. The point where separation first occurs in NASA Stage 67 is at \( U^* = 0.22 \). The same condition occurs at a value of 0.23 in the simplified stage, this translates to approximately a difference in crosswind velocity of 1.5m/s between the two stages. This is a percent difference of 5.13%, however this may be a slight over prediction as the difference in these two values is not much larger than the possible error due to the accuracy to which \( U^* \) is found. This difference stems from the fact that the simplified stage produces a slightly larger FPR. At this condition the separation occurs briefly before the flow reattaches and the separation area is small. The crosswind velocity is increased further until the flow remains separated up to the fan face within the NASA Stage 67 case; this occurs at \( U^* = 0.30 \). In Figure 3-7 a 180° slice of the inner nacelle
casing is unwrapped and shows the regions of separation at the conditions described prior as well as when the flow is fully attached \((U^* = 0.21)\) in both stages.

The regions of separated flow are similar between the two stages. The largest difference when comparing separation between the two stages is seen in the highly separated case. In the NASA Stage 67 case the flow separates and remains separated until the fan face, however in the simplified stage the flow reattaches slightly before the fan face. This occurs for the same reason as the reason for the delayed separation: the slightly larger FPR created by the simplified stage. This small difference in FPR is responsible for other differences between the two stages' performance with crosswind, as will be discussed later in this section.

![Figure 3-7: Areas of separated flow within the nacelle at the (a) fully attached condition, (b) separation point, and (c) highly separated condition.](image)

Figure 3-8 shows the Mach number contours on a plane tangent with the crosswind for increasing crosswind velocities. In the attached flow case the effects of the
crosswind is becoming apparent with larger flow acceleration around the nacelle lip followed by a decrease along the nacelle wall. In the highly separated case the low Mach region increases in thickness and represents an area of recirculating flow as shown by the streamlines. The effect of the simplified stage having a larger FPR is seen in the decreased thickness and axial length of the recirculating region along the nacelle wall. In the highly separated flow case the thickness of the recirculating flow decreases by 15.39% and the axial length decreases by 12.15% in the simplified stage.

![Mach number in plane tangent to crosswind velocity with attached flow and highly separated flow.](image)

Figure 3-8: Mach number in plane tangent to crosswind velocity with (a) attached flow and (b) highly separated flow.

The effect of this flow separation at the rotor leading edge is investigated next. Separation causes a decrease in stagnation pressure which is convected downstream to the rotor. In Figure 3-9 the stagnation pressure to freestream stagnation pressure ratio is shown at the fan leading edge, for both models, at the attached and highly separated flow conditions. In frame (a) the stagnation pressure distribution is mostly uniform, except within the region adjacent to the wall surfaces due to boundary layer losses, which is larger of the side of crosswind flow. In both stages the maximum thickness
of the distortion region ($P_t/P_{t,atm} < 1$) is increased by approximately 6% on the side of crosswind flow. The increase in thickness of the distortion region is relatively small and indicates that the flow is fully attached by the fan leading edge and that the crosswind does not have a large adverse effect the fan performance as will be confirmed later. In frame (b) the stagnation pressure drops due to the large separation. The difference in the minimum fan leading edge stagnation pressure between the two stages in the highly separated case is 3.4%, with the larger drop in stagnation pressure being in NASA Stage 67. The two stages show similar stagnation pressure distributions, with the location of the minimum stagnation pressure decreasing by 5.8% span in the radial direction in NASA Stage 67.

Figure 3-9: Stagnation pressure ratio at rotor leading edge with (a) attached flow and (b) highly separated flow.

To determine the impact of the flow separation on the fan rotor, we next access mass flux distributions at the fan leading edge. Figure 3-10 compares the mass flux distributions at the fan face for both stages at the fully attached condition as well as the highly separated condition. At $U^* = 0.21$ the mass distribution is relatively
uniform with a slight decrease along the walls as the velocity goes to zero, which causes a slight increase in mass flux near midspan. However, in the highly separated flow case there are larger differences in the mass flux deviations for the two models. In the regions of separation the mass flux is decreased as there is flow recirculation/reversal; this has the effect of causing the adjacent regions to have increased mass flux as the flow accelerates around the edges of the recirculating regions. This is apparent in the NASA Stage 67 case where the flow recirculation reaches the fan (seen as a negative value in the distribution plots), and a large increase is seen surrounding this region. The reason behind the slight drop in prediction performance in the highly separated case is due to the simplified stage experiencing flow reattachment slightly before the fan. This indicates that the simplified stage fan has a larger corrective effect on the distribution of mass flux, which stems from it having a larger FPR.

Figure 3-10: Mass flux distribution at the fan face for the (a) fully attached condition and (b) highly separated condition.

To further examine the effects that the separation has on the performance on the fan stage, as well as how accurately the simplified stage captures this, the absolute
flow angles and change in incidence angles are investigated at the rotor leading edge. In Figure 3-11 the absolute flow angles, $\alpha$, are shown along the 20%, 50%, and 80% span lines at the rotor leading edge for both stages at the attached ($U^* = 0.21$) and separated conditions $U^* = 0.30$. In the attached case the maximum variations in absolute flow angles are $-3.4^\circ$ and $5.7^\circ$; this confirms that the flow is largely uniform as there are only small changes in flow angles as seen both around the annulus and along the span. The maximum difference in the absolute flow angle between the two stages are $0.92^\circ$, $1.8^\circ$, and $3.2^\circ$ at the 20%, 50%, and 80% span lines respectively. In the separated case the absolute flow angles have larger fluctuations in the region of separated flow which then decrease as the flow becomes more uniform. The largest fluctuations in absolute flow angle are $20.6^\circ$ and $-14.9^\circ$ seen along the 80% span line, as expected, as this travels through the region directly affected by flow recirculation. The maximum difference in the absolute flow angles at the 20% and 50% span fractions are $3.4^\circ$ and $3.3^\circ$. A maximum difference of $23.0^\circ$ in the absolute flow angle is seen along the 80% span line at $\theta = 20^\circ$; the reason for this large difference is due to the distortion region being extended over a slightly larger $\theta$ range.

More important to consider is the incidence, as high incidence can lead to premature stall when it occurs near the tip of the fan blades [8]. In Figure 3-13 the change in incidence angle, $i$, from the incidence angle with no crosswind, $i_{U^* = 0}$, is shown. The comparison to incidence with no inlet distortion, as opposed to using the circumferential variations in incidence allows for the effect of operating at an off-design condition (crosswind) to be more clearly seen. The change in incidence angle for the simplified stage is calculated using the absolute flow angles, and the meridional velocity (shown in 3-12 normalised by blade speed) produced by the simplified stage CFD simulations; however the rotational speed is set to that of NASA Stage 67. This allows for a more direct comparison of how simplifying the stage affects the prediction of changes in incidence due to crosswind for a real machine. The incidence
further confirms what has been stated earlier, that the fan is largely unaffected at the attached condition, as the change in incidence is no more than 3.6° anywhere, and is heavily affected in the regions of separated flow as the change in incidence reaches 12.5°. The simplified stage is shown to capture the maximum change in incidence (also the location of maximum difference), which occurs in the highly separated case along the 80% span line to within 0.8° of what is found in NASA Stage 67; this indicates that the simplifications do not have a large impact on the prediction capability. It is important to note however that the change in incidence of 12.5° would almost certainly initiate stall in the physical blades, though the circumferential region over which this occurs is relatively small and would most likely cause the stall to decay.

Figure 3-11: Absolute flow angles at the fan leading edge for the (a) fully attached condition and (b) highly separated condition.
Figure 3-12: Meridional velocity normalised by the blade rotation speed at 50% span at the fan leading edge for the (a) fully attached condition and (b) highly separated condition.

Figure 3-13: Change in incidence angles from design at the fan leading edge for the (a) fully attached condition and (b) highly separated condition.
For airframers it is important to be able to measure the performance of a nacelle. To quantify the performance the quantity $DC60$ is commonly used [7, 8] and is defined as:

$$DC60 = \frac{p_{t60} - \bar{p}_t^A}{\bar{p}_t^A - p^A}$$

where $p_t$ and $p$ are the area averaged stagnation and static pressures at the fan leading edge and $p_{t,60}$ represents the lowest value of area averaged stagnation pressure over any $60^\circ$ circumferential sector at the fan leading edge. $DC60$ is a measure of inlet distortion and decreases as the fan leading edge separation increases. The $DC60$ values as well as their corresponding circumferential locations are shown in Table 3.2 for $U^* = 0.21$ and $U^* = 0.30$. In Figure 3-14 the absolute values of $DC60$ as $U^*$ varies is shown for both stages.

Table 3.2: Summary of the DC60 values.

<table>
<thead>
<tr>
<th></th>
<th>NASA Stage 67</th>
<th>Simplified Stage</th>
</tr>
</thead>
<tbody>
<tr>
<td>$DC60_{U^*=0.21}$</td>
<td>-0.0583</td>
<td>-0.0479</td>
</tr>
<tr>
<td>$\theta_{U^*=0.21}$ Range</td>
<td>-32 to 28</td>
<td>-31 to 29</td>
</tr>
<tr>
<td>$DC60_{U^*=0.30}$</td>
<td>-0.3461</td>
<td>-0.3776</td>
</tr>
<tr>
<td>$\theta_{U^*=0.30}$ Range</td>
<td>-25 to 35</td>
<td>-19 to 41</td>
</tr>
</tbody>
</table>
Figure 3-14: $DC60$ comparison between NASA Stage 67 and the simplified stage as crosswind velocity varies.

The largest difference in the $DC60$ prediction between the two stages occurs at $U^* = 0.30$ where the simplified stage over predicts $DC60$ by 9%. The simplified stage has a greater rise in $DC60$ magnitude when $U^* = 0.30$ due to the circumferential thickness of the separated region. Although the simplified stage under predicts the maximum decrease in stagnation pressure at the fan face as well as the axial length of the separated region the increase in circumferential thickness of the separated region causes the area averaged stagnation pressure of the $60^\circ$ sector to decrease, which leads to the higher $DC60$ magnitude. Figure 3-14 shows that the effect of stage simplification on the prediction of $DC60$ is minor over a wide range of $U^*$ with the only major difference being that the larger crosswind separation velocity causes the sharp increase in $DC60$ to be delayed.
3.5 Summary and Conclusions

In this paper the simplified stage body force design approach described in part 1 is used to predict crosswind separation velocities, as well as the associated effects on fan and nacelle performance. A stage based on NASA Stage 67 was generated. A comparison between this simplified stage and NASA Stage 67 is done with uniform inflow, using the Hall body force with an added compressibility correction. The simplified stage produces a 1.14% difference in FPR and approximately a 5% difference in the other variables of interest, such as thrust and overall work coefficient. Although the stage produced does not match the desired performance as closely as the example fan stage based on the Pratt & Whitney 1500G engine created in part 1, due to the increase in gas path complexity, it allows for the effects of simplifying the stage to be seen and provides a foundation for investigating non-uniform inflow without detailed stage information.

The intended use for the process described in Chapter 2 is to allow for assessing external flow-fan interactions when limited fan information is available. This is useful when airframers wish to investigate coupling between non-uniform inflows caused by off-design operation and the fan. The simplified stage and the original NASA Stage 67 were inserted into a nacelle. Both stages were run in full wheel body force simulations with varying crosswind speeds and the results were compared, and the impacts of simplifying the stage design were investigated. The simplified stage was able to predict the separation velocity to within 5.1±1.6%. The Mach number in the crosswind plane is examined, and although the height and width of the recirculation region decreases in the simplified stage the overall flow shows the same trends. Flow at the fan leading edge is reproduced well, with the minimum fan stagnation pressure and its location varying by 3.4% and 5.8% respectively between the two stages. The simplified stage does well at predicting the effect of non-uniformities on the fan, as it

\(^9\)Chapter 2 of this thesis
captures the maximum change in incidence angle to within $2.4^\circ$ at the 20%, 50%, and 80% span locations. To measure the nacelle performance the $DC60$ metric is used; the simplified stage showed a maximum difference in $DC60$ when $U^* = 0.30$ with an increase of 9%.

The intended level of fidelity of this process is to be such that it can be used for early design stages or in the case where no detailed geometry is available; the quantities examined in this paper show that the simplified stage is capable of reproducing the overall flow found when detailed geometry is used. These results show a positive indication that this process would work under other non-uniform inflow conditions, for example, angle of attack. With an angle of attack the $U^*$ would be larger when separation occurs and the simplified stage reproduces the flow more accurately before the flow separates meaning that the case modeled in this paper\textsuperscript{10}, pure crosswind, is one of the worst-case scenarios possible.

\textsuperscript{10}Chapter 3 of this thesis
3.6 Bibliography


Chapter 4

Summary, Contributions, and Future Work

In this thesis a process is described that generates a body force model of a jet engine fan stage along with the associated meridional geometry without the need for detailed stage geometry as an input. The body force models are then used to assess inlet-fan coupling interaction. In this chapter, the two works shown in this thesis are outlined with a description of how they relate to one another. The key contributions arising from this work are extracted, and recommendations for future work are discussed.

4.1 Summary

Many authors have worked on developing methods to simplify the modeling of turbomachinery, such as actuator disks and body force models; and several authors have studied the design of turbomachinery. However, very little work as been done on the combination of these two fields. This lack of previous research, as well as its practical uses, are the motivation behind the current work. Due to its possible benefits, a process was developed that requires minimal information to create a simplified model of a fan stage.
Chapter 2 addresses the development of this simplified approach, including the investigation of previous simplified models and the determining factors behind the selection of the body force model. The body force model, and an added compressibility correction is described and validated against higher fidelity bladed simulations. The intended use and the required information required for this process is explained. The 1D approach and the simplifications are stated and used to determine the appropriate gas paths based on the desired performance. The creation of blade camber surfaces are described, and the loading distributions are compared to those found in real machines. The process to predict off design performance is also detailed. Finally, an example of this process is shown, the results of which are investigated at both design and off design conditions. The process was shown to produce a stage that met the desired performance to within 1% at the design point.

Chapter 3 uses the simplified approach to predict crosswind separation, and its effects on fan and nacelle performance. The simplified approach is used to create a stage based on NASA Stage 67. The inputs for the process are explained, and the body force model produced is tested at the design condition to compare its similarity to NASA Stage 67. Both this simplified stage and NASA Stage 67 are inserted into a nacelle and full wheel crosswind simulations are run. The simplified stage produces similar overall results to those found in NASA Stage 67. It predicts the separation velocity to within approximately 5%, and captures the effect of inlet distortion on the maximum change incidence to within 0.8°. The effect on nacelle performance is measured using DC60. It was found that the maximum difference occurred with highly separated flow where the simplified stage over predicted by 9%.

These works relate to one another through the intended use of the process. Chapter 2 lays out the framework for the model generation process and demonstrates that it is capable of producing a fan stage model that meets desired performance. It also demonstrates the ability to create the model with minimal prior information. This
ability is important because the intended use for this approach is for those with little knowledge of turbomachinery or those lacking detailed stage geometry, such as the typical airframer. Chapter 3 shows the application of this approach to assess inlet-fan interaction, which would be of primary interest to an airframer. These two works therefore demonstrate the entire primary intended use of the process.

4.2 Contributions

The contribution of this thesis is a practical one, as the individual components of the process contain limited novelty, but the combination of these components offers a new way to generate the body force model under the constraints commonly seen in the aerospace industry. It allows for airframers to produce a model of a fan stage without needing extra expertise or the detailed geometry from the engine manufacturer. This ability is explained and shown in Chapter 2. The ability to produce this model allows for the airframer to assess inlet-fan interaction which is the intended main use for this process, and is demonstrated in Chapter 3.

The use of this process could be expanded into other areas of study where the fan is not the primary focus but needed in order to conduct the principal research.

4.3 Future Recommendations

This section contains recommendations for future work based on the observations made in this thesis.

The first recommendation is that a improvement be made to the axial velocity prediction. Currently it uses a 1D approach and is set to be equal along the span; it is also set equal at the leading and trailing edges of the blades. Changes in the axial velocity are seen in the regions adjacent to the walls, which leads to inaccuracies from the expected velocity triangles. A method could be developed, which uses the distance
from the wall and contraction/expansion rate of the passage to determine the change in spanwise velocity. This method could allow for more complex gas path shapes to be considered without affecting the overall accuracy in performance prediction.

The second recommendation would be to add a simple loss prediction model. Current work is being done to develop an analytical loss model [1], which would give an indication of expected losses for a relatively low computational cost. This information could be helpful during preliminary design.

It is also recommended that additional inlet distortions be investigated to further confirm that the simplified model adequately predicts the inlet-fan interactions.
4.4 Bibliography

Appendix A

Richardson Extrapolation Code

\[
\text{oneoverN} = \begin{bmatrix} \frac{1}{\text{Grid1}} & \frac{1}{\text{Grid2}} & \frac{1}{\text{Grid3}} \end{bmatrix} ; \%	ext{Grid from coarsest to finest}
\]

\[
\text{FPR} = \begin{bmatrix} \text{FPR1} & \text{FPR2} & \text{FPR3} \end{bmatrix} ; \%	ext{Corresponding variables (FPR in this example)}
\]

\[
p = \frac{\log((\text{FPR}(1) - \text{FPR}(2))/(\text{FPR}(2) - \text{FPR}(3))))/\log(r)}{\%	ext{theoretical order of convergence is } p = 2.0}
\]

\[
\text{FPR0} = \text{FPR}(3) + \frac{(\text{FPR}(3) - \text{FPR}(2))}{r^p - 1} ; \%	ext{Extrapolates to infinity grid}
\]

\[
\text{GC23} = 1.25 \times \frac{\left| \frac{(\text{FPR}(3) - \text{FPR}(2))}{\text{FPR}(3)} \right|}{r^p - 1} \times 100 ; \%	ext{Grid Convergence Index to determine error bands}
\]

\[
\text{GC12} = 1.25 \times \frac{\left| \frac{(\text{FPR}(2) - \text{FPR}(1))}{\text{FPR}(2)} \right|}{r^p - 1} \times 100 ; \%	ext{Factor of safety is 1.25 because 3 points used}
\]

\[
\text{range} = \frac{\text{GC23}}{(r^p \times \text{GC12})} ; \%	ext{check that the solutions were in the asymptotic range of convergence, should be close to 1}
\]
Appendix B

1D MATLAB Design Code

% Blade Design
clear
% Info gathered around PW1500G
% Flight Conditions
gamma=1.4;% Specific gravity
R=287.58;% J kg^-1 K^-1
T0=223;%K Static Temperature
P0=26.51000;%Pa Static Pressure
cp =1.0061000;%Pressure Coefficient
V0=230.28;%m/s design cruise velocity

% Geometric Parameters
rtip=1.85/2;%m tip radius
htr=0.3;%hub to tip ratio
fanchordratio=1/2.33318;% inverse ratio of fan blade span to fan axail coord
statorchordratio=2.25;%aspect ratio of stator
LoverD=0.25;%determines position of fan face. axial length to fan face over fan diameter from peters 0.25
pertipdelta=-0.000;%percent radius change from rotor tip LE to TE
Doyouwantellipnose=0;%0=no 1=yes
perellip =0.25;%percentage of linear spinner nose to curve over bigger= shorter
C3=0.5625;%Space bewteen Rotor TE and Stator LE Spacing variable
C1x=0.25;%Sets the level of curvature between Rotor TE and Stator LE variable. must be between 0 and 1
C5=2;%Length unitl nozzle contractionSpacing variable
Cout=2;%Length of nozzle contraction Spacing variable

% Performance Parameters
FPR=1.4;%Desired Pressure ratio
F=16.71000;%N Desired thrust
BR=18;%Number of rotor blades. If DF=0 then manually set balde numbers
BS=36;%Number of stator blades. If DF=0 then manually set balde numbers
DoyouwantouseDF=0;%Set 1 if want to use uses hub section, ensure that mid/tip sections meet requirements <0.6
\[DF_{\text{wanted}} = 0.2375; \] % Change to desired lit says no more than 0.6

\[
\text{stagemname} = 'hubcasingcurves.txt'; \text{Path for stagen input file}
\]

\[
\text{pointwisename} = 'pointwisesegment.dat'; \text{Path for pointwise file}
\]

\[
\text{offdesign} = 0; \text{Set = 1 if you want to run an offdesign prediction}
\]

\[
\text{mcortake} = 498.95; \text{kg/s}
\]

\[
P_{\text{toff}} = 101325; \text{Pa Stagnation pressure}
\]

\[
T_{\text{toff}} = 228.15; \text{K Stagnation temperature}
\]

\[
P_{19\text{off}} = 101325; \text{Pa Static pressure}
\]

\[
M19 = 1;
\]

\[
\rho0 = P0 / (R \times T0);
\]

% This calls a function which finds some flight stag quantities and basic fan geometry

\[
[M0, T0, Pt0, rhot0, rhub, A2, rmid] = \text{InitialQuantities(V0, R, T0, gamma, P0, rho0, rtip, htr)};
\]

% 1-D code that finds quantities through the engine

\[
[T2, Pt2, rhot2, Tt13, Pt13, T19, Pt19, T19, V19, P19, Tauf, mdot, A19, rhot19, rhot19, T2, P2, M2] = \text{OneDcode(R, T0, Pt0, rhot0, FPR, gamma, M19, cp, F, V0, P0, A2)};
\]

% This function finds the design coefficients at the LE

\[
[Mreltip, Mtip, Utip, \alpha, rpm, Umid, Rmid, Umhub, flowtip, stagetip, Rtip, flowmid, stagemid, Rmid, flowhub, stagehub, Rhub, V2] = \text{FindCoefficients(T2, cp, Tauf, FPR, M2, gamma, R, T2, rtip, rmid, rhub)};
\]

% This function finds overall area change required through blades

\[
[M13, rhot13, V13, A13] = \text{AreathroughStage(FPR, T19, P19, Pt19, V2, rhot19, T19, gamma, Tt13, mdot, R)};
\]

% Creates the hub/mid/casing curves

% Few hard coded numbers in this function that need may need to be added as

% Initial variable

\[
[rcas, xcas, xhub, xmid, rmid, SpinLE, RotLE, RotTE, StatLE, StatTE, Contract, Out, ContPt1, ContPt2, ContPt3] = \text{GeoCurves(rtip, pertipdelta, LoverD, htr, fanchordratio, statorchordratio, A2, A13, A19, Doyouwantellipnose, perellip, C1x, C3x, C5, Out)};
\]

\[
[M3, rhot3, V3, A3] = \text{StatorInlet(FPR, Pt19, rhot19, T19, gamma, mdot, R, rhub, rcas, StatLE)};
\]

% Converts to meridional velocities

\[
[V2mhub, V2mmid, V2mtip, V13mhub, V13mmid, V13mtip, V3nR] = \text{convertV(xhub, rhub, xmid, rmid, xcas, rcas, M2, V2, SpinLE, RotLE, RotTE, StatLE, StatTE, Contract, Out)};
\]

% This function finds the blade angles

\[
\]

newxcas = linspace(xcas(1), xcas(length(xcas)), 200);
newrcas = pchip(xcas, rcas, newxcas);
newxhub = linspace(xhub(SpinLE), xhub(length(xhub)), 200);
newrhub = pchip([xhub(SpinLE, ContPt1), xhub(ContPt3, Out)], [rhub(SpinLE, ContPt1), rhub(ContPt3, Out)]);
ContPt1) rhub(ContPt3:Out), newxhub;
testx=inspace(xhub(ContPt1),xhub((ContPt3)),200);
testx=pchip([xhub(ContPt1) xhub(ContPt3) xhub(StatLE)], [rhub(ContPt1) rhub(ContPt3) rhub(StatLE)], testx);
hold on
hubcurve=[newxhub xhub(RotLE) xhub(RotTE) xhub(StatLE) xhub(StatTE)];
    newrhub rhub(RotLE) rhub(RotTE) rhub(StatLE) rhub(StatTE);
    [temp, order] = sort(hubcurve(1,:));
    answer = hubcurve(:, order); newxhub=answer(1,:); newrhub=answer(2,:);
cascurve=[newxcas xcas(RotLE) xcas(RotTE) xcas(StatLE) xcas(StatTE)];
    newrcas rcas(RotLE) rcas(RotTE) rcas(StatLE) rcas(StatTE);
    [temp, order] = sort(cascurve(1,:));
    answer = cascurve(:, order); newxcas=answer(1,:); newrcas=answer(2,:);
newxhub=[0 newxhub] ; newrhub=[0 newrhub] ;
plot(newxcas, newrcas, newxhub, newrhub);
plot([xhub(RotLE) xcas(RotLE)], [rhub(RotLE) rcas(RotLE)], [xhub(RotTE) xcas(RotTE)], [rhub(RotTE) rcas(RotTE)], [xhub(StatLE) xcas(StatLE)], [rhub(StatLE) rcas(StatLE)], [xhub(StatTE) xcas(StatTE)], [rhub(StatTE) rcas(StatTE)]);
axis equal;

% % % % % % % % % % % % % % Does not need to iterate% % % % % % % % % % % % % % % %
%This function writes data in format for easy copy to stagen.dat
WriteforStagen(xhub, rhub, xcas, rcas, perspan, Rhubangles, Rmidangles, Rtipangles, Shubangles, Smidangles, Stipangles, xmid, rmid, RotLE, RotTE, StatLE, StatTE, stagenname);
%This function writes segment file for pointwise
PointwiseInput(newxhub, newrhub, newxcas, newrcas, xhub, rhub, xcas, rcas, RotLE, RotTE, StatLE, StatTE, pointwisename);
% %Check to see takeoff conditions
if offdesign==1;
    [mcordes, wscale, wcordes, wtakeoff, FPRtakeoff, chokedratio, Thursttakeoff] = takeoffspeed(mcortake, Tt0, Pt0, mdot, F, gamma, R, A19, A2, Rhubangles, Rmidangles, Rtipangles, Shubangles, Smidangles, Stipangles, xmid, rmid, RotLE, RotTE, StatLE, StatTE, stagenname);
end

function [M0, Tt0, Pt0, rho0, rhub, A2, rmid] = InitialQuantities(V0, R, gamma, P0, rho0, rtip, htr)
%This finds the function finds some initial quantities
M0=V0/sqrt(gamma*R*T0);%This is flight Mach Number
Tt0=T0*(((gamma−1)/2)*M0ˆ2);%This is flight Stag Temp
Pt0=P0*(((gamma−1)/2)*M0ˆ2).^(gamma/(gamma−1));%This is flight Stag Press
rho0=rho0*num(1+((gamma−1)/2)*M0^2)) %This is flight Stag Density
rhub=rtip*htr;%This is radius of hub at fan face
rmid=(rtip+rhub)/2;
A2=pi*rtip^2−pi*rhub^2;%This is Area of fan face
end

function [Tt2, Pt2, rho2, Tt13, Pt13, Tt19, Pt19, T19, V19, P19, tauf, mdot, A19, rho19, rhot19, T2, P2, M2] = OneDcode(R, Tt0, Pt0, rho0, FPR, gamma, M19, cp, F , V0, P0, A2)
%This finds the quantities through the engine
% Setting fan face stag=flight stag, assume no inlet losses
Tt2=Tt0;
Pt2=Pt0;
rhot2=rhot0;
% Find stag temp/press rise based on npoly and FPR
Tt13=Tt2*(FPR)\((\gamma-1)/(\gamma))\);
Pt13=FPR*Pt2;
% Setting stag stator TE=stag outlet, assume no losses through nozzle
Tt19=Tt13;
Pt19=Pt13;
% Finding static quantities at outlet
T19=Tt19/(1+((\gamma-1)/2)*M19^2);
V19=M19*sqrt(\(\gamma*R*T19\));
P19=Pt13*0.5283;% Gives static pressure required for choke
tau=Tt13/Tt2;
% Finds mass flow based on of thrust requirements and outlet choke conditions
mdot=F./(V19-V0+((P19-P0)*sqrt(Tt19))/(Pt19*(sqrt(\(\gamma*R\))*T19)))*((\gamma+1)/2)\((\gamma-1)/(2*(\gamma-1)))\);
% Finds outlet area based required for choking
A19=mdot*sqrt(Tt19)/(Pt19*sqrt(\(\gamma*R\))*((\gamma+1)/2)*(\gamma/(\gamma-1)));
% Finds Densities at outlet
rhot19=mdot/A19*V19;
rhot19=rhot19*(1+((\gamma-1)/2)*M2^2)\((-1*\gamma-1)/(2*\gamma-1))\);
% Solves for the Mach number at the fan face
fun=@(x)M2fun(x,mdot,A2,2,Tt2,\(\gamma,R\));
x0=[0.01];
M2=fzero(fun,x0,options);
% Finds static quantities at the fan face
T2=Tt2/(1+((\gamma-1)/2)*M2^2);
P2=Pt2/((1+((\gamma-1)/2)*M2^2)\((\gamma/(\gamma-1)))\);
end

function Fun=M2fun(x,mdot,A2,2,Tt2,\(\gamma,R\))
M2=x;
Fun(1)=mdot-((A2*Pt2)/(sqrt(Tt2)))*sqrt(\(\gamma*R\))*M2*(1+((\gamma-1)/2)*M2^2)^((-1*\gamma-1)/(2*\gamma-1)));
end

function [Mreltip,Mtip,Upit,w,rpm,Umid,Umid,Mhub,flowtip,stage1tip,Rtip,flowmid,stage1mid,Rmid,flowhub,stagehub,Rhub,V2]=
FindCoefficients(Tt2,cp,tau,FPR,M2,\(\gamma,R,T2,rtip,rmid,rhub\));
% This finds flow and stage loading coefficient as well as Reaction
Mreltip=(1.4/1.6)*FPR;% Finds Relative Mach at tip based on ratio from modern engines
Mtip=sqrt(Mreltip^2-M2^2);% Finds tip Mach number
Upit=Mtip*sqrt(\(\gamma*R*T2\));% Finds tip velocity, assumes radial constant temp at inlet
V2=M2*sqrt(\(\gamma*R*T2\));% This finds the axial velocity at LE
w=Upit/rtip;% Finds rotational velocity
rpm=w/2/pi*60;% Converts to rpm
Umid=w*rmid;% Finds velocity at LE midspan
M_{\text{mid}} = \frac{U_{\text{mid}}}{\sqrt{\gamma R T_2}}; \% \text{Finds Mach at mid LE}
U_{\text{hub}} = w_r h_{\text{hub}}; \% \text{Finds velocity at LE hub}
M_{\text{hub}} = \frac{U_{\text{hub}}}{\sqrt{\gamma R T_2}}; \% \text{Mach at hub LE}
flow_{\text{tip}} = \frac{V_2}{U_{\text{tip}}}; \% \text{Flow coefficient at LE tip}
stagetip = c_p (\tau_a - 1) \frac{R_{T_2}}{U_{\text{tip}}^2}; \% \text{Stage loading coefficient at tip LE}
R_{\text{tip}} = 1 - \frac{stagetip}{2}; \% \text{Reaction at tip LE}
flow_{\text{mid}} = \frac{V_2}{U_{\text{mid}}}; \% \text{Flow coefficient at LE mid}
stagemid = c_p (\tau_a - 1) \frac{R_{T_2}}{U_{\text{mid}}^2}; \% \text{Stage loading coefficient at mid LE}
R_{\text{mid}} = 1 - \frac{stagemid}{2}; \% \text{Reaction at mid LE}
flow_{\text{hub}} = \frac{V_2}{U_{\text{hub}}}; \% \text{Flow coefficient at LE hub}
stagehub = c_p (\tau_a - 1) \frac{R_{T_2}}{U_{\text{hub}}^2}; \% \text{Stage loading coefficient at hub LE}
R_{\text{hub}} = 1 - \frac{stagehub}{2}; \% \text{Reaction at hub LE}

function [M13, rhot13, V13, A13] = AreathroughStage(FPR, T19, P19, Pt19, V2, 
rhot19, Tt19, gamma, Tt13, mdot, R)
% Finds static values at stations through engine to find required areas
% Equations assume no loss between stator TE and outlet
% Assume constant axial velocity through stage
fun=@(x)M13fun(x, V2, Tt19, gamma, R);
x0 = [0.01];
options = optimset(’Display’, ’off’);
M13 = fsolve(fun, x0, options);
rhot13 = rhot19; % Assuming no losses
V13 = V2; % Constant axial velocity at LE and TE
A13 = mdot * \sqrt{\frac{T19}{(Pt19* (\sqrt{\gamma R}))}}*M13*(1+((\gamma-1)/2)*M13^2)
    \cdot (-1*(((\gamma+1)/(2*(\gamma-1)))))
end

function Fun = M13fun(x, V2, Tt19, gamma, R)
M13 = x;
Fun(1) = M13 - (V2/((\gamma R T19)/(1+((\gamma-1)/2)*M13^2)))^0.5;
end

function [rcas, xcas, xhub, rhub, xmid, rmid, SpinLE, RotLE, RotTE, StatLE, StatTE, 
Contract, Out, ContPt1, ContPt2, ContPt3] = GeoCurves(rtip, pertipdelta, 
LoverD, ltr, fanchordratio, statorchordratio, A2, A13, A19, 
Doyouwantellipnose, perellip, C1x, C3, C5, Cout)
% This is used to generate the hub/mid/casing curves
Spinpts=50; \% # of axial pts on spinner
Rotpts=30; \% # of axial pts in rotor
Statpts=30; \% # of axial pts in stator
SpinLE=2;
RotLE=SpinLE+Spinpts;
RotTE=RotLE+Rotpts;
ContPt1=RotTE+1;
ContPt2=ContPt1+1;
ContPt3=ContPt2+1;
StatLE=ContPt3+1;
StatTE=StatLE+Statpts;
Contract=StatTE+1;
Out=Contract+1;
% Pt 1 x
xcas(1,1) = 0;
%Rot LE x
xcas(1,RotLE)=xcas(1,1)+LoverD*rtip*2;

%RotLE–RotTE
xcas(1,RotLE:RotTE)=linspace(xcas(1,RotLE),xcas(1,RotLE)+(rtip−htr*rtip)
  +fanchordratio,Rotpts+1);

%Contraction 1 x
xcas(1,ContPt1)=xcas(1,RotTE)+0.25*C1x*(xcas(1,RotTE)−xcas(1,RotLE));
xcas(1,ContPt2)=xcas(1,RotTE)+C1x*(xcas(1,RotTE)−xcas(1,RotLE));
xcas(1,ContPt3)=xcas(1,RotTE)+2.5*C1x*(xcas(1,RotTE)−xcas(1,RotLE));
xhub=xcas;%Set to use same x pts for hub/mid/casing
xmid=xhub;

%Casing Curve, currently constant r of rtip except at Rotor TE and outlet
rcas(1,:RotLE)=rtip;
rcas(1,RotLE:RotTE)=linspace(rcas(1,RotLE),rtip−pertipdelta*rtip,Rotpts+1);
rcas(1,RotTE:Contract)=rcas(RotTE);

%Hub Curve
%Pt 1 & 2
rhub(1,:SpinLE)=0;
%Rot LE
rhub(1,RotLE)=sqrt((pi*rtip^2−A2)/pi);

%Rot LE to TE
rhub(1,RotLE:RotTE)=linspace(rhub(1,RotLE),sqrt((rcas(1,RotTE)) ^2−(A13/pi)),Rotpts+1);
mR=(rhub(1,RotTE)−rhub(1,RotLE))/(xhub(1,RotTE)−xhub(1,RotLE));
bR=rhub(1,RotLE)−mR*(xhub(1,RotLE));

%Spin LE
xhub(1,SpinLE)=(0−bR)/mR;

rcas(1,SpinLE:RotLE)=linspace(xhub(SpinLE),xhub(RotLE),Spinpts+1);
rhub(1,SpinLE:RotLE)=mR*xhub(1,SpinLE:RotLE)+bR;

xcas=xcas;%Set to use same x pts for hub/mid/casing
xmid=xhub;

%Contraction 1 r
rhub(1,ContPt1)=mR*xhub(1,ContPt1)+bR;
rhub(1,ContPt2)=mR*xhub(1,ContPt2)+bR;
rhub(1,ContPt3)=mR*xhub(1,ContPt2)+bR;

%StatLE x, r
xcas(1,StatLE)=xcas(1,ContPt2)+C3*((rtip−htr*rtip)*fanchordratio);

rhub(1,StatLE:Contract)=rhub(1,ContPt3);

%StatLE – StatTE x
xcas(1,StatLE:StatTE)=linspace(xcas(1,StatLE),xcas(1,StatLE)+(rcas(1,
  StatLE)−rhub(1,StatLE))/statorchordratio,Statpts+1);

%Contraction 2 x
xcas(1,Contract)=xcas(1,StatTE)+C5*(xcas(1,RotTE)−xcas(1,RotLE));

%Out x
xcas(1,Out)=xcas(1,Contract)+Cout*(xcas(1,RotTE)−xcas(1,RotLE));

rhub=xcas;%Set to use same x pts for hub/mid/casing
xmid=xhub;

%Stator TE to Contract r
rhub(1,StatLE:Contract)=rhub(1,ContPt3);

%Mid Span radius
rmid(1,:Contract)=(rcas(1,1:Contract)+rhub(1,1:Contract))./2;
rmid(1,Out)=rmid(1,Contract);
rhub(1,Out) = \(-1\ast((A19-4\pi rmid(1,Out)^2)/(4\ast rmid(1,Out)\pi))\);
rcas(1,Out)=2\ast rmid(1,Out)-rhub(1,Out); %Find rmid first
% Spinner Nose if elliptical chosen
if Doyouwantellipnose==1;
xhub(1,SpinLE:RotLE)=linspace(perelli***(xhub(RotLE)-xhub(SpinLE))+
xhub(SpinLE),xhub(RotLE),Spinpts+1);
fun=@(x) hfun(x,mR,bR,rhub,RotLE,xhub,SpinLE);
x0=[0.01];
options = optimset('Display','off');
h=fsove(fun,x0,options);
b=sqrt(((rhub(RotLE))^2)/(1-((xhub(RotLE)-h)^2)/((xhub(SpinLE)-h)^2)));
a=xhub(SpinLE)-h;
rhub(1,SpinLE:RotLE)=sqrt(((b^2)*(((1-((xhub(SpinLE:RotLE)-h)^2))/(a^2))));
end
end

function Fun = hfun(x,mR,bR,rhub,RotLE,xhub,SpinLE)
h=x;
Fun=-2\ast mR\ast bR\ast h-(bR^2)+(((rhub(RotLE))^2)/(1-((xhub(RotLE)-h)^2)/((
xhub(SpinLE)-h)^2)))-(mR^2)*(h^2)+(((xhub(SpinLE)-h)^2)*mR^2));
end

function [M3, rhot3 , V3, A3] = StatorInlet(FPR, Pt19 , rhot19 , Tt19 , gamma, mdot ,
\ R, rhub, rcas, StatLE);
% Finds info at stator inlet
A3=\pi \ast rcas(StatLE)^2 - \pi \ast rhub(StatLE)^2;
fun=@(x) M3fun(x,mdot,A3,Pt19,Tt19,gamma,R);
x0=[0.01];
options = optimset('Display','off');
M3=fsove(fun,x0,options);
rhot3=rhot19; % Assuming no losses
T3=Tt19./(1+((gamma-1)/2)*M3^2);
V3=M3*sqrt(gamma*R*T3);
end

function Fun = M3fun(x,mdot,A3, Pt19, Tt19, gamma, R)
M3=x;
Fun(1)=mdot-((A3*Pt19)/(sqrt(Tt19)))*sqrt(gamma/R)*M3*(1+((gamma-1)/2)*
M3^2)*((gamma-1)/(2*(gamma-1)));
end

function [ V2mhub, V2mmid, V2mtip, V13mhub, V13mmid, V13mtip , VmR] = convertV(
xhub, rhub, xmid, rmid, xcas, rcas, M2, V2, SpinLE, RotLE, RotTE, StatLE, StatTE ,
Contract, Out)
% This converts the velocities to meridional velocities to be used
V2mhub=abs(V2/sind(atand((xhub(RotLE)-xhub(RotLE-1))/(rhub(RotLE)-
RotLE-1)))));
V2mmid=abs(V2/sind(atand((xmid(RotLE)-xmid(RotLE-1))/(rmid(RotLE)-
rmid(RotLE-1)))));
V2mtip=abs(V2/sind(atand((xcas(RotLE)-xcas(RotLE-1))/(rcas(RotLE)-
rcas(RotLE-1)))));
81
\[ \text{V13mhub} = \text{abs}(\text{V2} / \sin(\text{atan}((\text{xhub}(\text{RotTE}) - \text{xhub}(\text{RotTE} - 1)) / ((\text{rhub}(\text{RotTE}) - \text{rhub}(\text{RotTE} - 1)))))); \]

\[ \text{V13mmid} = \text{abs}(\text{V2} / \sin(\text{atan}((\text{xmid}(\text{RotTE}) - \text{xmid}(\text{RotTE} - 1)) / ((\text{rmid}(\text{RotTE}) - \text{rmid}(\text{RotTE} - 1)))))); \]

\[ \text{V13mtip} = \text{abs}(\text{V2} / \sin(\text{atan}((\text{xcas}(\text{RotTE}) - \text{xcas}(\text{RotTE} - 1)) / ((\text{rcas}(\text{RotTE}) - \text{rcas}(\text{RotTE} - 1)))))); \]

\[ j = \text{RotLE} + 1; \]

\[ \text{for } i = 1: \text{RotTE} - \text{RotLE} - 1; \]

\[ \text{VmR}(i) = \text{abs}(\text{V2} / \sin(\text{atan}((\text{xhub}(j) - \text{xhub}(j - 1)) / ((\text{rhub}(j) - \text{rhub}(j - 1)))))); \]

\[ j = j + 1; \]

\[ \text{end} \]

\[ \text{end} \]

\[ \text{function } [\text{perspan}, \text{Rhubangles}, \text{Rmidangles}, \text{Rtipangles}, \text{Shubangles}, \text{Smidangles}, \text{Stipangles}, \text{B2hub}, \text{B2mid}, \text{B2tip}, \text{devhub}, \text{devmid}, \text{devtip}, \text{devhubS}, \text{devmidS}, \text{devtipS}, \text{alpha2hub}, \text{alpha2mid}, \text{alpha2tip}, \text{flowhub13}, \text{cthetat2hub}, \text{cthetat2mid}, \text{cthetat2tip}] = \text{BladeAngles}((\text{rhub}, \text{rmid}, \text{rcas}, \text{w}, \text{V2}, \text{cp}, \text{tauf}, \text{Tt2}, \text{Uhub}, \text{Umid}, \text{Utip}, \text{gamma}, \text{R}, \text{T2}, \text{M2}, \text{xcas}, \text{BR}, \text{BS}, \text{V2mhub}, \text{V2mmid}, \text{V2mtip}, \text{V13mhub}, \text{V13mmid}, \text{V13mtip}, \text{SpinLE}, \text{RotLE}, \text{StatLE}, \text{StatTE}, \text{Contract}, \text{Out}, \text{V3}, \text{DoyouwanttouseDF}, \text{DFwanted}, \text{V19}) \]

\%This function finds the blade angles
\%Assume inlet flow angle is 0
\%This sets the exit blade angle to radius of TE
\text{Uhub13} = \text{w} \times \text{rhub13}; \%Finds new velocity at TE radius
\text{flowhub13} = \text{V13mhub} / (\text{Uhub13}); \%Flow coeff at this TE radius
\text{stagehub13} = \text{cp} \times (\text{tauf} - 1) \times \text{Tt2} / (\text{Uhub13}^2); \%Stage loading at TE radius
\text{Rhub13} = 1 - \text{stagehub13} / 2; \%Reaction at TE radius
\text{B2hub} = \text{atan}(\text{(stagehub13 - 1)} / (-1 \times \text{flowhub13})); \%This gives exit flow angle
\text{B1hub} = \text{atan}((\text{Uhub} / \text{V2mhub})); \%This gives inlet flow angle
\%Repeated for mid span
\text{Umid} = \text{w} \times \text{rmid13}; \%Rhumb13 = \text{rhub}(\text{RotLE}); \%This sets the exit blade angle to radius of TE
\text{Umid13} = \text{w} \times \text{rmid13}; \%Finds new velocity at TE radius
\text{flowmid13} = \text{V13mmid} / (\text{Umid13}); \%Flow coeff at this TE radius
\text{stagemid13} = \text{cp} \times (\text{tauf} - 1) \times \text{Tt2} / (\text{Umid13}^2); \%Stage loading at TE radius
\text{Rmid13} = 1 - \text{stagemid13} / 2; \%Reaction at TE radius
\text{B2mid} = \text{atan}((\text{(stagemid13 - 1)} / (-1 \times \text{flowmid13}))); \%This gives exit flow angle
\text{B1mid} = \text{atan}((\text{Umid} / \text{V2mmid})); \%This gives inlet flow angle
\%Repeated for tip span
\text{rtip13} = \text{rcas}(\text{RotLE}); \%Rhumb13 = \text{rhub}(\text{RotLE}); \%This sets the exit blade angle to radius of TE
\text{Utip13} = \text{w} \times \text{rtip13}; \%Finds new velocity at TE radius
\text{flowtip13} = \text{V13mtip} / (\text{Utip13}); \%Flow coeff at this TE radius
\text{stagetip13} = \text{cp} \times (\text{tauf} - 1) \times \text{Tt2} / (\text{Utip13}^2); \%Stage loading at TE radius
\text{Rtip13} = 1 - \text{stagetip13} / 2; \%Reaction at TE radius
\text{B2tip} = \text{atan}((\text{(stagetip13 - 1)} / (-1 \times \text{flowtip13}))); \%This gives exit flow angle
\text{B1tip} = \text{atan}((\text{Utip} / \text{V2mtip})); \%This gives inlet flow angle
\%This section finds stator LE angles for hub/mid/tip
\text{wtheta2tip} = \text{V13mtip} \times \text{tan}(\text{B2tip}); \%ctheta2tip = \text{Utip13} - \text{wtheta2tip}; \%This gives exit flow angle
\text{alpha2tip} = \text{atan}(\text{cthetat2tip} / \text{V3}); \%This gives inlet flow angle
\text{wtheta2mid} = \text{V13mmid} \times \text{tan}(\text{B2mid}); \%This gives inlet flow angle
\text{cthetat2mid} = \text{Umid13} - \text{wtheta2mid}; \%This gives exit flow angle
\text{alpha2mid} = \text{atan}(\text{cthetat2mid} / \text{V3}); \%
wtheta2hub = V13mhub * \tan(B2hub) \\
ctheta2hub = Uhub13 - wtheta2hub \\
alpha2hub = \arctan(\frac{ctheta2hub}{V3}) \\
% This is the outlet angle of the stator 
alpha3 = 0; \\
% Rotor 
% This finds rotor chord 
chordhub = \sqrt{(xhub(RotTE) - xhub(RotLE))^2 + (rhub(RotTE) - rhub(RotLE))^2}; 
chordmid = \sqrt{(xmid(RotTE) - xmid(RotLE))^2 + (rmid(RotTE) - rmid(RotLE))^2}; 
chordcas = \sqrt{(xcas(RotTE) - xcas(RotLE))^2 + (rcas(RotTE) - rcas(RotLE))^2}; 
% This finds stagger angle 
zwe = 0.8; % This is the optimal zweifel's loading comes from nasa paper 
soldxhub = ((2 * \cosd(B2hub)) / (zwe * \cosd(B1hub))) * \sin(B1hub - B2hub); 
ahub = (soldxhub - \sin(B1hub) * \cos(B1hub) + \sin(B2hub) * \cos(B2hub)) / (\sin(B1hub) - \sin(B2hub)); 
fun = @(x) boverlfun(x, B2hub, B1hub); 
x0 = [0.01]; 
options = optimset('Display', 'off'); 
boverl = fsolve(fun, x0, options); 
staggerhubnew = B2hub - 180/pi * \arctan(boverl / 0.390625); 
soldxmid = ((2 * \cosd(B2mid)) / (zwe * \cosd(B1mid))) * \sin(B1mid - B2mid); 
amid = (soldxmid - \sin(B1mid) * \cos(B1mid) + \sin(B2mid) * \cos(B2mid)) / (\sin(B1mid) - \sin(B2mid)); 
% staggermidnew = \arctan(cmd/soldxmid); 
fun = @(x) boverlfun(x, B2mid, B1mid); 
x0 = [0.01]; 
options = optimset('Display', 'off'); 
boverl = fsolve(fun, x0, options); 
staggermidnew = B2mid - 180/pi * \arctan(boverl / 0.390625); 
soldtip = ((2 * \cosd(B2tip)) / (zwe * \cosd(B1tip))) * \sin(B1tip - B2tip); 
atip = (soldtip - \sin(B1tip) * \cos(B1tip) + \sin(B2tip) * \cos(B2tip)) / (\sin(B1tip) - \sin(B2tip)); 
ctip = \sin(B1tip) - atip * (\cos(B1tip) - \cos(B2tip)) - \sin(B2tip)^2; 
fun = @(x) boverlfun(x, B2tip, B1tip); 
x0 = [0.01]; 
options = optimset('Display', 'off'); 
boverl = fsolve(fun, x0, options); 
staggertipnew = B2tip - 180/pi * \arctan(boverl / 0.390625); 
if DoyouwanttoUseDF==1; 
fun = @(x) BRfun(x, rhub, RotTE, chordhub, staggerhubnew, V13mhub, B2hub, 
V2mhub, B1hub, ctheta2hub, DFwanted); 
x0 = [0.01]; 
options = optimset('Display', 'off'); 
BR = fsolve(fun, x0, options); 
BR = double(BR); 
BR = ceil(BR); 
end 
% Finds blade metal angle 
bmiahub = B1hub - 2; % Assuming 2 incidence 
% syms bmeahub; % blade metal exit angle 
solverhub = (2 * pi * ((rhub(RotTE))/BR)/(chordhub/cosd(staggerhubnew))); 
aoverl = 0.375;
\% mhub = 0.23*(2*aoverl)^2 + B2hub/500;
\% eqnhub = bmeahub - ((B2hub - bmiahub*mhub*sqrt(soverlhub))/(1-mhub*sqrt(soverlhub))) == 0;
devehub = (0.23*(2*aoverl)^2 + B2hub/500)*((B1hub-2)-B2hub)*soverlhub ^ 0.5;
\% Carter
bmeahub = B2hub - devhub;
bmiahub = B1mid - 2;
\% syms bmeamid; \% blade metal exit angle
soverlmid = (2*pi*((rmid(RotTE)))/BR)/(chordmid/cosd(staggermidnew));
aoverl1 = 0.375;
\% mmid = 0.23*(2*aoverl)^2 + B2mid/500;
\% eqnmid = bmeamid - ((B2mid - bmiamid*mmid*sqrt(soverlmid))/(1-mpid*sqrt(soverlmid))) == 0;
devidm = (0.23*(2*aoverl)^2 + B2mid/500)*((B1mid-2)-B2mid)*soverlmid ^ 0.5;
\% Carter
bmeamid = B2mid - devmid;
bmiatip = B1tip - 2;
\% syms bmeatip; \% blade metal exit angle
soverltip = (2*pi*((rcas(RotTE)))/BR)/(chordcas/cosd(staggettnew));
aoverl = 0.375;
\% mtip = 0.23*(2*aoverl)^2 + B2tip/500;
\% eqntip = bmeatip - ((B2tip - bmiatip*mtip*sqrt(soverltip))/(1-mtip*sqrt(soverltip))) == 0;
devtip = (0.23*(2*aoverl)^2 + B2tip/500)*((B1tip-2)-B2tip)*soverltip ^ 0.5;
\% Carter
bmeatip = B2tip - devtip;
\% Stator
chordhubS = sqrt((xhub(StatTE)-xhub(StatLE))^2+(rhub(StatTE)-rhub(StatLE))^2);
chordmidS = sqrt((xmid(StatTE)-xmid(StatLE))^2+(rmid(StatTE)-rmid(StatLE))^2);
chordcasS = sqrt((xcas(StatTE)-xcas(StatLE))^2+(rcas(StatTE)-rcas(StatLE))^2);
zwe = 0.8; \% This is the optimal zweifel's loading comes from nasa paper
soldxhubS = ((2*cosd(alpha3))/(zwe*cosd(alpha2hub)))*sind(alpha2hub-alpha3);
ahubS = soldxhubS - sind(alpha2hub)*cosd(alpha2hub)+sind(alpha3)*cosd(alpha3)/(sind(alpha2hub)-sind(alpha3));
chubS = sind(alpha2hub)^2-ahubS*(cosd(alpha2hub)-cosd(alpha3)) - sind(alpha3)^2;
staggerhubnewS = tand(chubS/soldxhubS);
\% staggerhubnewS = 0.5*(alpha2hub+alpha3) \% For circ arc camber
soldxmidS = ((2*cosd(alpha3))/(zwe*cosd(alpha2mid)))*sind(alpha2mid-alpha3);
amidS = soldxmidS - sind(alpha2mid)*cosd(alpha2mid)+sind(alpha3)*cosd(alpha3)/(sind(alpha2mid)-sind(alpha3));
cmidS = sind(alpha2mid)^2-amidS*(cosd(alpha2mid)-cosd(alpha3)) - sind(alpha3)^2;
staggermidnewS = tand(cmidS/soldxmidS);
\% staggermidnewS = 0.5*(alpha2mid+alpha3) \% For circ arc camber
soldxtipS = ((2*cosd(alpha3))/(zwe*cosd(alpha2tip)))*sind(alpha2tip-alpha3);
atipS = soldxtipS - sind(alpha2tip)*cosd(alpha2tip)+sind(alpha3)*cosd(alpha3)/(sind(alpha2tip)-sind(alpha3));
ctipS=sind(alpha2tip)^2-atipS*(cosd(alpha2tip)-cosd(alpha3))-sind(alpha3)^2;
staggertipnewS=atan2d(ctipS/soldxtipS);
staggertipnewS=0.5*(alpha2tip+alpha3)；%For circ arc camber
if DoyouwanttouseDF==1:
    fun= @(x) BSfun(x, rhub, StatTE, chordhubS, staggerhubnewS, V19, alpha3, V13mhub, B2hub, ctheta2hub, DFwanted);
x0=[0.01];
options = optimset('Display','off');
BS=fsolve(fun,x0,options);
BS=double(BS);
end
bmiahubS=alpha2hub;%Assuming 0 incidence
%syms bmeahubS;% blade metal exit angle
soverlhubS=(2*pi*((rhub(StatLE)+rhub(StatTE))/2)/BS)/(chordhubS/cosd(staggerhubnewS));
aoverlS=0.375;%location or max camber over chord 0.5 for circ arc mine is /2
%mhubS=0.23*(2*aoverlS)^2 + alpha3/500;
%eqnhubS=bmeahubS-((alpha3-bmiahubS*mhubS*sqrt(soverlhubS))/(1-mhubS*sqrt(soverlhubS)))==0;
%bmeahubS=vpasolve(eqnhubS,bmeahubS);
%bmeahubS=double(bmeahubS);
dehubS=(0.23*(2*aoverlS)^2 + alpha3/500)*(alpha2hub-alpha3)*soverlhubS^0.5; %Carter
bmeahubS=alpha3-devhubS;
bmiamidS=alpha2mid;%Assuming 0 incidence
%syms bmeamidS;% blade metal exit angle
soverlmidS=(2*pi*((rmid(StatLE)+rmid(StatTE))/2)/BS)/(chordmidS/cosd(staggermidnewS));
aoverlS=0.375;%location or max camber over chord 0.5 for circ arc mine is /2
%mmidS=0.23*(2*aoverlS)^2 + alpha3/500;
%eqnmidS=bmeamidS-((alpha3-bmiamidS*mmidS*sqrt(soverlmidS))/(1-mmidS*sqrt(soverlmidS)))==0;
%bmeamidS=vpasolve(eqnmidS,bmeamidS);
%bmeamidS=double(bmeamidS);
devmidS=(0.23*(2*aoverlS)^2 + alpha3/500)*(alpha2mid-alpha3)*soverlmidS^0.5; %Carter
bmeamidS=alpha3-devmidS;
bmiatipS=alpha2tip;%Assuming 0 incidence
%syms bmeatipS;% blade metal exit angle
soverltipS=(2*pi*((rcas(StatLE)+rcas(StatTE))/2)/BS)/(chordcasS/cosd(staggertipnewS));
aoverlS=0.375;%location or max camber over chord 0.5 for circ arc mine is /2
%mtipS=0.23*(2*aoverlS)^2 + alpha3/500;
%eqntipS=bmeatipS-((alpha3-bmiatipS*mtipS*sqrt(soverltipS))/(1-mtipS*sqrt(soverltipS)))==0;
%bmeatipS=vpasolve(eqntipS,bmeatipS);
%bmeatipS=double(bmeatipS);
devtipS=(0.23*(2*aoverlS)^2 + alpha3/500)*(alpha2tip-alpha3)*soverltipS^0.5; %Carter
bmeatipS=alpha3-devtipS;
%Uses circular arc camber to get blades angles through blade
perspan=linspace(0,100,(RotTE-RotLE)/2+1);
Rhubxoverr=(perspan/100)*(cosd(bmeahub+90)-cosd(bmiahub+90))+cosd(bmiahub+90);
Rhubangles=-1*(acosd(Rhubxoverr)-90);
Rmidxoverr=(perspan/100)*(cosd(bmeamid+90)-cosd(bmiamid+90))+cosd(bmiamid+90);
Rmidangles=-1*(acosd(Rmidxoverr)-90);
Rtipxoverr=(perspan/100)*(cosd(bmeatip+90)-cosd(bmiatip+90))+cosd(bmiatip+90);
Rtipangles=-1*(acosd(Rtipxoverr)-90);
Shubxoverr=(perspan/100)*(cosd(bmeahubS+90)-cosd(bmiahubS+90))+cosd(bmiahubS+90);
Shubangles=(acosd(Shubxoverr)-90);
Smidxoverr=(perspan/100)*(cosd(bmeamidS+90)-cosd(bmiamidS+90))+cosd(bmiamidS+90);
Smidangles=(acosd(Smidxoverr)-90);
Stipxoverr=(perspan/100)*(cosd(bmeatipS+90)-cosd(bmiatipS+90))+cosd(bmiatipS+90);
Stipangles=(acosd(Stipxoverr)-90);
%Modifies Blade Camber So Turning takes place in first 50% blade
perspan=perspan/2;
perspan(length(perspan)+1:length(perspan)+(RotTE-RotLE)/2)=linspace(50.1,100,(RotTE-RotLE)/2);
Rhubangles(length(Rhubangles)+1:length(Rhubangles)+(RotTE-RotLE)/2)=Rhubangles(length(Rhubangles));
Rmidangles(length(Rmidangles)+1:length(Rmidangles)+(RotTE-RotLE)/2)=Rmidangles(length(Rmidangles));
Rtipangles(length(Rtipangles)+1:length(Rtipangles)+(RotTE-RotLE)/2)=Rtipangles(length(Rtipangles));
Shubangles(length(Shubangles)+1:length(Shubangles)+(RotTE-RotLE)/2)=Shubangles(length(Shubangles));
Smidangles(length(Smidangles)+1:length(Smidangles)+(RotTE-RotLE)/2)=Smidangles(length(Smidangles));
Stipangles(length(Stipangles)+1:length(Stipangles)+(RotTE-RotLE)/2)=Stipangles(length(Stipangles));
end

function Fun = boverlfun(x,B2hub,B1hub)
boverl=x;
Fun(1)=B2hub-B1hub+180/pi*(atan(boverl/0.390625)+atan(boverl/0.140625));
end

function [] = WriteforStagen(xhub,rhub,xcas,rcas,perspan,Rhubangles,
Rmidangles,Rtipangles,Shubangles,Smidangles,Stipangles,xmid,rmid,
RotLE,RotTE,StatLE,StatTE,stagename)
%This function creates a formatted text file for easy copy paste into stagen.dat file
fileID=fopen(stagename,'wt');%Change path as required
npts=length(xhub);
fprintf(fileID,'%1s %3.0f %1s\n','Hubx',npts,'pts');
fprintf(fileID,'%12.6f',xhub);
fprintf(fileID,'%0s\n\n%1s %3.0f %1s\n','','Hubr',npts,'pts');
printf(fileID, '%12.6f', rhub);
printf(fileID, '%0s
\n%1s %1s %1s %1s\n', '', 'RLEx', 'RTEx', 'RLEr', 'RTEr');
printf(fileID, '%12.6f %11.6f %11.6f %11.6f', xhub(RotLE), xhub(RotTE), rhub(RotLE), rhub(RotTE));
printf(fileID, '%0s
\n%1s %1s %1s %1s\n', '', 'SLEx', 'STEx', 'SLEr', 'STEr');
printf(fileID, '%12.6f %11.6f %11.6f %11.6f', xhub(StatLE), xhub(StatTE), rhub(StatLE), rhub(StatTE));
for i = 1:length(ncas);
   printf(fileID, '%0s
\n%1s %3.0f %1s\n', '', 'Casingx', npts, 'pts');
   printf(fileID, '%12.6f', xcas);
   printf(fileID, '%0s
\n%1s %3.0f %1s\n', '', 'Casingr', npts, 'pts');
   printf(fileID, '%12.6f', rcas);
end
for i = 1:length(Rhubangles);
   printf(fileID, '%12.4f %12.4f\n', (perspan(i)/100), Rhubangles(i));
end
for i = 1:length(Rmidangles);
   printf(fileID, '%12.4f %12.4f\n', (perspan(i)/100), Rmidangles(i));
end
for i = 1:length(Rtipangles);
   printf(fileID, '%12.4f %12.4f\n', (perspan(i)/100), Rtipangles(i));
end
for i = 1:length(Shubangles);
   printf(fileID, '%12.4f %12.4f\n', (perspan(i)/100), Shubangles(i));
end
for i = 1:length(Smidangles);
   printf(fileID, '%12.4f %12.4f\n', (perspan(i)/100), Smidangles(i));
end
fprintf(fileID, '\%ls\n', 'Stator Tip Angles');
for i = 1:length(Stipangles);
    fprintf(fileID, '\%12.4f \%12.4f\n', (perspan(i)/100), Stipangles(i));
end
end

function [] = PointwiseInput(newxhub, newrhub, newxcas, newrcas, xhub, rhub,
  xcas, rcas, RotLE, RotTE, StatLE, StatTE, pointwisename)

%This converts the velocities to meridional velocities to be used
fileID=fopen(pointwisename, 'wt');%Change path as required
npts=length(newxhub);
fprintf(fileID, '\%3.0f\n', npts);
for i = 1:npts;
    fprintf(fileID, '\%12.6f \%12.6f \%12.6f\n', newxhub(i),0, newrhub(i));
end
npts=length(newxcas);
fprintf(fileID, '\%3.0f\n', npts);
for i = 1:npts;
    fprintf(fileID, '\%12.6f \%12.6f \%12.6f\n', newxcas(i),0, newrcas(i));
end

%Rot LE
fprintf(fileID, '\%3.0f\n', 2);
fprintf(fileID, '\%12.6f \%12.6f \%12.6f\n', xhub(RotLE),0, rhub(RotLE));
fprintf(fileID, '\%12.6f \%12.6f \%12.6f\n', xcas(RotLE),0, rcas(RotLE));

%Rot TE
fprintf(fileID, '\%3.0f\n', 2);
fprintf(fileID, '\%12.6f \%12.6f \%12.6f\n', xhub(RotTE),0, rhub(RotTE));
fprintf(fileID, '\%12.6f \%12.6f \%12.6f\n', xcas(RotTE),0, rcas(RotTE));

%Stat LE
fprintf(fileID, '\%3.0f\n', 2);
fprintf(fileID, '\%12.6f \%12.6f \%12.6f\n', xhub(StatLE),0, rhub(StatLE));
fprintf(fileID, '\%12.6f \%12.6f \%12.6f\n', xcas(StatLE),0, rcas(StatLE));

%Stat TE
fprintf(fileID, '\%3.0f\n', 2);
fprintf(fileID, '\%12.6f \%12.6f \%12.6f\n', xhub(StatTE),0, rhub(StatTE));
fprintf(fileID, '\%12.6f \%12.6f \%12.6f\n', xcas(StatTE),0, rcas(StatTE));
fclose(fileID);
end

function [mcor, wscale, wcordes, wtakeoff, FPRtakeoff, chokedratio,
Thrusttakeoff] = takeoffspeed(mcor, Tt0, Pt0, mdot, F, gamma, R, A19, A2,
Rhubangles, Rmidangles, Rtipangles, rhub, rmid, rcas, RotTE, cp, A13, xhub,
xicas, w, Ptoff, Ttoff, P19off, RotLE)
mcor=mdot*sqrt(Tt0/Pt0);
wscale=mcor/((gamma−1)/2)∗M2take^2);
V2take=M2take*sqrt(gamma>R*T2take);
\[ V_{13mhub} = \left( \frac{V_2 \text{take}}{\sin(\text{atan}(x_{\text{hub}}(\text{RotTE}) - x_{\text{hub}}(\text{RotTE} - 1)))} / (r_{\text{hub}}(\text{RotTE}) - r_{\text{hub}}(\text{RotTE} - 1))) \right); \]
\[ V_{13mmid} = \left( \frac{V_2 \text{take}}{\sin(\text{atan}(x_{\text{mid}}(\text{RotTE}) - x_{\text{mid}}(\text{RotTE} - 1)))} / (r_{\text{mid}}(\text{RotTE}) - r_{\text{mid}}(\text{RotTE} - 1))) \right); \]
\[ V_{13mtip} = \left( \frac{V_2 \text{take}}{\sin(\text{atan}(x_{\text{cas}}(\text{RotTE}) - x_{\text{cas}}(\text{RotTE} - 1)))} / (r_{\text{cas}}(\text{RotTE}) - r_{\text{cas}}(\text{RotTE} - 1))) \right); \]
\[ w_{\text{theta2hub}} = \frac{-1}{\pi} \cdot V_{13mhub} \cdot \tan(R_{\text{hubangles}}(31)); \]
\[ w_{\text{theta2mid}} = \frac{-1}{\pi} \cdot V_{13mmid} \cdot \tan(R_{\text{midangles}}(31)); \]
\[ w_{\text{theta2tip}} = \frac{-1}{\pi} \cdot V_{13mtip} \cdot \tan(R_{\text{tipangles}}(31)); \]
\[ T_{t13hub} = \frac{(w_{\text{takeoff}}^2 \cdot r_{\text{hub}}(\text{RotTE})^2 - w_{\text{takeoff}} \cdot r_{\text{hub}}(\text{RotTE}) \cdot w_{\text{theta2hub}})}{cp + T_{\text{toff}}}; \]
\[ \text{work13hub} = cp \cdot \left( \frac{T_{t13hub} - T_{\text{toff}}}{(w_{\text{takeoff}} \cdot r_{\text{mid}}(\text{RotLE})^2)} \right); \]
\[ T_{t13mid} = \frac{(w_{\text{takeoff}}^2 \cdot r_{\text{mid}}(\text{RotTE})^2 - w_{\text{takeoff}} \cdot r_{\text{mid}}(\text{RotTE}) \cdot w_{\text{theta2mid}})}{cp + T_{\text{toff}}}; \]
\[ \text{work13mid} = cp \cdot \left( \frac{T_{t13mid} - T_{\text{toff}}}{(w_{\text{takeoff}} \cdot r_{\text{mid}}(\text{RotLE})^2)} \right); \]
\[ T_{t13tip} = \frac{(w_{\text{takeoff}}^2 \cdot r_{\text{cas}}(\text{RotTE})^2 - w_{\text{takeoff}} \cdot r_{\text{cas}}(\text{RotTE}) \cdot w_{\text{theta2tip}})}{cp + T_{\text{toff}}}; \]
\[ \text{work13tip} = cp \cdot \left( \frac{T_{t13tip} - T_{\text{toff}}}{(w_{\text{takeoff}} \cdot r_{\text{mid}}(\text{RotLE})^2)} \right); \]
\[ P_{t13hub} = P_{\text{toff}} \cdot \left( \frac{T_{t13hub}}{T_{\text{toff}}} \right)^{\left( \frac{\gamma}{\gamma - 1} \right)}; \]
\[ P_{t13mid} = P_{\text{toff}} \cdot \left( \frac{T_{t13mid}}{T_{\text{toff}}} \right)^{\left( \frac{\gamma}{\gamma - 1} \right)}; \]
\[ P_{t13tip} = P_{\text{toff}} \cdot \left( \frac{T_{t13tip}}{T_{\text{toff}}} \right)^{\left( \frac{\gamma}{\gamma - 1} \right)}; \]
\[ p = \text{polyfit}([\text{work13hub} \\ \text{work13mid} \\ \text{work13tip}], [r_{\text{hub}}(\text{RotTE}) \\ r_{\text{mid}}(\text{RotTE}) \\ r_{\text{cas}}(\text{RotTE})], 2); \]
\[ \text{spanPt} = \text{linspace}([\text{work13hub} \ \ \text{work13tip}], 100); \]
\[ \text{spanr} = \text{polyval}(p, \text{spanPt}); \]
\[ \text{figure}(2); \]
\[ \text{plot}(	ext{spanPt}, (\text{spanr} - \min(\text{spanr})) / (\max(\text{spanr}) - \min(\text{spanr})) \cdot 100); \]
\[ \text{hold on}; \]
\[ \text{totalarea} = \pi \cdot r_{\text{cas}}(\text{RotTE})^2 - \pi \cdot r_{\text{hub}}(\text{RotTE})^2; \]
\[ \text{for} \ i = 1:length(\text{spanr}) - 1; \]
\[ \text{areaslice}(i) = \pi \cdot \text{spanr}(i+1)^2 - \pi \cdot \text{spanr}(i)^2; \]
\[ \text{wieavg}(i) = (\text{areaslice}(i) / \text{totalarea}) \cdot \text{spanPt}(i+1); \]
\[ \text{end} \]
\[ \text{FPRtakeoff} = \text{sum}(\text{wieavg}); \]
\[ \text{Pt13} = \text{FPRtakeoff} \cdot P_{\text{toff}}; \]
\[ \text{chokedratio} = \frac{\text{Pt13}}{P_{\text{off}}}; \]
\[ \text{fun} = @(x) \text{M19takefun}(x, P_{\text{off}}, P_{\text{toff}}, \gamma, R); \]
\[ x0 = [0.01]; \]
\[ \text{options} = \text{optimset}('\text{Display}', 'off'); \]
\[ M19 = \text{fsolve}(\text{fun}, x0, \text{options}); \]
\[ T_{t19} = T_{\text{toff}} \cdot \left( \left( \frac{\text{FPRtakeoff}}{P_{\text{off}}} \right)^{\left( \frac{\gamma - 1}{(1\cdot \gamma - 1)} \right)} \right); \]
\[ T_{19} = T_{t19} \cdot (1 + ((\gamma - 1)/2) \cdot M19.2) \cdot (-1); \]
\[ V_{19} = M19 \cdot \sqrt{\gamma \cdot R \cdot T_{19}}; \]
\[ \text{Thrusttakeoff} = (\text{mcortake} \cdot (V_{19})) / 1000; \]
\[ \text{end} \]
\[ \text{function} \ \text{Fun} = \text{M2takefun}(x, \text{mcortake}, A2, P_{\text{toff}}, T_{\text{off}}, \gamma, R) \]
\[ \\
\]
\[ \text{function} \ \text{Fun} = \text{M19takefun}(x, P_{\text{off}}, P_{\text{toff}}, \gamma) \]
\[ \\
\]
89
Vita Auctoris

Name: Quentin J. Minaker

Place of Birth: Kingston, Ontario, Canada

Year of Birth: 1994

Education:
- M.A.Sc. in Mechanical Engineering
  University of Windsor, 2016-2019
- B.A.Sc. in Mechanical Engineering with Aerospace Option
  University of Windsor, 2012-2016