Development of a Body Force Description for Compressor Stability Assessment

by

George Kiwada

B.S. Mechanical Engineering, University of Michigan (2005)
B.S. Aerospace Engineering, University of Michigan (2005)

Submitted to the Department of Aeronautics and Astronautics in partial fulfillment of the requirements for the degree of

Master of Science in Aeronautics and Astronautics

at the

MASSACHUSETTS INSTITUTE OF TECHNOLOGY

February 2008

© Massachusetts Institute of Technology 2008. All rights reserved.

Author .......................................................... Department of Aeronautics and Astronautics
December 4th, 2007

Certified by ...................................................... Edward Greitzer
H.N. Slater Professor in Aeronautics and Astronautics
Thesis Supervisor

Certified by ........................................................ Choon Sooi Tan
Senior Research Engineer
Thesis Supervisor

Accepted by ...................................................... David L. Darmofal
Associate Professor of Aeronautics and Astronautics
Chair, Committee on Graduate Students
Development of a Body Force Description for
Compressor Stability Assessment
by
George Kiwada
Submitted to the Department of Aeronautics and Astronautics
on December 4th, 2007, in partial fulfillment of the
requirements for the degree of
Master of Science in Aeronautics and Astronautics

Abstract
This thesis presents a methodology for a body force description of a compressor with
particular application to compressor stability calculations. The methodology is based on
extracting blade forces from an axisymmetric flow description and reinterpreting the
blade force as a body force acting throughout the fluid. A “blade force average,” which
translates a three dimensional flow into the axisymmetric description used in this
methodology, is described. The methodology is demonstrated using flow fields from
three dimensional computational fluid dynamics (CFD) to form body force distributions
over the range of flow coefficients in which converged solutions can be obtained and
using two dimensional CFD to extend the description to lower flows.

The connection between specific flow features and the body force is described for a
sample compressor, along with the effect of using averages other than the body force
average when reducing the three dimensional flow to an axisymmetric description. To
assess the body force modeling procedure, flow fields produced by using the body force
as input to a compressor flow computation are presented and compared to the flow fields
originally used to produce the body forces. While not conclusive, the comparison
provides a positive indication that the body force modeling can be useful as part of a stall
prediction procedure.

Thesis Supervisors: Edward Greitzer
                        Choon Tan
Titles: H.N. Slater Professor in Aeronautics and Astronautics
            Senior Research Engineer
Acknowledgements

I would first like to thank Mitsubishi Heavy Industries for funding my research and for flying me to Japan to see their facilities. In particular I would like to thank Sumiu Uchida, both for his bi-weekly input at our video conferences and the personalized tour in Takasago.

I am also grateful to Prof. Greitzer and Dr. Tan, my research and thesis supervisors for the opportunity to work on this project and all the work they have put into making me a better engineer, researcher, and writer. I would also like to thank Prof. Spakovsky for his work on the MHI project and the insight he has provided during our weekly meetings, and Dr. Adamczyk for helping me understand the mathematical concepts of this work.

I am indebted to all of the other members of the MHI team: Dr. Yifang Gong, Prof. Jun Li, Prof. Qiushi Li, Thomas Walker, and Georg Reichstein, for their hard work, helpful advice, support, and most importantly their friendship. Also I want to acknowledge Mikhail Krotov for his previous work on the project and for answering my many questions.

I would also like to acknowledge Professor David Miller for his initial financial and continuing intellectual support, particularly for giving me the opportunity to TA his 16.82x design class; this was an experience I will always treasure.

I would like to thank all of my friends at MIT and in the Gas Turbine Lab whom I have not already acknowledged, especially Dave Tarr for always being willing to help me relieve stress by tossing the football around, Andy Hill for his camaraderie in TAing, classes, and generally helping me get my feet on the ground here at MIT. I would also like to thank my office mates, Nick Chan and Andreas Peters.

Finally, I would like to thank my family for their love and support throughout my education, and my girlfriend Megan for her love and understanding.
Table of Contents

1 Introduction 15
  1.1 Background 16
    1.1.1 Compressor Stability 16
    1.1.2 Stall Inception 16
    1.1.3 Body Force based Stability Modeling 17
    1.1.4 MIT-MHI Compressor Stability Project 17
  1.2 Thesis Contents 18
    1.2. Contributions of this Work 19
  1.3 Organization of this Document 19

2 Body Forces 20
  2.1 Body Forces 20
  2.2 The Blade Force Average 22
  2.3 Extracting Body Forces 24
  2.4 Accounting for Quadratic Terms in the Averaging Process 26
  2.5 Example of the Averaging and the Quadratic Force 27

3 Body Force Extraction Procedure 29
  3.1 Body Force Extraction Code 29
  3.2 Sample Cases to Assess the Procedure 32
    3.2.1 Test Case 1: Solid Body Rotation 32
    3.2.2 Test Case 2: Simulated Inlet Guide Vane 33
    3.3 Test Case 3: Metal Blockage 35
    3.2.4 Test Case 4: Force Extraction from Euler Code Results 37
  3.3 Force Extraction Procedure for Different Inputs 38
  3.4 Summary 40

4 Single Stage Compressor Results 41
  4.1 Body forces from a Sample Compressor 42
4.1.1 Forces in the Inlet Guide Vane 42
4.1.2 Forces in the Rotor 44
4.1.3 Forces in the Stator 47
4.2 Quadratic Force in a Sample Compressor 50
  4.2.1 Quadratic Force in the Inlet Guide Vane 51
  4.2.2 Quadratic Force in the Rotor 52
  4.2.3 Quadratic Force in the Stator 54

5 Using Body Force Distributions in Stall Prediction 57
  5.1 Overview of Body Force Modeling Procedure 58
  5.2 Extracting the Body Forces from CFD 60
  5.3 Extending the Body Forces 60
    5.3.1 Estimating the Flow Coefficient of the Peak Body Force 62
    5.3.2 Estimating the Rate of Decrease in Body Force 63
  5.4 Distributing the Extended Force 66
  5.6 Summary 68

6 Results from Body Force Modeling Procedure 69
  6.1 Steady State Flow Fields at High Flow Coefficients 70
    6.1.1 Comparison with SLC 72
  6.2 Flow Fields using Extended Forces 73

7 Future Work 76
  7.1 Replacing FINE with T-block 76
  7.2 Improvements to the Force Extension Procedure 77
    7.2.1 Two Dimensional CFD Body Force Curves at Hub and Tip 77
    7.2.2 SLC Body Force Curves Past Stall 77
  7.3 Further Interpretation of the Body Forces 78
  7.4 Reconciling the Differences Between UNSCOMP and Input Flow 78
  7.4 General Suggestions for Future Work 79
    7.4.1 SLC Based on a Blade Force Average 79
### 7.4.2 Sensitivity of the Procedure to the Fidelity of the Force

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>8 Summary and Conclusions</td>
<td>80</td>
</tr>
<tr>
<td>A Derivation of the Flux Variables</td>
<td>83</td>
</tr>
<tr>
<td>B Body Force Distributions</td>
<td>89</td>
</tr>
<tr>
<td>References</td>
<td>99</td>
</tr>
</tbody>
</table>
## List of Figures

1-1 Modules needed for prediction of compressor instability  
2-1 Control Volume in a Stator  
2-2 Graphical Representation of Averaging the Flux  
2-3 Two Streams of Different Impulse Mixing in a Pipe  
3-1 Generic Computational Cell  
3-2 Force from Solid Body Rotation Test Case  
3-3 Comparison of Predicted vs. Extracted Force for Simulated IGV  
3-4 Axial Force from Blockage Test Case  
3-5 Radial Force from Blockage Test Case  
3-6 Enlarged Plot of Axial Force from Blockage Test Case  
3-7 Derivative of Blockage from Central Difference Method  
3-8 Axial Force Extracted Compared to Input File, Euler Code Test Case  
3-9 Tangential Force Extracted Compared to Input File, Euler Code Test Case  
3-10 Body Force Extraction Procedure with 3D CFD as Input  
3-11 Body Force Extraction Procedure with Experimental Data as Input  
3-12 Body Force Extraction Procedure with SLC as Input  
4-1 MHI Compressor Characteristic from Measurements and 3D CFD  
4-2 IGV Body Force Distribution, 87% Design Flow  
4-3 Total Tangential IGV Force at Midspan  
4-4 Rotor Force Distribution, 100% Design Flow  
4-5 Rotor Force Distribution, 80% Design Flow (near stall)  
4-6 Rotor Body Force distribution, Blade Tip 100% Design Flow
5-5 Rotor Tangential Force Midspan 64
5-6 Axial Force Rotor 96% Span 65
5-7 Axial Force Stator 4% Span 65
5-8 Rotor Force Distribution, 76% design flow 66
5-9 2D CFD Force Compared to the Extended Body Force at 64% Design Flow 67
6-1 Pressure Rise Characteristic from FINE and UNSCOMP 70
6-2 Axial Flow at Rotor Exit from UNSCOMP and FINE 71
6-3 Axial Flow at Stator Exit from UNSCOMP and FINE 71
6-4 Axial Velocity Rotor Exit from UNSCOMP, FINE, and SLC, 71
6-5 Axial Velocity Stator Exit from UNSCOMP, FINE, and SLC, 73
6-6 Stator Exit Velocity from FINE and UNSCOMP for
Different Operating Conditions 74
6-7 Pressure Rise Characteristics from FINE and UNSCOMP (extended force) 75
7-1 Pressure Rise Characteristics from Measurement, FINEturbo, UNSCOMP, and T-block 76
Nomenclature

\( r = \) radial direction
\( \theta = \) circumferential direction
\( x = \) axial direction
\( P = \) pressure
\( V = \) velocity
\( S = \) blade spacing, chord to chord
\( F_x = \) Axial force distribution in the radial and axial directions
\( F_\theta = \) Tangential force distribution in the radial and axial directions
\( F_r = \) Radial force distribution in the radial and axial directions
\( \Phi = \) Flow Coefficient, \( V_x/\)wheelspeed
\( \Psi = \) Pressure rise (total pressure to static pressure)
\( \lambda = \frac{r(\theta_2 - \theta_1)}{S} = \) Metal blockage term
\( \overline{Q} = \frac{1}{r(\theta_2 - \theta_1)} \int_{\theta_1}^{\theta_2} Qd\theta = \) Theta based average of an arbitrary flow quantity “Q”

Blade Geometry Nomenclature
**Abbreviations**

SLC = Streamline Curvature  
CFD = Computational Fluid Dynamics  
MHI = Mitsubishi Heavy Industries

**Terminology**

Axisymmetric: Uniform in the θ direction

Blade Force: The force exerted by the blade surfaces on the fluid in a control volume bound by the blades

Blade Force Average: A form of directed averaging a three dimensional flow into an axisymmetric flow that allows the blade force to be determined from axisymmetric flow

Body Force: The blade force redistributed to act uniformly on the fluid mass

Flux variable: multidimensional quantity representing the flux of mass and momentum

Two-dimensional: In this document usually refers to a x-θ plane
Chapter 1

Introduction

Aerodynamic instability in axial compressors has limited the performance of gas turbine engines since their first use. Both ground- and air-based gas turbines can encounter types of instability (rotating stall and surge) associated with stall and these instabilities can give rise to serious aerodynamic or aeromechanical consequences. Accurate prediction of instability onset has thus been a long-standing goal for compressor technologists.

A research project is underway at MIT to develop a methodology for compressor stability prediction based on recent advances in the understanding of compressor stability. Previous stability estimates have been based on a variety of combinations of empirical information and two-dimensional models of individual blade rows or stages. Methods used in industry are mainly based on steady flow calculation and correlations, while compressor stability is fundamentally unsteady. The procedure described here includes a description of unsteady behavior of the compressor, using body forces to represent the blade rows.

Previous work by Gong [1] has demonstrated the potential for capturing compressor instability onset using body force distributions to model the effect of the blade rows. That initial work, which was aimed more at showing possibilities of the method, rather than development of a predictive methodology, used a number of approximations for the body force. This thesis outlines a procedure for extracting body force distributions from computational simulations of flow in the compressor over the range in which such
simulations exist, plus a method for extending these body forces over the additional flow range needed for stability assessment.

1.1 Background

1.1.1 Compressor Stability

The pressure rise produced by a compressor running at constant speed generally increases with decreasing mass flow, up to a point, known as the stall point. Beyond this point further decrease in mass flow leads to large amplitude fluid oscillations. The oscillations are manifestations of instabilities and are known as rotating stall or surge (see Cumpsty [2] for a detailed description). The instability onset point is defined as the operating condition at which disturbances in the flow grow rather than decay.

1.1.2 Stall Inception

Two distinct paths to instability have been identified, spike stall and modal stall (Camp and Day [3]). In the former the disturbance has a short wave length, on the order of the blade chord but with a finite amplitude. In the latter the wave length is on the order of the compressor circumference but the amplitude is small. Spike stall is thus a nonlinear process while modal stall is linear in the initial stages. The time scales with which these disturbances grow also differ. Spikes grow into stall on the order of one rotor revolution while modes grow an order of magnitude more slowly (Camp and Day [3]). A description of stability needs to be able to capture the growth of both spike and modal disturbances to have a complete stall prediction capability.
1.1.3 Body Force Based Stability Modeling

A basic statement about compressor stability is that it cannot be determined from examination of the individual compressor blade rows in isolation and that the entire compressor must be examined as system of interacting blade rows. However, it does not appear to be necessary to describe the unsteady flow at a blade to blade level within each row (Xu, Hynes, and Denton[4]) and a quasi-axisymmetric disk of force (or body force) is therefore appropriate for stall prediction. Quasi-axisymmetric in this context means that the blade row is represented as a collection of axisymmetric distributions of force for different steady operating points. The force in each disk, however, can vary locally to respond to asymmetric flow.

The computational procedure developed by Gong [1] uses such a body force distribution to simulate the effect of blade rows. To determine the compressor instability point the method allows for input of local regions of low or reverse flow (i.e. specified flow perturbations) at a specified flow coefficient, and examination of whether the disturbances grow or decay. The method has been used successfully to show both modal and spike stall.

1.1.4 MIT MHI Compressor Stability Project

The MIT-MHI compressor stability project is an attempt to extend the body force stability modeling into a predictive capability. The different aspects of the overall procedure are shown in Figure 1-1.
The proposed procedure begins with the compressor geometry and ends with instability criteria. Boxes 6-7 are developments of the body force stability method. The work included in this thesis is represented by box 3, in which "CV analysis" stands for control volume analysis.

1.2 Thesis Contents

This thesis documents the procedure developed to produce an appropriate set of body force distributions for use in stall prediction. The procedure is based on determining body force distributions from computational simulation plus extending these force distributions for flow coefficients beyond which the simulation does not converge.
1.2.1 Contributions of this Work

1) A comprehensive explanation of the body forces and quadratic terms that arise from averaging three dimensional turbo machinery flows with examples of the magnitude and influence of the quadratic terms.

2) A procedure for developing body force distributions for use in an instability prediction methodology, which includes conditions from design flow to zero flow.

1.3 Organization of this Document

The chapters in this document are organized into two groups, Chapters 2-4 and chapters 5-6, to reflect the distinct nature of the above two contributions. Chapters 2-4 present a discussion of the body forces starting from defining the terminology and concepts (Chapter 2), moving to the specific procedure for evaluating the body forces (Chapter 3), and presenting quantitative examples of the body forces from a representative compressor (Chapter 4). These three chapters establish a basic methodology for describing a compressor in terms of body forces.

Chapters 5 and 6 use the methodology to create an appropriate body force description for use in compressor stability modeling. A procedure is described to create body force distributions where three dimensional converged CFD solutions do not exist (chapter 5). The adequacy of the proposed procedure in describing compressor behavior is assessed in Chapter 6.
Chapter 2

Body Forces

This chapter outlines the concepts and motivation behind the specific techniques employed to describe a compressor in terms of body forces. The chapter also defines concepts and terminology to be used later in this document.

2.1 Body Forces

The forces which act on fluid particles are either surface forces (acting on the boundary of the system) such as pressure and shear forces, or body forces, such as gravity, which act throughout the mass of the fluid (Greitzer, Tan, Graf [5]). Recent compressor modeling procedures proposed by Longly [6], Xu Hynes and Denton [4], and Gong [1] represent the blade rows of compressors through body forces. The basic idea is that instead of individual blades exerting surface forces on the flow there is a body force distributed in the blade row region\(^1\). Replacing the blades with body forces is useful because the calculations involve orders of magnitude less computations than full three dimensional unsteady blade row calculations (Xu [8]).

Procedures for simulating the compressor through body forces typically depend on functional models for body forces. Given a sufficiently detailed flow field, however, it is possible to determine the blade forces directly. The blade forces can then be recast as body forces for use in the compressor modeling procedures. We have defined the process of determining the blade forces and translating them into a body force distribution as

\(^1\) Streamline curvature calculations and actuator-disk methods are also forms of body-force description
body force extraction. Body force extraction is nothing more mysterious than smearing the blade forces across the bladed region of the compressor.

The body force extraction procedure envisioned may be called upon to produce body force distributions from a variety of possible inputs. For example the procedure might be used to extract body forces from an experimentally measured flow field, an axisymmetric stream line curvature (SLC) flow field, or a detailed three dimensional computational flow field. While common computational fluid dynamics (CFD) codes such as FINE and Fluent can output the forces on the blades directly, other computational procedures such as stream line curvature do not have this capability. One thing these codes do share is that they produce flow fields i.e. velocities and pressures. A procedure which determines the forces on the blades from the flow fields can thus be used with any of the common computational methods, as well as experimentally measured flow fields, to create body force distributions.

![Figure 2-1 Control Volume in a Stator](image)

To define the body force distribution we start with a control volume between two blades shown in Figure 2-1. There are six surfaces for this control volume, two of which are the...
blade suction and pressure sides. All the force exerted by the blades on the fluid within this control volume comes from these two blade surfaces, which act to change the impulse of the fluid in this control volume. The blade forces can therefore be determined by examining the flux of impulse into and out of the control volume.

For this examination we begin with the incompressible form of the Navier-Stokes equations, in this case cylindrical (Greitzer, Tan, and Graf [5]).

\[
\begin{align*}
\frac{1}{r} \frac{\partial V_r}{\partial r} + \frac{1}{r} \frac{\partial V_\theta}{\partial \theta} + \frac{\partial V_z}{\partial z} - V_\theta^2 &= -\frac{1}{\rho} \frac{\partial P}{\partial r} + \hat{\rho} \cdot \nabla \tau + \frac{\Omega V_\theta}{r} + \frac{\Omega^2}{r^2} + F_r, \\
\frac{1}{r} \frac{\partial V_\theta}{\partial r} + \frac{1}{r} \frac{\partial V_\theta}{\partial \theta} + \frac{\partial V_z}{\partial z} + V_z V_\theta &= -\frac{1}{r\rho} \frac{\partial P}{\partial \theta} + \hat{\theta} \cdot \nabla \tau - \frac{\Omega V_r}{r} + F_\theta, \\
\frac{1}{r} \frac{\partial V_z}{\partial r} + \frac{1}{r} \frac{\partial V_\theta}{\partial \theta} + \frac{\partial V_z}{\partial z} &= -\frac{1}{\rho} \frac{\partial P}{\partial z} + \hat{z} \cdot \nabla \tau + F_z.
\end{align*}
\]

In turbomachinery the forcing terms in Equation 2-1 (\(F_x, F_\theta,\) and \(F_r\)) come from the blade forces. The blade forces can be represented axisymmetrically by adding the forces on suction surface to the blade and the forces on the pressure surface of the blade into an axisymmetric force distribution. This axisymmetric force distribution can be applied throughout the axial region occupied by the blade row, thereby translating the blade forces into a body force distribution.

2.2 The Blade Force Average

The previous section demonstrated that the blade force can be described axisymmetrically; therefore it should be possible to average the flow and extract the body force distribution. Blade force averaging is designed for this purpose. The term blade
force average means a directed form of averaging which preserves the blade force in the same way that thrust averaging is a form of averaging that preserves thrust.

The blade force average is significantly different than previous passage averages such as those used by Smith [8] or Horlock and Marsh [9] in that it does not assume that the circumferential variations in the flow are small and it defines the axisymmetric flow with more variables (nine in the incompressible case\(^2\)). Accounting for the circumferential variation is possible because of the detailed flow field information available from three dimensional CFD, which was not accessible to earlier fluid dynamicists. The only quantity that is neglected in the averaging procedure developed here is the shear stress on the walls of the control volume not bounded by the blades.

\[
\int H dA = \bar{H} \times \text{Area}
\]

Figure 2-2 Graphical Representation of Averaging the Flux

Figure 2-2 shows two equivalent control volumes in a blade passage. In the control volume in (a) the momentum flux in and out of the control volume is determined from the computational nodes present on the control volume surfaces. In the control volume in (b) the total flux on each face is represented by a single node with the flow variables at this node constructed so the total flux through a surface in (b) is the same as the total flux

\[V_x, V_r, V^2, V^2, V_x V_r, V_x V_\theta, V_r V_\theta, P\]

\(^2\)
through the same surface in (a). Representing the momentum flux through a surface with a single point allows the flow to be expressed axisymmetrically. The new axisymmetric flow field is a blade force average of the original 3D flow field.

2.3 Extracting Body Forces

This section describes the framework for extracting the body force from a flow field. A more detailed description of the code and its assessment is contained in Chapter 3.

Conceptually the blade forces are determined by balancing the momentum flux (including the cross momentum transfer terms, such as the radial transport of axial momentum) through the control volume with the pressure forces on the surfaces of the control volume not bounded by the blades. Any imbalance is assumed to be the force exerted on the fluid by the blades. This approach assumes that the shear stress exerted on the surfaces of control volume not bounded by the blades is small.

The body force extraction is carried out on a blade force average of the flow. To implement the blade force average the flow equations are written in terms of flux variables F, G, and H (Gong [10]):

$$\frac{\partial}{\partial x} F + \frac{\partial}{\partial \theta} G + \frac{\partial}{\partial r} H = S \quad (2-2.)$$

In Equation 2-2 the flux variables F, G, H, and S are defined as:

$$F = \begin{bmatrix} rpV_x \\ rpV_x^2 + rP \\ rpV_x rV_o \\ rpV_x V_r \end{bmatrix} \lambda \quad (2-3.)$$
The different quantities in the flux variables $F, G, H$ and $S$ represent different quantities that are taken into account when balancing the force on a control volume. For example, the first component of $F$ represents the axial mass flow, the second component is the axial impulse, the third component is the axial transfer of tangential momentum, and the fourth component represents the axial transfer of radial momentum. To create a blade force average of the flow the $F, G, H$ and $S$ are circumferentially averaged ($\theta$ averaged).

The blade forces are found by balancing flux variables on four surfaces of the axisymmetric control volume. Since the flux variables are circumferentially averaged the $\theta$-derivative of $G$ in Equation 2-2 is by definition zero. The fluxes of $F$ and $H$ are therefore compared with the source term $S$ to determine the blade forces within the control volume. These blade forces are the body forces used in the stability model.
2.4 Accounting for Quadratic Terms in the Averaging Process

To use an axisymmetric representation of a non-axisymmetric flow it is necessary to account for quadratic terms which arise in the averaging processes. For example consider an arbitrary flow quantity $Q$. The average of the square is not the same as the square of the average.

$$
\overline{Q^2} \neq \overline{Q}^2 \quad (2-7.)
$$

If we express the flow quantity as an average plus a deviation

$$
Q(\theta) = \overline{Q} + Q'(\theta), \quad (2-8.)
$$

the inequality in Equation 2-7 is resolved but quadratic terms appear

$$
\overline{Q^2} = \overline{Q}^2 + 2\overline{Q}Q' + Q'^2. \quad (2-9.)
$$

Adamczyk’s averaging procedure [11] tracked each of these terms explicitly, however for body force extraction it is sufficient to account for them in bulk. The quadratic terms can be bookkept as a fictitious force, $F_{\text{quadratic}}$, as in Equation 2-10.

$$
\frac{\partial}{\partial r} \lambda r \overline{V_r^2} + \frac{\partial}{\partial x} \lambda r w \overline{V_x V_r} - \lambda \overline{V_x^2} + \frac{r}{\rho} \frac{\partial}{\partial r} \lambda \overline{P} = (F_{\text{blade}})_r + (F_{\text{quadratic}})_r \quad (2-10.)
$$

$$
\frac{\partial}{\partial r} \lambda r \overline{V_x^2} + \frac{\partial}{\partial x} \lambda r \overline{V_x V_\theta} + \lambda \overline{V_x V_\theta} = (F_{\text{blade}})_\theta + (F_{\text{quadratic}})_\theta \quad (2-10.)
$$

$$
\frac{\partial}{\partial r} \lambda r \overline{V_\theta^2} + \frac{\partial}{\partial x} \lambda r \overline{V_\theta V_r} + \frac{r}{\rho} \frac{\partial}{\partial x} \lambda \overline{P} = (F_{\text{blade}})_x + (F_{\text{quadratic}})_x
$$
Note that if instead of representing the flow in terms of products of averages \( \langle V_x V_r \rangle \) the axisymmetric flow were described in terms of averaged products \( \langle V_x V_r \rangle \) the quadratic terms do not appear explicitly in the Navier Stokes equations.

\[
\begin{align*}
\frac{\partial}{\partial r} \lambda r V_r^2 + \frac{\partial}{\partial x} \lambda r V_x V_r - \lambda V_\theta^2 + r \frac{\partial}{\partial r} \lambda \bar{P} &= F_r \\
\frac{\partial}{\partial r} \lambda r V_r V_\theta + \frac{\partial}{\partial x} \lambda r V_x V_\theta + \lambda V_\theta V_\theta &= F_\theta
\end{align*}
\]

The blade force average represents the flow in terms of averaged flux variables. This is the equivalent to using the averages of products as in Equation 2-11, and therefore the blade force average does not need quadratic force terms.

When using any other average other than the blade force average the quadratic force must be accounted for. Not doing so will lead to an error in the extracted blade force. One way of accounting for the quadratic force is to treat it as a source term, similar to the centrifugal force term, which must be subtracted from the left hand side of the equation when solving for the force on the blades. The content of the quadratic terms depends on the averaging procedure used and there is no generic expression for quadratic force.

### 2.5 Example of the Averaging and the Quadratic Force

A simple example can be given to illustrate the effect of the quadratic terms. Suppose two streams with different total pressures mix at constant area. If a control volume is drawn enclosing the mixing region, as in Figure 2-3, there is no force applied by the duct.
walls on the fluid in the control volume. For constant density, the area averaged velocity is the same at inlet and exit, but the (uniform) static pressure at inlet is different from that at exit. If the axial momentum at the inlet and exit is determined using the static pressure and the area averaged velocity, there will be a change in fluid impulse from inlet to exit. This change in impulse leads to an incorrect computation of the force exerted by the duct in the axial direction. This fictitious force is the effect of the quadratic terms.

\[ \rho V_x^2 + P = C + d \]

\[ \rho V_x^2 + P = C - \frac{d}{3} \]

\[ \rho V_x^2 + P = C \]

**Figure 2-3 Two streams of different impulse mixing in a pipe**

Figure 2-3 is an example of the statement: "No uniform flow exists, in general, which simultaneously matches all the significant stream fluxes, aerothermodynamic and geometric parameters of a non-uniform flow." (Pianko and Wazelt [12]). In other words there is no universal averaging procedure for turbo machinery; every type of averaging preserves some quantities while changing others (Cumpsty and Horlock [13]). Blade force averaging is a type of directed averaging that preserves the force the blades exert on the fluid within a control volume. In the example in Figure 2-3 if the flow at the inlet and exit had been averaged so that the momentum fluxes from the nonuniform flow were preserved there would be no fictitious force. Averaging to preserve the momentum flux is the idea behind the body force averaging discussed in this chapter.
Chapter 3

Body Force Extraction Procedure

This chapter describes the procedure for extracting the body forces from a computationally or experimentally specified flow field. This flow field can include detailed information in the blade passages, or simply specify conditions at the inlet and exit of the blade rows, without affecting the procedure for extracting the body forces. The amount of spatial information provided by the flow field, however, will affect the fidelity of the body forces produced. Four test cases are presented as a means of assessing the methodology.

3.1 Body Force Extraction Code

The body force extraction code takes as inputs the \( \theta \)-average of the flux variables defined in Equations 2-6 in Chapter 2. The body force extraction is performed on an axisymmetric flow, so the equations can be simplified to the following form.

\[
\frac{\partial}{\partial x} F + \frac{\partial}{\partial r} H = S \quad (3-1.)
\]

\[ F = \begin{bmatrix} rpV_x \\ rpV_x^2 + rP \\ rpV_x rV_\theta \\ rpV_r V_r \end{bmatrix} \lambda \quad (3-2.) \]

\[ H = \begin{bmatrix} rpV_r \\ rpV_x V_r \\ rpV_r rV_\theta \\ rpV_r^2 + rP \end{bmatrix} \lambda \quad (3-3.) \]
The blade force averaged flow is already expressed in terms of the flux variables, but for any other form of averaging $F$, $H$ and $S$ must be calculated from the axisymmetric flow variables available.

The flow field is divided into computational cells. Depending on the input data the flow is defined on the cell nodes or the cell centers. Because the flow is axisymmetric each cell, Figure 3-1, has only four surfaces: two nominally axial faces and two nominally radial faces. Values of the state variables at the cell walls are determined by averaging either adjacent nodes or adjacent cell centers, depending on which is specified. Since the procedure is a control volume analysis defining the flow on the cell nodes is more rigorous because nodes are on the surfaces of the control volume.
The derivatives of the flux variable $F$ are determined by calculating the flux of $F$ through the area projected normal to the axial direction of each surface and dividing the result by the cell volume.

$$\frac{\partial F}{\partial x} = \frac{F_3 A_{3\text{axial}} + F_4 A_{4\text{axial}} - \left(F_1 A_{1\text{axial}} + F_2 A_{2\text{axial}}\right)}{\text{Volume}_{\text{cell}}}
$$

(3-5.)

The derivative of $G$ is determined similarly using the radial normal areas of each surface.

$$\frac{\partial H}{\partial r} = \frac{H_3 A_{3\text{radial}} + H_4 A_{4\text{radial}} - \left(H_1 A_{1\text{radial}} + H_2 A_{2\text{radial}}\right)}{\text{Volume}_{\text{cell}}}
$$

(3-6.)

The sum of the derivatives of $F$ and $G$ in the cell is equal to the source term $S$.

$$\frac{\partial}{\partial x} F + \frac{\partial}{\partial r} H = S = \begin{bmatrix} 0 \\ \lambda r \rho F_x + r P \frac{\partial \lambda}{\partial x} \\ \lambda r \rho F_\theta \\ \lambda \rho V_\theta^2 + \lambda P + \lambda r \rho F_r + r P \frac{\partial \lambda}{\partial r} \end{bmatrix}
$$

(3-7.)

The force acting on the cell in each direction is determined from the expressions that define $S$.

$$\begin{bmatrix} F_x \\ F_\theta \\ F_r \end{bmatrix} = \begin{bmatrix} S_x - r P \frac{\partial \lambda}{\partial x} \\ \lambda r \rho \\ S_\theta \\ \lambda r \rho r \\ S_r - (\lambda \rho V_\theta^2 + \lambda P + r P \frac{\partial \lambda}{\partial r}) \end{bmatrix}
$$

(3-8.)
3.2 Sample Cases to Assess the Procedure

Four test cases were used to check the consistency of the body force extraction codes.

The first is solid body rotation in an annular duct, the second adds a region of flow blockage to purely axial flow, and the third test case is a theoretical representation of an inlet guide vane. A fourth test extracts the forces from an axisymmetric Euler flow simulation of a compressor. The flow field in all cases is axisymmetric so there is no need for averaging in the $\Theta$-direction. The body forces for each case can be determined analytically providing a metric to compare against.

3.2.1 Test Case 1: Solid Body Rotation

The first test case is a basic check of the minimum level of error to be expected in the radial and tangential forces. It is flow in a straight annular duct with body rotation.

There is no axial or radial velocity. There should be no force in any direction.

$$
\begin{align*}
V_x &= 0 \\
V_\theta &= \Omega r \\
V_r &= 0
\end{align*}
$$

(3-9.)

$$
\frac{\partial P}{\partial r} = \frac{\rho V_\theta^2}{r}
$$

$$
P = \frac{1}{2} \rho \Omega^2 r^2
$$

The test case has zero axial velocity and no axial variation in any other flow variables and it does not test the axial force extraction part of the code. The results are shown as force per unit mass, with the forces normalized by the maximum tangential velocity squared divided by duct length.
The variation in force in Figure 3-2 is of the order $10^{-15}$ which is the minimum numerical intrinsic to the force extraction code.

### 3.2.2 Test Case 2: Simulated Inlet Guide Vane

Test case 2 simulates the effect of an inlet guide vane. The flow is purely axial at the inlet. As the flow moves over a set region of the grid the circulation is increased to create a free vortex. The strength of the circulation increases linearly with axial position until a location after which the circulation is constant.

\[
\begin{align*}
V_x &= \text{Const} \\
V_r &= 0 \\
V_\theta &= 0 & \text{Before the IGV} \\
V_\theta &= \frac{\Gamma(x-x_0)}{2\pi r} & \text{In the IGV} \\
V_\theta &= \frac{\Gamma*\text{Length}_{IGV}}{2\pi r} & \text{After the IGV}
\end{align*}
\]
The static pressure is specified so there is zero radial force:

\[ \frac{\partial p}{\partial r} = \frac{\rho V^2}{r} \]

\[ P = \text{Const} - \frac{\rho \Gamma^2 (x - x_0)}{8\pi^2 r^2} \quad \text{In the IGV} \tag{3-11.} \]

\[ P = \text{Const} - \frac{\rho \Gamma^2 \text{Length}_{IGV}}{8\pi^2 r^2} \quad \text{After the IGV} \]

There should be zero force applied to the fluid except for the region of increasing circulation, the simulated IGV. In the simulated IGV the axial and tangential forces are expressed as:

\[ F_\theta = \frac{V_x \Gamma}{2\pi r} \]

\[ F_x = -\frac{\Gamma^2 (x - x_0)}{4\pi^2 r^2} \quad \text{(3-12.)} \]

Figure 3-3 show plots of the expected force compared with the actual force extracted. The only areas where there are differences are where the force has an infinite slope. The code uses a central difference method to determine derivatives; and cannot replicate an infinite slope.

Figure 3-3 Comparison of Predicted vs. Extracted Force for the Simulated IGV
3.2.3 Test Case 3: Metal Blockage

Case 3 examines the error in the axial force as well as the ability to handle blockage and gradients in the flow variables. The flow is purely axial but has contraction and expansion of the duct due to the presence of blades, i.e. metal blockage as in a real compressor. There should be no axial force for an inviscid flow, and the radial force should equal \( \frac{1}{\rho} \frac{\partial p}{\partial r} \). The results are shown in Figure 3-4 and Figure 3-5. The force is plotted as force per unit mass, normalized by the inlet velocity squared divided by the duct length.

![Figure 3-4 Axial Force from Blockage Test Case](image)

![Figure 3-5 Radial Force from Blockage Test Case](image)
While the radial force extracted shows excellent agreement with predictions the axial force contains errors at the leading and trailing edges of the blades, shown in more detail in Figure 3-6.

The errors in the axial force come from the central difference method the code uses to calculate the derivatives of the blockage, similar to the errors discussed in section 3.2.2. The leading and trailing edges of the blades are rounded, so the blockage has a near infinite slope at the transition into and out of the blade regions. The central difference method cannot capture the slopes at the leading or trailing edges, as shown in Figure 3-7, and therefore incorrectly accounts for the effect of blockage in these regions.
3.2.4 Test Case 4: Force Extraction from Euler Code Results

Case 4 is designed to assess all of the features of the force extraction code, except metal blockage, in a realistic flow. An Euler code (developed by Gong [1]) was used to simulate the flow in a sample compressor. The Euler code did not account for metal blockage. The blade rows were represented by known input body forces, providing a metric to compare the extracted body forces against. Figure 3-8 and Figure 3-9 show the results of this comparison.

The forces produced by the force extraction procedure match the input forces used by the Euler code exactly (within the numerical accuracy of the code).
3.3 Force Extraction Procedure for Different Inputs

As mentioned in Chapter 2 the body force extraction procedure is intended to operate on a variety of different inputs. Figure 3-10 through Figure 3-12 are flow charts which diagram the specifics of the procedure when using three dimensional CFD, experimental data, or SLC as input.

![Figure 3-10 Body Force Extraction Procedure with 3D CFD as Input](image)

Figure 3-10 represents a basic methodology for creating body force distributions from three dimensional flow fields. In step 1 a three dimensional CFD code is used to produce flow fields from the input geometry and operating conditions. In step 2 the flow variables are averaged using the blade force average developed in Chapter 2. In step 3 the forces are extracted using the procedure described in section 3.1.

Figure 3-11 and Figure 3-12 show the way the body force extraction procedure could be applied to experimental data or SLC flow fields inputs. It has been assumed that body
force averaged inputs are not available so that the quadratic force (section 2.4) must be modeled.
3.4 Summary

A computational procedure has been developed to extract body forces from arbitrary flow fields. The body force extraction code operates on an axisymmetric flow and determines the force in a computational cell from the net flux of the flux variables (F, H, and S). Chapters 2 and 3 form a methodology for extracting body forces from flow fields. This methodology will be applied to a compressor in Chapter 4.
Chapter 4

Single Stage Compressor Results

This chapter presents distributions of body forces and quadratic forces derived from three-dimensional CFD simulations of a single stage compressor tested at MHI Takasago R&D center. The goal of this chapter is to illustrate the description of a compressor in terms of body forces and to demonstrate the role of averaging in creating such a body force description.

The example compressor is a low speed ($M_{tip}\approx 0.6$) single stage machine comprised of an inlet guide vane blade row, a rotor, a stator, and an outlet guide vane. Measurements show that the machine exhibits hub separation over a flow range from near design to stall but that instability inception is characterized by spikes at the rotor tip. The blades are radial so the radial body force is small in comparison to the axial or tangential body force. Therefore we will not discuss the radial body force.

The presentation used is in terms of vectors that show the distribution of body force. The x-component of the vector represents the axial force and the y-component represents the tangential force. The axial location is normalized by the largest value of axial chord for the blade shown. Plotting the force vectors is a convenient way of displaying the force in the axial and $\theta$ directions simultaneously. Appendix B contains a collection of these plots at multiple operating points for each blade row.
4.1 Body Forces from a Sample Compressor

This section presents the body forces from the sample compressor and relates them to local flow features. Figure 4-1 shows the pressure rise characteristic of the compressor in normalized coordinates. Both the flow coefficient ($\Phi$), inlet velocity over wheel speed, and the total to static pressure rise ($\Psi$) are normalized by their design values. Stall occurs at approximately 75% of the design flow coefficient.

![Figure 4-1 Compressor Characteristic from Measurements and 3D CFD](image)

4.1.1 Forces in the Inlet Guide Vane

The distribution of IGV forces is similar for all the operating points because the flow upstream of the IGV is purely axial and there is no change in incidence angle with flow coefficient. Figure 4-2 shows the distribution of body force in the IGV at a representative flow coefficient, 87% of design.
The distribution of force in the IGV is similar to the chord wise distribution of loading on an external wing, with a peak loading occurring near 25% chord. The magnitude of the forces, which decrease as \( \Phi^2 \), can be seen in Figure 4-3 which gives the total (integrated) tangential force as a function of flow coefficient, at the mean radius of the IGV.
4.1.2 Forces in the Rotor

The rotor force distributions are given in Figure 4-4 and Figure 4-5. There is a concentration of force near the leading edge of the blade. For high flow coefficients the force distribution is similar to the IGV distribution, with loading concentrated at 25% chord. As flow coefficient decreases, however, the peak in loading moves closer to the leading edge. The axial force in the rotor is positive, indicating increasing static pressure through the rotor.

Figure 4-4 Rotor Force Distribution, 100% Design Flow

Figure 4-5 Rotor Force Distribution, 80% Design Flow (near stall)
The force distributions at the rotor tip (Figure 4-6 through Figure 4-8) are an indication of the onset of rotor tip stall. A small region of reverse loading, characterized by force vectors that turn the flow in the opposite direction of the rest of the blade, exists at the trailing edge at high flow coefficient. Some vectors in the reverse loading region also show negative axial force, associated with a local decrease in static pressure. This reverse in loading combined with pressure loss is characteristic of a local region of separation. As flow coefficient decreases the region of reverse loading increases in size indicating that the tip separation is growing.
The tip separation can be seen in contour plots of \( \theta \)-averaged axial velocity, Figure 4-9 and Figure 4-10. There is a region of low velocity flow in the rotor tip area (95-100\% span), which increases in size as flow coefficient decreases.

Figure 4-9 Rotor Flow Coefficient, 87\% of Design Flow

Figure 4-10 Rotor Flow Coefficient, 80\% of Design Flow (near stall)
4.1.3 Forces in the Stator

Figure 4-11 and Figure 4-12 show the distribution of body force in the stator. The hub station is at 2\% span and the tip station is at 98\% span.

Figure 4-11 Stator Force Distribution 100\% Design Flow

Figure 4-12 Stator Force Distribution 80\% Design Flow (near stall)

The stator hub force distributions show reversed loading associated with hub separation even at design flow. As seen in the contour plots of Figure 4-13 and Figure 4-14, the separated flow region grows radially with decreasing flow coefficient.
Figure 4-13 Stator Flow Coefficient, 100% Design Flow

Figure 4-14 Stator Flow Coefficient, 80% Design Flow (near stall)

Figure 4-15 through Figure 4-18 show a trend in the body force resulting from the radial growth of the separation. At high flow coefficients the force distributions near the hub (5-25% span) have distributions similar to those in the IGV. However, with decreasing flow the separation reaches further up the span and the reverse loading is seen at progressively larger radii.
The hub separation reaches the 10% span station at approximately 94% of design flow, as indicated by the reverse loading seen in Figure 4-17. The reverse loading is closer to the leading edge in Figure 4-18 indicating that the separated flow has moved closer to the leading edge at 10% span. Figure 4-15 though Figure 4-18 show that as hub separation grows radially and axially the body force distribution changes to include a region of reverse loading.
4.2 Quadratic Force in a Sample Compressor

Future applications of the body force extraction procedure may not have blade force averaged flow descriptions available. The implications of using an average other than the blade force average must therefore be examined. This section quantifies the quadratic forces obtained from an area averaging procedure and links them to flow features. The quadratic terms are directly related to the tangential non-uniformity of the flow. The quadratic force is the effect the quadratic terms have on a momentum based force extraction. They are therefore the relevant measure of the influence of the averaging method on the body forces produced by the proposed modeling procedure.

The aim of this example is to quantify the magnitude of the quadratic terms in a compressor, to link the presence of the quadratic terms to local flow features, and to demonstrate what regions of the compressor are sensitive to the type of averaging selected. As stated by Cumpsty and Horlock [13] “quantitatively the various averages differ very little for flows likely to occur in gas turbines,” but examples here show there are regimes (especially in separated flow) where the type of averaging is important.

The quadratic forces are determined by subtracting the forces extracted using a blade force average from the forces extracted using an area average of the same flow. The difference between these two sets of forces is the quadratic force.
4.2.1 Quadratic Force in the Inlet Guide Vane

The flow in the IGV is nearly uniform in the tangential direction, therefore the quadratic forces are ten times smaller than the blade forces. The results indicate that the forces extracted from the IGV are insensitive to which of the two averaging procedure is used, in line with the findings of Cumpsty and Horlock. The other blade rows in the machine will show larger quadratic forces.

![Body Force vs Quadratic Force in the IGV](image)

*Figure 4-19 Quadratic Force Compared to Body Force in the IGV, 80% Design Flow (near stall)*
4.2.2 Quadratic Force in the Rotor

At flow coefficients near design the rotor has small quadratic forces indicating a smooth attached flow throughout the blade row, Figure 4-20. As flow coefficient decreases quadratic forces become larger and more significant especially in the tip region, Figure 4-21. As the angle of incidence increases the blade loading increases and the boundary layer on the blade also increases in thickness. The boundary layer is a \( \theta \) non-uniformity and will therefore lead to a quadratic force. As the boundary layer becomes thicker the quadratic forces become larger.

![Figure 4-20 Quadratic Force Compared to Body Force in the Rotor, 100% Design Flow](image1)

![Figure 4-21 Quadratic Force Compared to Body Force in the Rotor, 80% Design Flow (near stall)](image2)
The body force distributions in section 4.1 indicated tip separation. The large quadratic forces in the rotor tip in Figure 4-21 also support this and at near stall conditions the quadratic forces in the rotor tip are comparable to the actual forces. A body force analysis that does not use a body force average, i.e. that ignores the non-uniformity, could lead to errors in the force of up to 100% in the rotor tip region at low flow coefficients.
4.2.3 Quadratic Force in the Stator

The quadratic forces in the stator are small everywhere except the hub region. The θ non-uniformities in the hub correspond to differences between area averaged flow and body force averaged flow, leading to quadratic forces.

Figure 4-22 Quadratic Force Compared to Body Force in the Stator 100% Design Flow

Figure 4-23 Quadratic Force Compared to Body Force in the Stator, 80% Design Flow (near stall)

Figure 4-22 and Figure 4-23 show decreasing quadratic force in the hub region as flow coefficient also decreases. Normally increasing separation with decreasing flow is
associated with an increase in quadratic forces. In this case this decrease indicates extreme hub separation as the flow decreases. As the hub separation increases the separated flow grows in the $\theta$ direction. Figure 4-24 shows the axial velocity at 40% chord at near design flow, the blank area of the plot represents the presence of the blade. The dark blue region of flow is the separation; at this flow coefficient hub separation covers about 50% of the $\theta$ direction in the hub. Figure 4-25 shows the axial velocity at the same location as Figure 4-24 but at a flow coefficient near stall. The region of flow separation at the hub is larger in both the radial and tangential directions. The hub flow is now more uniform than near the design flow and the quadratic forces in the hub region have actually decreased compared to the design value.

Figure 4-24 Axial Velocity in Stator at 40% Chord, 100% Design Flow
In summary, quadratic forces indicate the effect of averaging on the calculated body force. It is seen that there are regions where the averaging procedure leads to large differences. In particular, in regions with flow separation, in this case the stator hub and rotor tip, the calculated body forces can vary by 100%. Experimental measurements indicate this compressor is tip critical, and it is expected that accurate body force distributions in the rotor tip region are important for stall prediction. If area averaged flow quantities were used instead of blade force averaging the rotor tip force distributions could have errors of 100% or more.

Chapters 2 through 4 have established the basic body force methodology. Chapter 5 will build on this to create a complete set of body forces for a single stage compressor.
Chapter 5

Using Body Force Distributions in Stall Prediction

The stall prediction procedure being developed uses body forces to represent the effect of the blade rows on the flow in the compressor. The local regions of reverse flow associated with spikes imply that body forces at certain locations must be described from design conditions all the way to negative flow. However, current CFD procedures may not be able to provide a converged solution at flows below stall (or even near stall) condition. This chapter thus describes a procedure for obtaining body force distributions in flow regimes where three-dimensional CFD is not available.

For a stall simulation method to be truly predictive the body forces must come from computational flow fields rather than, for example, experimental data or data matched streamline curvature calculations. No one code currently exists which can provide detailed flow fields over the desired range in a short enough time to be useful in a design cycle. A combination of computations and modeling is thus used to provide a complete set of body forces.

In this chapter two dimensional CFD results are used to guide the body force estimation to flow coefficients at and beyond stall. To demonstrate the procedure here, two dimensional CFD curves from the mean radius are used as a basis for body force estimation at all radii in the machine. It is planned that two-dimensional CFD for hub and tip sections will be used as soon as they are available.
5.1 Overview of Body Force Modeling Procedure

Figure 5-1 outlines the proposed procedure for obtaining a set of body force distributions over the desired range of operating points from design flow to well below the stall point. In boxes 1-3, the procedure uses the methodology developed in Chapter 3. However, additional steps are now included to extend the body force distributions to lower flow coefficients. Boxes 4 and 5 show the concepts extend the integrated body forces (overall body forces for each blade section) to lower flow coefficients using two dimensional CFD as a guide. Box 6 illustrates the procedure for specifying the axial distribution of the body force in each section using the distributions from converged three dimensional CFD solutions.
Procedure for obtaining a complete set of body force distributions

1) Start with blade geometry
   3D CFD

2) 3D Flow Fields
   Body Force Extraction

3) 3D Body Force

4) Integrated Body Force Curves
   Integrate in x
   Extrapolate integrated Force curve

5) Extended Body Force Curves
   Scale Body Force Distributions to match integrated value

6) Body Force distributed locally at flow coefficients at and beyond stall

Dashed boxes represent future steps that will augment the current procedure

Figure 5-1 Body Force Extension Procedure
5.2 Extracting the Body Forces from CFD

The body force extraction procedure for three dimensional CFD has already been described in detail in Chapters 2 through 4, and this section will give only a summary of the method.

A blade force average flow field is first defined to determine the force exerted by the compressor blades. The resulting body forces are an axisymmetric representation of the blade forces. Using the blade force average removes the so called “quadratic terms” that arise from averaging. The resulting body forces are distributed as functions of axial and radial coordinates.

The remaining challenge is to capturing the flow at or beyond stall point. None of the codes tested have provided converged solutions up to the stall point. Presumably an unsteady full wheel calculation for the whole machine could do this but the time needed would be prohibitive.

5.3 Extending the Body Forces

This section describes the first step in developing body forces at flow coefficients at which there are no converged three-dimensional CFD results. The procedure is represented by boxes 4 and 5 of Figure 5-1. The force extension begins conceptually at the level of the integrated (net) effects of the blades, and then includes a model for the axial description of the body forces. The net effect of a blade row, ie the overall static
pressure change and flow turning, is reflected in the integrated body forces and the first step of the extension process estimates these integrated forces.

The trend in body force as a function of flow from the three dimensional CFD results is shown in the integrated force acting on the flow for each blade radius, Figure 5-2 shows this for the axial force at midspan in the rotor. The three-dimensional CFD used to date has only provided converged solutions to ~80% of the design flow\(^1\). Two dimensional CFD results can be used to produce integrated body forces at flow coefficients below this to extend the results in Figure 5-2.

![Figure 5-2 Integrated Axial Force, Rotor Midspan, 3D CFD](image)

The net force applied by the blade row decreases as the compressor operating point moves below stall. The flow coefficient and the rate at which the magnitude of the body force begin to decrease are thus important parameters. In the procedure described the

\(^1\) We have had encouraging results from the TBLOCK code, used in a time unsteady mode, but these results have not yet been reduced for body force extraction
flow coefficient at which the forces begin to decrease (the “peak”) is found before determining the rate of decrease.

5.3.1 Estimating the Flow Coefficient of the Peak Body Force

Two dimensional CFD curves are used to determine the location of the peak in body force. The rotor and stator show a peak in both axial and tangential force at roughly although not exactly the same flow coefficient as the peak in compressor pressure rise. For this compressor the stator peak is consistently at a slightly higher (2.5% $\Phi_{\text{des}}$) flow coefficient. Therefore the peaks in the extended body force distributions should differ between the rotor and stator. This is consistent with the trends indicated by the three dimensional CFD which shows a negative slope in the rotor but which has begun to level off in the stator, Figure 5-3.

![Figure 5-3 Integrated Axial Force, Stator Midspan 3D CFD](image)

The two-dimensional results do not reflect three-dimensional effects, such as streamline contraction, and the flow coefficient associated with the peak is not necessarily the same
as for the three dimensional flow, we thus set the flow coefficients corresponding to peak force in the two dimensional results to correspond to the peak indicated by the three dimensional CFD results. In other words the flow coefficients of the three-dimensional peaks are taken as the point for which the body forces should decrease and the two-dimensional force curves are "pinned" to these points.

5.3.2 Estimating the Rate of Decrease in Body Force

It is also necessary to determine the rate at which the body forces decrease. Again the two dimensional CFD results are used to determine this rate. The integrated forces from the two dimensional CFD are transposed so that the peak in force corresponds to the peak chosen by the procedure just described in section 5.3.1. The magnitude of the integrated two dimensional forces is also increased or decreased to match the last converged three dimension CFD body force. The transposed curves thus now represent the integrated body force for flow coefficients beyond those available from three dimensional CFD. Figure 5-4 and Figure 5-5 are examples of these extended force curves for the rotor.
In some regions of the machine the local flow coefficients are already quite low, in some cases near zero. Therefore less extension of the forces from the three dimensional results is needed in these regions, we thus only consider the transposed two dimensional curve at lower flow coefficients than the lowest three dimensional CFD results.
For low flow regions there are no large radial velocities, such as the rotor tip, Figure 5-6, the extended force curves transition smoothly from the three dimensional results.

However in the stator hub, where there are large radial streamline shifts, the extended forces have a different slope than indicated by the three dimensional CFD forces\(^2\), Figure 5-7. It is possible that using streamline curvature for the extended body forces could alleviate some of this problem since the changes in streamline height would be captured.

\[ \text{Figure 5-6 Axial Force Rotor 96\% Span} \]

\[ \text{Figure 5-7 Axial Force Stator 4\% Span} \]

\(^2\) The behavior of the integrated force as a function of \( \Phi \) in the stator hub from the three-dimensional CFD is not yet fully understood and needs further examination.
5.4 Distributing the Extended Force

This section describes the procedure for distributing the extended integrated force (section 5.3) the chordwise direction, represented by box 6 in Figure 5-1.

The stall prediction procedure requires the axial and radial distribution of body force over a range of flows. The extended force curves carry information about the radial function of the force, but they do not contain information on the axial distribution. Because there is little information on the axial distribution of force at stalled flow conditions, we use the calculated distributions from two dimensional and three dimensional CFD.

Three-dimensional CFD results normalized by the integrated force at the corresponding flow coefficient, are used to supply the distributions. Multiplying this normalized distribution by the value from the extended integrated force curves creates a distribution of force. Figure 5-8 is an example of one of the distributions for the extended force.

![Figure 5-8 Rotor Force Distribution, 76% Design Flow](image-url)
Two dimensional CFD based force distributions exist for the extended flow coefficients; however these distributions are currently available for the mean radius only. However these distributions can be compared with the scaled three dimensional distributions. Figure 5-9 shows both the two dimensional CFD force distribution and the extended force distribution. The overall force from both of these distributions is similar, and the shapes of the distributions are roughly similar in that both show a concentration of loading near the leading edge. It is planned that when the distribution from hub and tip are available these will be interrogated to see if they should be used.

![2D CFD Force Compared to the Extended Body Force at 64% Design Flow](image)

The stall prediction procedure requires the local flow coefficient at every axial location where force is specified. The force curves have been extended based on the overall force and the blade inlet local flow coefficient because the chordwise local flow coefficient is not known. The present method of choosing the local flow coefficient is use the flow coefficient at the blade inlet for every axial location at that same radius. This assumption needs to be examined.
5.6 Summary

A method for extrapolating body force distributions to lower flow coefficients than are currently available from three dimensional CFD has been developed. Two dimensional CFD results are used to describe the body forces, with the flow coefficient at which the body forces begin to decrease chosen to match the peak implied by the three dimensional CFD results.

The resulting body force description spans the required range of operating conditions and is distributed axially and radially for each blade row. The force distributions from the extension procedure are generally similar to distributions derived from 2D CFD simulations at stalled flow coefficients. The true test of the modeling procedure, however, is using the forces in the stall estimation procedure, preliminary results of which are shown in the next chapter.
Chapter 6

Results from Body Force Modeling Procedure

Chapters 2 through 5 have described a procedure for developing a set of body forces for use as part of a stall prediction methodology. The ultimate test of this procedure is the stall prediction, but one intermediate assessment is the comparison of outputs from a flow computation based on these forces with the flow field from which the forces were extracted. Such a consistency check is discussed in this chapter.

A flow computation using body forces, UNSCOMP, has been described by Gong [1]. UNSCOMP was used to answer the following questions: first, are the body forces correctly captured and described for the flow computation, and second, is the set of body force distributions self-consistent up to stall? If these questions are answered positively the proposed modeling procedure fulfills the objective of this thesis.

The body forces used for this chapter are derived from a three dimensional CFD program (FINE turbo) and a two dimensional computation (FLUENT), as described in chapters 3 to 5. The flow fields and pressure rise characteristics from these two CFD codes will thus be used as the basis for comparing the accuracy of the UNSCOMP results. An additional comparison will be made with stream line curvature (SLC) results, to give measure of how the results from the body force modeling procedure compare to the results from an independent computation.
6.1 Steady State Flow Fields at High Flow Coefficients

For the initial tests of the body force extraction procedure the UNSCOMP code was run with a single set of body force distributions derived from FINE. More specifically the body forces for one flow coefficient were input into UNSCOMP and the flow computed for these forces. All the flow coefficients examined were within the range of converged results from FINE turbo, so none of these distributions contain any extended body forces.

Figure 6-1 shows the overall pressure rise produced by UNSCOMP and FINE. The two are similar with the maximum error being 10% in pressure rise at 83% design flow. This is an indicator that the body forces produced through the proposed procedure are captured and correctly described.

![Figure 6-1 Pressure Rise Characteristics from FINE and UNSCOMP](image)
Despite the similarity of the FINE and UNSCOMP pressure rises there are major differences in the flow fields produced by these two codes. As shown in Figure 6-2 and Figure 6-3 UNSCOMP (which is an Euler solver) does not capture the hub separation present in the FINE flow fields.

Figure 6-2 Axial Flow at Rotor Exit from UNSCOMP and FINE

Figure 6-3 Axial Flow at Stator Exit from UNSCOMP and FINE
Figure 6-2 and Figure 6-3 show the axial velocity at rotor and stator exits from UNSCOMP and from FINE. The UNSCOMP flow field has higher axial velocities in both the rotor and the stator hub regions. There does not appear to be separation at the stator hub in the UNSCOMP calculation, but there is in the FINE flow field.

6.1.1 Comparison with SLC

To address the differences between the UNSCOMP flow fields and those predicted by FINE a Streamline Curvature (SLC) simulation of the compressor was completed using as input the loss and deviation predicted by FINE.

Figure 6-4 Axial Velocity Rotor Exit from UNSCOM, FINE, and SLC
Figure 6-4 and Figure 6-5 show axial velocities from FINE, SLC, and UNSCOMP at rotor and stator exits. The loss and deviation in the SLC calculations is taken from the FINE results. The SLC results also do not indicate the hub separation seen in FINE, but the flow field is similar to that given by UNSCOMP. The main point is that the SLC results are derived using different outputs from FINE than those used in UNSCOMP but the flow fields are similar. This provides confidence that the UNSCOMP solution (steady, axisymmetric, inviscid) is consistent with the inputs. However, it raises concerns about the assumptions made in the body force extraction and the UNSCOMP flow solver. This will be discussed further in Chapter 7, section 7.4.

6.2 Flow Fields using Extended Forces

To examine whether the set of body forces is self-consistent, Figure 6-6 shows results from an UNCSOMP computation in which the local body forces are functions of the local flow, so the blade rows respond in a manner similar to the actual situation.
The UNSCOMP flow fields in Figure 6-3 did not show the hub separation seen in the FINE results. The UNSCOMP flow fields in Figure 6-6 do show the development of hub separation as the flow is decreased. However the extent of the hub flow defect is much less than in the FINE results.

![Figure 6-6 Stator Exit Velocity from FINE and UNSCOMP for Different Operating Conditions](image)

Figure 6-7 shows the pressure rise from the version of UNSCOMP which accounts for body force dependence on local flow coefficient along with the pressure rise from FINE. The addition of the extended body force distributions allows UNSCOMP to converge for flow coefficients as low as 70% of the design flow, lower than the anticipated stall point for this compressor. Therefore the body force modeling procedure provides a useful range of operating points for stall prediction. At this time stall prediction with UNSCOMP is ongoing.
The results in this section are not conclusive but they do give a positive indication that
the body force extraction methodology can provide capability adequate for the stall
prediction procedure. The flow fields produced in UNSCOMP show hub separation in
qualitative accord with FINE. Also UNSCOMP can produce results for flow coefficients
lower than the anticipated stall point. The next question to address is whether the results
from this particular set of body forces are able to capture the stall onset behavior.
Chapter 7

Future Work

The work included in this thesis is one part of a larger project focused on stall prediction. There is future work suggested by the thesis which is of immediate interest. This chapter details some possible improvements to the body force procedure methodology in short-term as well as long-term.

7.1 Replacing FINE with T-block

The current three dimensional CFD results were obtained using NUMECA’s FINEturb which produced a lower pressure rise than observed experimentally. Recent results from the T-block code of John Denton, run in unsteady mode, have been very promising. As shown in Figure 7-1 the pressure rise characteristic from T-block is much closer to the experiment than the FINE characteristic. In addition it appears that the unsteady T-block code can provide converged solutions at lower flow coefficients than FINE.

Figure 7-1 Pressure Rise Characteristics from Measurement, FINEturb, UNSCOMP, and T-block
We thus plan to replace FINE with T-block in the body force extraction procedure. One additional step required before the T-block results can be processed is to time average the unsteady flow fields. Other than the time average the force extraction procedure is the same.

7.2 Improvements to the Force Extension Procedure

Currently 2D CFD results for the mean radius of the compressor are used to construct body force curves at low flow coefficients. Other computational results are now becoming available to augment the current procedure.

7.2.1 Two Dimensional CFD Body Force Curves at Hub and Tip

To take account of the difference in blading along the span in the MHI we are carrying out (T. Walker) two dimensional CFD calculations for sections near the hub and tip (10% and 90% span) of the machine. These should provide more accurate force curves for these regions. Body force curves at the hub, mean, and tip of the blade could form the basis for a radially interpolated set of curves that would apply to all radii.

7.2.2 SLC Body Force Curves Past Stall

The two dimensional CFD results do not account for the streamline contraction and radial displacement present in the real machine. A streamline curvature (SLC) code can capture these effects. We are working (Q. Li) to implement the loss and deviation from the two dimensional CFD results as inputs for the SLC code in order to improve the description of axisymmetric flow below the stall point.
These SLC results would be used to create new body force curves for the extension procedure, replacing the two dimensional CFD based curves currently used. The current extension procedure is based on determining overall blade behavior and using CFD to provide the distribution of local blade forces. Therefore the SLC simulations only need to provide flow information at the inlet and exit of the blade rows.

7.3 Further Interpretation of the Body Forces

The body force modeling procedure has concentrated on reproducing the blade forces present in a real flow. However a need exists to interpret the effect of these forces in terms of traditional quantities such as pressure rise, streamlines, and flow turning. A theoretical study of the effects of changing axial and tangential force on the flow, both on a local and gross level, would provide more in understanding of the interaction between the body force and the flow field. Ideally this deeper understanding of the body forces could enable the ability to implement a desired change to the compressor flow by adjusting the body forces.

7.4 Reconciling the Differences Between UNSCOMP and Input Flow Fields

Section 6.1 discussed a comparison of the flow fields produced by UNSCOMP and streamline curvature when using three-dimensional CFD results from FINE as input. The UNSCOMP results were expected to be much closer to the input flow, and the reason for the differences between the UNSCOMP and FINE flow fields has not yet been adequately explained. A possible source for this discrepancy is that the mixing planes present in the FINE calculation create inconsistencies in the body forces extracted from
the flow. The unsteady T-block does not use mixing planes and therefore may reduce or
eliminate the discrepancy between the input and predicted flow fields.

7.5 General Suggestions for Future work

7.5.1 SLC Based on a Blade Force Average

Chapters 2 through 4 demonstrated that the type of averaging selected to create an
axisymmetric flow in turbo machinery can have a significant impact on the body forces.
Stream Line Curvature codes are a convenient tool for the early design of compressors;
SLC codes predict axisymmetric flows. A stream line curvature code based around a
body force average of the flow could help incorporate at least some parts of the stall
prediction methodology into the early design of a compressor.

7.5.2 Sensitivity of the Procedure to the Spatial Fidelity of the Force

The current procedure is focused on providing a detailed body force distribution in the
axial and radial direction. The stall prediction procedure, as initially demonstrated with
simple distributions (Gong [1]), is not accurate enough for use in design. It would be
useful to know the minimum spatial fidelity needed in the body force distribution to
obtain a suitably accurate stall prediction from the procedure.
Chapter 8

Summary and Conclusions

1. This thesis presents a methodology to produce body force distributions that represent a compressor from design flow to well past stall. The methodology extracts body forces from axisymmetric flow fields derived from three-dimensional CFD and extends the description to flow coefficients below which the three-dimensional solutions are available through the use of two dimensional CFD flow fields. Both the extraction and the extension of the body forces can also be performed with flow fields produced by stream line curvature codes or experimental measurements.

2. The effectiveness of the body forces produced though this methodology at reproducing the flow field and pressure rise from sample compressor has been tested. Pressure rise is reproduced along with the hub separation, although the hub separation appears at a lower flow coefficient than observed in the input flow fields.

3. The body forces produced by this methodology cover a sufficient range of flow conditions to attempt to model the stall behavior for a compressor of interest. The stall prediction is ongoing.

4. A specific form of averaging, the blade force average, is presented for use in this methodology. The blade force average included contributions from the θ
nonuniformities and thus allows the blade forces to be determined from an axisymmetric description of a three dimensional flow field. The effect on the body forces extraction of using an average other than the blade force average was examined in Chapter 4, and it is found to be significant (up to 100% of the blade force) for near stall flows in the hub and tip regions of the machine.
Appendix A

Derivation of the Governing Equations Expressed in Flux Variables

This flux variable form of the governing equations was developed by Yifang Gong and these derivations are taken from his personal communications [10].

Starting with conservation of mass:

$$\frac{\partial (\lambda \rho)}{\partial t} + \frac{\partial (\lambda \rho V_x)}{\partial x} + \frac{\partial (\lambda \rho V_\theta)}{r \partial \theta} + \frac{\partial (r \lambda \rho V_r)}{r \partial r} = 0$$

Rearrange to the following form:

$$\frac{\partial (\lambda \rho)}{\partial t} + \frac{\partial (\lambda \rho V_x)}{\partial x} + \frac{\partial (\lambda \rho V_\theta)}{\partial \theta} + \frac{\partial (r \lambda \rho V_r)}{r \partial r} = 0$$

Conservation of momentum in cylindrical coordinates:

$$\frac{\partial V_x}{\partial t} + V_x \frac{\partial V_x}{\partial x} + V_\theta \frac{\partial V_x}{r \partial \theta} + V_r \frac{\partial V_x}{\partial r} = -\frac{1}{\rho} \frac{\partial P}{\partial x} + F_x$$

$$\frac{\partial V_\theta}{\partial t} + V_x \frac{\partial V_\theta}{\partial x} + V_\theta \frac{\partial V_\theta}{r \partial \theta} + V_r \frac{\partial V_\theta}{\partial r} + \frac{V_r V_\theta}{r} = -\frac{1}{\rho} \frac{\partial P}{\partial \theta} + F_\theta$$

$$\frac{\partial V_r}{\partial t} + V_x \frac{\partial V_r}{\partial x} + V_\theta \frac{\partial V_r}{r \partial \theta} + V_r \frac{\partial V_r}{\partial r} - \frac{V_\theta^2}{r} = -\frac{1}{\rho} \frac{\partial P}{\partial r} + F_r$$

Multiplying the $\theta$ momentum equation by $r$ yields:

$$\frac{\partial r V_\theta}{\partial t} + V_x \frac{\partial r V_\theta}{\partial x} + V_\theta \frac{\partial r V_\theta}{r \partial \theta} + V_r \frac{\partial r V_\theta}{\partial r} + \frac{1}{\rho} \frac{\partial P}{\partial \theta} + r F_\theta$$

Noting:

$$V_r \left( r \frac{\partial V_\theta}{\partial r} \right) = V_r \left( \frac{\partial r V_\theta}{\partial r} - V_\theta \frac{\partial r}{\partial r} \right) = V_r \frac{\partial r V_\theta}{\partial r} - V_r V_\theta$$

I will continue by examining each component of momentum separately:
**X momentum**

Start by multiplying by $\lambda r$ and adding $V_x \cdot \text{continuity}$, remember that continuity $= 0$ so this is in essence adding zero to both sides of the equation:

$$r \rho \lambda \frac{\partial V_x}{\partial t} + r \rho \lambda V_x \frac{\partial V_x}{\partial x} + r \rho \lambda V_\theta \frac{\partial V_x}{\partial \theta} + r \rho \lambda V_r \frac{\partial V_x}{\partial r} = -\lambda r \frac{\partial P}{\partial x} + r \lambda \rho F_x$$

$$+ V_x \frac{\partial (\lambda r V_x)}{\partial t} + V_x \frac{\partial (\lambda r V_x)}{\partial x} + V_x \frac{\partial (\lambda r V_\theta)}{\partial \theta} + V_x \frac{\partial (\lambda r V_r)}{\partial r} = 0$$

Becomes:

$$\frac{\partial (\lambda r V_x)}{\partial t} + \frac{\partial (\lambda r V_x)}{\partial x} + \frac{\partial (\lambda r V_x)}{\partial \theta} + \frac{\partial (\lambda r V_x)}{\partial r} = -\lambda r \frac{\partial P}{\partial x} + r \lambda \rho F_x$$

Or:

$$\frac{\partial (\lambda r V_x)}{\partial t} + \frac{\partial \left[\lambda r (\rho V_x^2 + P)\right]}{\partial x} + \frac{\partial (\lambda r V_x V_\theta)}{\partial \theta} + \frac{\partial (\lambda r V_x V_r)}{\partial r} = r P \frac{\partial \lambda}{\partial x} + r \lambda \rho F_x$$

Where:

$$\frac{\partial (P \lambda r)}{\partial x} = \lambda r \frac{\partial P}{\partial x} + r P \frac{\partial \lambda}{\partial x}$$
\(\theta\) momentum

Start by multiplying by \(\lambda r\) and adding \(rV_\theta^*\)continuity, remember that continuity = 0 so this is in essence adding zero to both sides of the equation:

\[
\lambda r_\theta \frac{\partial rV_\theta}{\partial t} + \lambda r_\theta \frac{\partial rV_\theta}{\partial x} + \lambda r_\theta \frac{\partial V_\theta}{\partial \theta} + \lambda r_\theta \frac{\partial rV_\theta}{\partial r} = -\lambda r \frac{\partial P}{\partial \theta} + \lambda r^2 F_\theta
\]

\[+ rV_\theta \frac{\partial (\lambda rP)}{\partial t} + rV_\theta \frac{\partial (\lambda rV_\theta^2)}{\partial x} + rV_\theta \frac{\partial (\lambda rV_\theta^2)}{\partial \theta} + rV_\theta \frac{\partial (r\lambda rV_\theta)}{\partial r} = 0\]

Becomes

\[
\frac{\partial (\lambda r^2 V_\theta)}{\partial t} + \frac{\partial (\lambda r^2 V_\theta^2)}{\partial x} + \frac{\partial (\lambda r^2 V_\theta)}{\partial \theta} + \frac{\partial (\lambda r^2 V_\theta^2)}{\partial r} = -\lambda r \frac{\partial P}{\partial \theta} + \lambda r^2 F_\theta
\]

Or:

\[
\frac{\partial (\lambda r^2 V_\theta)}{\partial t} + \frac{\partial (\lambda r^2 V_\theta^2)}{\partial x} + \frac{\partial \left[\lambda r (V_\theta^2 + P)\right]}{\partial \theta} + \frac{\partial (\lambda r^2 V_\theta^2)}{\partial r} = rP \frac{\partial \lambda}{\partial \theta} + \lambda r^2 F_\theta
\]

Where:

\[
\frac{\partial (P\lambda)}{\partial \theta} = \lambda r \frac{\partial P}{\partial \theta} + rP \frac{\partial \lambda}{\partial \theta}
\]
R momentum

Start by multiplying by $\lambda \rho r$ and adding $V_r \cdot \text{continuity}$, remember that continuity = 0 so

this is in essence adding zero to both sides of the equation:

$$\frac{\partial V}{\partial t} + \lambda \rho \frac{V_x}{\partial x} + \lambda \rho V_\theta \frac{\partial V}{\partial \theta} + \lambda \rho V_r \frac{\partial V}{\partial r} - \lambda \rho V_\theta^2 = -\lambda \rho \frac{\partial P}{\partial r} + \lambda \rho r F_r$$

$$+ V_r \frac{\partial (\lambda \rho)}{\partial t} + V_r \frac{\partial (\lambda \rho V_x)}{\partial x} + V_r \frac{\partial (\lambda \rho V_\theta)}{\partial \theta} + V_r \frac{\partial (r \lambda \rho V_r)}{\partial r} = 0$$

Becomes:

$$\frac{\partial (\lambda \rho V_r)}{\partial t} + \frac{\partial (\lambda \rho V_x)}{\partial x} + \frac{\partial (\lambda \rho V_\theta)}{\partial \theta} + \frac{\partial (\lambda \rho V_r^2)}{\partial r} = \lambda \rho V_\theta^2 - \lambda r \frac{\partial P}{\partial r} + \lambda \rho r F_r$$

Or:

$$\frac{\partial (\lambda \rho V_r)}{\partial t} + \frac{\partial (\lambda \rho V_x)}{\partial x} + \frac{\partial (\lambda \rho V_\theta)}{\partial \theta} + \frac{\partial (r \lambda \rho (V_r^2 + P))}{\partial r} = r P \frac{\partial \lambda}{\partial r} + \lambda P + \lambda \rho V_\theta^2 + \lambda \rho F_r$$

Where:

$$\frac{\partial (P \lambda r)}{\partial r} = \lambda r \frac{\partial P}{\partial r} + P \left[ \lambda \frac{\partial r}{\partial r} + r \frac{\partial \lambda}{\partial r} \right]$$
The governing equations can now be written in the following form:

\[
\frac{\partial}{\partial t} \begin{bmatrix}
\lambda r p \\
\lambda r p V_x \\
\lambda r^2 p V_\theta \\
\lambda r p V_r
\end{bmatrix} + \frac{\partial}{\partial x} \begin{bmatrix}
\lambda r p V_x \\
\lambda r p V_x^2 + r P \\
\lambda r p V_x r V_\theta \\
\lambda r p V_x V_r
\end{bmatrix} + \frac{\partial}{\partial \theta} \begin{bmatrix}
\lambda r p V_\theta \\
\lambda r p V_x V_\theta \\
\lambda r p V_\theta^2 + r P \\
\lambda r p V_r V_\theta
\end{bmatrix} + \frac{\partial}{\partial r} \begin{bmatrix}
\lambda r p V_r \\
\lambda r p V_x V_r \\
\lambda(r p V_r^2 + r P)
\end{bmatrix} = \begin{bmatrix}
0 \\
\lambda r p F_x + r P \frac{\partial \lambda}{\partial x} \\
r P \frac{\partial \lambda}{\partial \theta} + \lambda r p F_\theta \\
\lambda r V_\theta^2 + \lambda P + \lambda r p F_r + r P \frac{\partial \lambda}{\partial r}
\end{bmatrix}
\]

For a steady flow this equation is represented with the flux variables “F”, “G”, and “H”

\[
\frac{\partial}{\partial x} F + \frac{\partial}{\partial \theta} G + \frac{\partial}{\partial r} H = S
\]

Where “S” is the source term
Control Volume Form

The flux variable form of the governing equations can be recast into a control volume form, shown in the following equations for an axisymmetric flow:

\[
\begin{align*}
\frac{F_x A_{\text{ axial}} + F_z A_{\text{ axial}} - (F_x A_{\text{ axial}} + F_z A_{\text{ axial}})}{\text{Volume}_{\text{cell}}} + \frac{H_{\text{ axial}} A_{\text{ axial}} + H_z A_{\text{ axial}} - (H_x A_{\text{ axial}} + H_z A_{\text{ axial}})}{\text{Volume}_{\text{cell}}} & = \left[ \begin{array}{c}
0 \\
\lambda r \rho F_x + r P \frac{\partial \lambda}{\partial x} \\
\lambda r \rho F_\theta \\
\lambda \rho V_x^2 + \lambda P + \lambda r \rho F_r + r P \frac{\partial \lambda}{\partial r}
\end{array} \right] \\
\lambda r \rho \cdot \text{Volume}_{\text{cell}}
\end{align*}
\]

The control volume can be solved explicitly for each force component, shown here for the axial force:

\[
F_x = \frac{(\lambda (r \rho V_x^2 + r P) A_{\text{ axial}} + (\lambda (r \rho V_x^2 + r P) A_{\text{ axial}} - (\lambda (r \rho V_x^2 + r P) A_{\text{ axial}} + (\lambda (r \rho V_x^2 + r P) A_{\text{ axial}})}{\text{Volume}_{\text{cell}}} \\
+ \frac{(\lambda r \rho V_x V_r) A_{\text{ radial}} + (\lambda r \rho V_x V_r) A_{\text{ radial}} - ((\lambda r \rho V_x V_r) A_{\text{ radial}} + (\lambda r \rho V_x V_r) A_{\text{ radial}})}{\text{Volume}_{\text{cell}}} \lambda r \rho \cdot \frac{\partial \lambda}{\partial x}}
\]

Conceptually the above equation is a balance of the axial momentum and pressure determined on all the surfaces of the control volume.
APPENDIX B

Body Force and Quadratic Force Plots for the Sample Compressor

B.1 Body Force Distributions in the Compressor

The following plots show the magnitude of the quadratic force in the sample compressor. The forces are shown for three representative flow coefficients, one near design, one near stall, and one flow coefficient in between. The plots show a distribution of force vectors along the blade, the axial location is normalized by the longest value of axial chord for the blade being plotted. Plotting the force vectors is a convenient way of displaying the force in the axial and Θ directions simultaneously. The forces are normalized by

\[
\frac{1}{2} \frac{\sum \nabla^2 \phi}{U_{\text{wheel}}^2 L_{\text{compressor}}}
\]

since the forces extracted by the code are expressed in force per unit mass, i.e., units of acceleration. In these plots magnitude of the vectors has been scaled by a factor of 50 to keep the vector lengths from obscuring the axial locations of the vectors.
Forces in the Inlet Guide Vane

Figure B-1 IGV Force Distributions 100% Design Flow

Figure B-2 IGV Force Distribution 87% Design Flow

Figure B-3 IGV Force Distribution, 80% Design Flow (near stall)
Forces in the Rotor

Figure B-4 Rotor Force Distribution 100% Design Flow

Figure B-5 Rotor Force Distribution 87% Design Flow

Figure B-6 Rotor Force Distribution 80% Design Flow (near stall)
Forces in the Stator

Figure B-7 Stator Force Distribution 100% Design Flow

Figure B-8 Stator Force Distribution 87% Design Flow

Figure B-9 Stator Force Distribution 80% Design Flow (near stall)
Forces in the Outlet Guide Vanes

Figure B-10 OGV Force Distribution 100% Design Flow

Figure B-11 OGV Force Distribution 87% Design Flow

Figure 12 OGV Force Distribution 80% Design Flow (near stall)
B.2 Quadratic Force Distributions in the Compressor

The quadratic force was determined by first calculating the blade force using a body force average and the subtracting the force extracted from an area average of the flow field.

The sample compressor has radial blades, leading to little to no radial force present in the compressor. Therefore I will only examine the quadratic forces in the axial and tangential directions.

The following plots show the magnitude of the quadratic force in the sample compressor. The quadratic forces are shown for three representative flow coefficients, one near design, one near stall, and one flow coefficient in between. The plots show a distribution of force vectors along the blade, the axial location is normalized by the longest value of axial chord for the blade being plotted. Plotting the force vectors is a convenient way of displaying the force in the axial and Θ directions simultaneously. The forces are

\[ \frac{1}{2} \frac{U_{wheel}^2}{L_{compressor}} \]

normalized by since the forces extracted by the code are expressed in force per unit mass, ie units of acceleration. As in the Body Force sections the vectors magnitude has been scaled down by a factor of 50 for clearer plots.
Quadratic Forces in the Inlet Guide Vane

Figure B-13 IGV Quadratic Force 100% Design Flow

Figure B14 IGV Quadratic Force 87% Design Flow

Figure B-15 IGV Quadratic Force 80% Design Flow (near stall)
Quadratic Forces in the Rotor

Figure B-16 Rotor Quadratic Force 100% Design Flow

Figure B-17 Rotor Quadratic Force 87% Design Flow

Figure B-18 Rotor Quadratic Force 80% Design Flow (near stall)
Quadratic Forces in the Stator

Figure B-19 Stator Quadratic Force 100\% Design Flow

Figure B-20 Stator Quadratic Force 87\% Design Flow

Figure B-21 Stator Quadratic Force 80\% Design Flow (near stall)
Quadratic Forces in the Outlet Guide Vane

Figure B-22 OGV Quadratic Force 100% Design Flow

Figure B-23 OGV Quadratic Force 87% Design Flow

Figure B-24 OGV Quadratic Force 80% Design Flow (near stall)
References


