2006

A Computational Study of Axial Compressor Rotor Casing Treatments and Stator Land Seals

Charles C. Cates
Virginia Commonwealth University

Follow this and additional works at: https://scholarscompass.vcu.edu/etd
Part of the Engineering Commons

© The Author

Downloaded from
https://scholarscompass.vcu.edu/etd/920

This Thesis is brought to you for free and open access by the Graduate School at VCU Scholars Compass. It has been accepted for inclusion in Theses and Dissertations by an authorized administrator of VCU Scholars Compass. For more information, please contact libcompass@vcu.edu.
A COMPUTATIONAL STUDY OF AXIAL COMPRESSOR ROTOR CASING TREATMENTS AND STATOR LAND SEALS

A thesis submitted in partial fulfillment of the requirements for the degree of Master of Science in Mechanical Engineering at Virginia Commonwealth University.

by

CHARLES C. CATES

Bachelor of Science in Mechanical Engineering
Virginia Commonwealth University, Richmond, Virginia, May 2002

Director: JAMES T. MCLESKEY, JR., PH.D.
DEPARTMENT OF MECHANICAL ENGINEERING

Virginia Commonwealth University
Richmond, Virginia
December, 2006
Acknowledgment

First I would like to thank my advisor, Dr. James McLeskey, for his excellent advice, persistence, and steadfast patience over the last several years. His guidance and support helped me immensely as I neared the completion of this thesis. I would also like to thank Dr. Dan Cook and Dr. Eric Sandgren, who guided me at the beginning of this path. Dr. Steven Wellborn of the Compressor Aerodynamics group at Rolls-Royce Corporation in Indianapolis, Indiana was also a great source of ideas, suggestions, and assistance in learning ADPAC and Plot3D.

I would like to thank Dr. P. Worth Longest and Dr. David Primeaux for their willingness to serve as members of my thesis advisory committee.

On a personal note, I would like to thank my parents Steve and Janet and sister Alice for their constant love, motivation, and support through many years of school, and for being the shaping force behind my love of science and engineering.

Finally, I would like to thank my fiancée Kimberly. Her love, support, and understanding through the final months of completing this thesis were absolutely essential.
# Table of Contents

LIST OF FIGURES ........................................................................................................... V

LIST OF TABLES ............................................................................................................... VIII

ABSTRACT ........................................................................................................................ IX

CHAPTER 1. INTRODUCTION .......................................................................................... 1
  1.1 Motivation .................................................................................................................. 1
  1.2 Objective of Study ..................................................................................................... 2
  1.3 Outline ....................................................................................................................... 3

CHAPTER 2. BACKGROUND AND TERMINOLOGY ......................................................... 4
  2.1 Areas of Interest ....................................................................................................... 4
    2.1.1 Stator Labyrinth Seals ...................................................................................... 7
    2.1.2 Compressor Rotor Casing Treatment .............................................................. 9
  2.2 Background and CFD History .................................................................................. 13
  2.3 Computational Resources ....................................................................................... 16
  2.4 Literature Survey ..................................................................................................... 17
    2.4.1 Stator Seals ...................................................................................................... 17
    2.4.2 Compressor Rotor Casing Treatments ............................................................ 23

CHAPTER 3. THEORY ..................................................................................................... 28
  3.1 Governing Equations .............................................................................................. 28
  3.2 ADPAC CFD Code .................................................................................................. 32
    3.2.1 ADPAC Solution Process ............................................................................... 33
    3.2.2 ADPAC Procedure of Current Work ................................................................ 38
  3.3 Stator Seals and Fluent Background ....................................................................... 45
    3.3.1 Fluent Procedure of Current Work .................................................................. 46

CHAPTER 4. COMPUTATIONAL RESULTS AND DISCUSSION ...................................... 52
  4.1 Stator Seal Cavity Investigation ............................................................................. 52
    4.1.1 Stator One Geometry Results ......................................................................... 53
    4.1.2 Stator Two Seal Cavity Results ..................................................................... 61
    4.1.3 Stator Seal Study Conclusions ....................................................................... 69
  4.2 Compressor Rotor Casing Treatment Study .......................................................... 71
    4.2.1 Circumferential Grooves Results ................................................................... 73
    4.2.2 Compressor Rotor Casing Treatment Study Conclusions ............................ 96

CHAPTER 5. CONCLUSIONS ........................................................................................... 98
List of Figures

Figure 1: Cutaway of a 9-Stage Compressor with Detail of Stator-Rotor-Stator Stage with Stator Land Seal, From Heidegger et. al. [27] ............................................................................................................. 5

Figure 2: Solid frame mock-up of Stator-Rotor-Stator stage with Circumferential Groove Rotor Endwall Casing Treatment .......................................................................................................................... 7

Figure 3: Expanded view of stator labyrinth seals; rotating components in dark shading, non-rotating components in light shading ............................................................................................................. 8

Figure 4: Sample Compressor Map, Percent Corrected Speed Lines in Black, Compressor Efficiency Islands in Blue .............................................................................................................................................. 10

Figure 5: Wireframe Image of all Solid Surfaces in Stator-Rotor-Stator Stage; With Circumferential Groove Casing Treatment Above Rotor ................................................................................................................................... 37

Figure 6: View of Full Modeled Flowpath with Solid Surfaces Highlighted ......................................................................................................................................................... 38

Figure 7: Zoomed in View of Rotor Endwall Casing Treatment and Rotor Blade Tip Gap ...................................................................................................................................................... 39

Figure 8: View From Above Rotor Passage; Blade Airfoil Profile in White, Case Mesh and Groove Mesh Visible in Pink .................................................................................................................................................. 40

Figure 9: Rotor Casing Treatment Mesh Showing Point-to-Point Correlation of Mesh ......................................................................................................................................................... 41

Figure 10: Rotor Hub, Blade, Endwall, and Casing Treatment Mesh with Stator Passages Outlined on Either Side .................................................................................................................................................. 42

Figure 11: Full Passage Mesh; Rotating Components in Red; Non-Rotating Components in Pink .................................................................................................................................................. 43

Figure 12: Stator 1 Seal Area Detail With Measurement Stations 1-4 Labeled ......................................................................................................................................................... 48

Figure 13: Stator 1 Seal Tooth Gap Detail; Left: Baseling, 0.010"; Top Right: 0.005"; Bottom Right: 0.015" ......................................................................................................................................................... 49

Figure 14: Stator 2 Detail with Measurement Stations Labeled ......................................................................................................................................................... 50
Figure 16: Stator 1 Cavity Reference Locations

Figure 21: Axisymmetrically Averaged Velocity Vectors; Top: Baseline, 0.010", Bottom Left: 0.005".
Bottom Right: 0.015"

Figure 22: Stator 2 Cavity Reference Locations

Figure 23: Stator 2 Static Pressure Throughout Cavity

Figure 24: Stator 2 Total Pressure Throughout Cavity

Figure 25: Stator 2 Tangential Velocity Throughout Cavity

Figure 26: Stator 2 Total Temperature Throughout Cavity

Figure 27: Axisymmetrically Averaged Velocity Vectors Through Stator 2 Seal Cavity; Left: 0.0 lbm/s Through Center Seal, With Detail, Right: 0.32 lbm/s Through Center Seal

Figure 28: Baseline Untreated Rotor Convergence Plots

Figure 29: Rotor Mesh With Casing Treatment Convergence Plots

Figure 30: Efficiency and Pressure Ratio versus Compressor Inlet Corrected Flow

Figure 31: Contour Plot Showing Separation Bubble off Rotor Suction Surface, No Casing Treatment

Figure 32: Contour Plot Showing Separation Surface off Rotor Suction Surface, With Casing Treatment

Figure 33: Streamlines From the Rotor Tip Colored by Velocity Magnitude, No Casing Treatment

Figure 34: Streamlines From the Rotor Tip Colored by Velocity Magnitude, With Casing Treatment

Figure 35: Reverse Angle of Particle Traces Released From Rotor Tip Suction Side, No Casing Treatment

Figure 36: Reverse Angle of Particle Traces Released From Rotor Tip Suction Side, With Casing

Figure 37: Flow in to and out of Grooves, Looking Axially Backwards From Suction Surface

Figure 38: Flow Injected in to and out of the First Groove

Figure 39: Flow Injected in to and out of the Second Groove

Figure 40: Flow Injected in to and out of the Third Groove

Figure 41: Flow Injected in to and out of the Fourth Groove

Figure 42: Flow Injected in to and out of the Fifth Groove
Figure 43: Streamlines Showing Combination of Outflow From All Five Circumferential Grooves 94
Figure 44-48: Grooves 1-5 (Left to Right, Top to Bottom) Showing Structure of Flow Immediately Above Rotor Blade 95
List of Tables

Table 1: Stator 1 Seal Cavity Mass Flow as a Percentage of Total Corrected Stator Mass Flow 54
Table 2: Stator 2 Cavity Mass Flow as a Percentage of Total Corrected Stator Mass Flow 62
Abstract

A COMPUTATIONAL STUDY OF AXIAL COMPRESSOR ROTOR CASING TREATMENTS AND STATOR LAND SEALS

By Charles C. Cates, M.S.M.E.

A thesis submitted in partial fulfillment of the requirements for the degree of Master of Science in Mechanical Engineering at Virginia Commonwealth University.

Virginia Commonwealth University, 2006

Major Director: James T. McLeskey, Jr., Ph.D.
Department of Mechanical Engineering

As fuel prices soar ever higher, aircraft manufacturers and their airline customers demand that the next generation of engines used on their aircraft push the limits of efficiency and capability. This study consists of a computational examination of two currently accepted methods of axial compressor performance improvement in terms of surge margin and efficiency, rotor casing treatments and stator land seals.

ADPAC and Fluent CFD solvers were used in the analysis of circumferential groove casing treatments and two types of stator seals, one typical of a front stage stator
and one typical of a rear stage stator. The computational solutions and visualizations allowed for greater understanding of the complex flows inherent in each of these features. It was found that rotor tip vortex control plays a large part in the surge margin gains from a circumferential groove casing treatment. The efficiency gains of knife seals were dependent primarily on the gap size of the seals.
Chapter 1. INTRODUCTION

1.1 Motivation

With the recent introduction of high-efficiency airliners such as the Boeing 787 and the Airbus A380, greater pressure has been placed on engine manufacturers to produce engines with both higher peak efficiency and higher peak performance. For example, Boeing claims that when the 787 is first flown in 2007, it will use 15 to 20 percent less fuel on a per-passenger basis than other current products in the 200 to 250-passenger range. They expect that approximately half of that fuel economy boost will come from gains in engine efficiency alone [1]. Likewise, Airbus cites 13% lower fuel burn than its closest competitor to the A380, with the fuel economy expected to be 95 miles per gallon per passenger. The A380 is also expected carry 35% more passengers at a 15% lower seat cost per mile with the noise level at take-off only half that of its current closest competitor [2].

Aircraft manufacturers and their airline customers want their aircraft to be able to "fly higher, faster, farther, cleaner, quieter, and more efficiently" than ever before. All of those qualifications relate directly to the performance of their gas turbine engines. To design an engine with such demanding expectations requires outstanding predictive
capability of every system and component on that engine to ensure that accurate models are available to the design team.

Computational fluid dynamics (CFD) analysis has long been used as an investigative tool for many different types of flows through turbomachinery. Using CFD gives designers the opportunity to make quick and inexpensive adjustments in geometry using automated grid generation tools, as compared to having to machine and experimentally test new parts each time a change is desired. CFD also has the ability to isolate and track a single geometric parameter allowing designers to focus in on one particular area of interest. The numerical solutions can provide at a minimum guidance on trends, and higher order accurate solutions can predict performance parameters with a high degree of certainty.

1.2 Objective of Study

The objective of this work is to perform a computational study examining current accepted methods of efficiency and surge margin enhancement in turbomachinery. Specific areas of interest lie in the modeling of a circumferential groove compressor rotor casing treatment and stator land knife seals.

Currently it is understood that rotor casing treatments have a beneficial effect on compressor surge margin with only a slight penalty on overall operating efficiency, but the mechanism behind this improvement is not fully understood. Similarly with stator knife seals, it is known that the seals have a large direct correlation to operating efficiency of the compressor, however the deterministic variables defining the seal
performance are not fully understood. It is the objective of this work to build on the understanding of these mechanisms, hopefully enabling future designers to create even more effective designs.

1.3 Outline

In Chapter 2, the background and other interest in this work will be presented and important terms will be defined. A survey of published literature will also be presented showing the efforts of previous researchers. In Chapter 3, the theory behind computational fluid flow and current codes and solution techniques will be laid out. It will also discuss the procedures used in the current study. Chapter 4 will present the results of the computational study and a discussion of their meaning and significance. Finally, Chapter 5 will include the conclusions of this study and recommendations for future areas of study to further advance the understanding of this field of study.
Chapter 2. BACKGROUND AND TERMINOLOGY

2.1 Areas of Interest

CFD analysis is a useful technique in different types of fluid flows: internal and external, subsonic and supersonic. However CFD becomes even more useful in situations where there is a high degree of difficulty in obtaining experimental data. These situations could include areas where the geometry is specified to have extremely tight tolerances or the geometry is complex in general. They could also include areas that have extremely high temperatures, pressures, or inertial forces where instrumentation is unlikely to survive. In such instances, instrumentation costs can skyrocket. In high speed turbomachinery like an axial compressor many of these conditions exist, making CFD a very valuable design tool indeed. This study in particular examines two components of axial compressors, both of which have instrumentation placement challenges.

For reference, Figure 1 gives a good cutaway view of a typical 9-stage axial compressor. This particular design has 5 stages of variable stator geometry (CVG). Along the bottom of the picture is the centerline of the engine, with the hub. Attached to the hub are compressor wheels, which have the compressor blades attached in the
flowpath of the engine. At the outer portion is the compressor case, which houses the compressor variable geometry hardware and has the stators mounted. Generally, the outer portion is stationary while the inner is rotating.

Figure 1: Cutaway of a 9-Stage Compressor with Detail of Stator-Rotor-Stator Stage with Stator Land Seal, From Heidegger et. al. [27]
The first major area of interest lies in the stator labyrinth seals and their associated cavities. This is the area at the compressor hub that allows clearance between the rotating shaft and the stationary stator vanes. Figure 1 shows an expanded view of a sample seal area. Note that the stator is attached to the compressor case and therefore is stationary. The rotors are attached to the spinning hub. The rotor and stator blades are in the flowpath, and there is clearance necessary to prevent the stator from rubbing against the rotating hub.

The second major area of interest is at the rotor endwall where complex types of compressor rotor casing treatments are applied. Figure 2 shows a full annotated stator-rotor-stator compressor stage. Rotating components are displayed in red and non-rotating components are displayed in pink. The white arrows show the direction of flow movement through the geometry. That particular endwall treatment is known as circumferential grooves. The next sections detail these areas of interest, and their significance in compressor performance parameters such as efficiency, pressure rise, and compressor stall and surge.
2.1.1 Stator Labyrinth Seals

Stator seals are a requirement in every high-speed axial compressor, with the purpose being to provide clearance to separate the stationary stator blade and rotating main shaft while also preventing so-called “leakage flow” from moving through the cavity. Because the function of the stator blade is to act as a diffuser by turning the flow...
back into the axial direction, decreasing the absolute velocity, and increasing the static pressure, the fluid flows backwards through the seal cavity with respect to the flow in the main cavity. Obviously there are losses associated with this flow as a result of the interaction between the main axial direction flow and the seal cavity backward flowing path. Losses also occur due to the increase in work from the loss of static pressure in the main flow path. These losses should be minimized so as not to incur a penalty in overall compressor efficiency.

![Diagram of stator labyrinth seals](image)

*Figure 3: Expanded view of stator labyrinth seals; rotating components in dark shading, non-rotating components in light shading*

The representative seal cavity shown in Figure 3 is a fairly typical two-knife labyrinth seal geometry found in a high-speed compressor. For reference, the dark shaded areas are rotating while the light shaded areas are non-rotating. As discussed earlier, the flow in the main section with the blades and vanes moves to the right, from the upstream rotor to the downstream rotor. The stator blade decelerates the flow and
causes a rise in static pressure. Because the flow downstream of the stator has a higher static pressure than that upstream of the stator, the fluid in the seal cavity is forced backwards (right to left in Figure 3) with respect to the main flow.

As compressor designs become ever more aggressive in terms of pressure rise across each stage, the differential across the stator seal cavity will become larger. Therefore, the impetus for losses induced from leakage flow will also become larger. To minimize inefficiency and overall negative effects on the compressor due to this backward moving flow and its interaction with the main flow path, its mass rate, flow path, and key deterministic geometric variables must be established and fully understood.

Some interesting work has been done by other researchers in the area of stator seals and is discussed in the literature survey section on Stator Seals.

2.1.2 Compressor Rotor Casing Treatment

The aerodynamic stability of a gas turbine compressor is one of the most important measures available to designers. In fact, aerodynamic stability is one of the fundamental limits in the compressor design process. As a background in the general philosophy, the purpose of a compressor is to provide a stable mass flow rate of air and a stable increase in pressure from the surrounding atmosphere to the combustor. To represent these goals visually, a compressor map plots the overall pressure rise of the machine against the mass flow rate of the compressor, usually corrected to the compressor’s design point or ISA standard day inlet conditions. Standard day at sea level is 59 degrees Fahrenheit ambient temperature and 14.696 psi ambient pressure. A simple
“cartoon” compressor map is shown in Figure 4. Stability is most often measured by viewing the operating characteristic line, also known as the constant speed line of the compressor, with respect to the surge line.

Figure 4: Sample Compressor Map, Percent Corrected Speed Lines in Black, Compressor Efficiency Islands in Blue

In Figure 4, the operating line of the sample compressor would most likely run somewhat parallel to the surge line and through the peak efficiency islands as speed increases. At the low-pressure rise end of the constant speed line, the compressor is considered to be choked. Compressor choke occurs when the compressor is providing a large mass rate of flow at a shaft speed but is unable to provide any pressure rise. As such, this is a very inefficient point of operation for the compressor.
At the other end of the speed line is the stall point of the compressor. Stall, also known as surge, is an unstable flow condition that is very dangerous in aircraft gas turbine engines, as it can cause excessive vibration, noise, low performance, and can lead to serious compressor damage or complete failure. It occurs when the compressor is providing its maximum amount of pressure rise at a shaft speed and is suddenly unable to sustain that pressure rise. It occurs most frequently in transient operation, especially Bodie maneuvers where the throttle is chopped from maximum to idle then returned to maximum. Surge can also be brought on by inlet distortion, compressor deterioration, foreign objects and debris (FOD), or a number of other variables. Suddenly the compressor experiences reverse flow that can range from benign and recoverable to extremely violent and damaging to the compressor hardware depending on the current operating point of the compressor.

Because of the inefficient operation in a choked condition and the detrimental and possibly devastating effects of compressor surge, engines are designed with the operating point somewhere in the middle of the speed line. This gives the engine a significant degree of both choke margin and stall margin. As the engine wears over time, the performance of the compressor degrades and the stall margin decreases. This is one of many considerations given when determining the amount of stall margin that is necessary for a compressor that will be in-service on an aircraft for many years.

Compressor aerodynamic stall comes in several different varieties, defined by where the stall cell originates. Endwall stall originates at the rotor tips as a result of the vortex that is shed from the tip of every rotor blade as it passes by the compressor case.
The vortices interact with the free-stream flow and cause flow instabilities and separation, leading to the compressor being unable to sustain its pressure rise. The vortices can also shed in random and unpredictable patterns, leading to compressor instability and harmful flow pulsations.

Blade stall occurs in either rotor blades or stator vanes, and is very similar to aerodynamic stall in the wing of an aircraft. A high angle of attack causes a separation bubble off of the suction surface of the blade, which reduces the amount of aerodynamic lift. The stall cell can expand to multiple blades and vanes and prevent the compressor from reaching its required pressure rise. Modern compressors commonly control the onset of blade stall with the use of Compressor Variable Geometry (CVG). With CVG, the compressor stator vanes are pivoted so that the angle of attack can be altered. With the development of CVG blade stall became much less of a concern in design, and compressor operating ranges were extended a great deal.

Hub stall originates at the rotor hub, usually as a result of the boundary layer interaction with the free-stream flow. It occurs primarily in compressors where the majority of the aerodynamic loading is along the hub as opposed to at the blade tips. When the hub boundary layer grows, it restricts the amount of flow that the compressor is able to provide. As the pressure rise and stage loading increases, it can lead to cells where the flow is completely stopped. This condition can rapidly deteriorate into a full compressor surge.

Under many off-design or distorted inlet conditions, aerodynamic stall of the compressor begins in the endwall region, between the rotor blade and the compressor...
casing. Experimental and numerical data [3] [4] have shown that the blade tip loading and interaction of the tip leakage vortex and the passage shock are contributing factors to the inception of endwall stall (as opposed to blade stall [5]). When endwall stall is delayed, the mass flow range and overall compressor operating range is effectively increased. Therefore, development of solid techniques for increasing aerodynamic stability has major benefits to the compressor designer.

Many different methods have been developed for increasing aerodynamic stability including Tip Injection [6], J-grooves [7] [8], Axial Grooves [9], Axial Skewed Slots [10], Recessed Vanes [11], and Circumferential Grooves [12] [13]. Most have been shown to effectively delay stall and give a marked improvement in the compressor operating range. The research on these casing treatments has shed light on the flow features involved with the casing treatment and at the endwall, however the exact reasons for the delay in the inception of stall are still not completely understood. To design better casing treatments in the future, an understanding of the flow features is necessary. Computational fluid dynamics offers an attractive and inexpensive method for visualizing and understanding the reasons behind the increased mass flow range.

2.2 Background and CFD History

The field of Computational Fluid Dynamics is a very large and fast-moving one, with advances being made in both software and hardware on a continual basis. CFD has been used to solve an incredibly diverse range of fluid dynamics related problems including but not limited to aerodynamics, hydrodynamics, combustion, heat flow,
environmental modeling, biofluid flows, and meteorology. CFD was first used in the design process, R&D, and manufacture cycle in the aerospace industry beginning in the 1960s [14]. Other industries began adopting CFD into their product design cycles when they saw the advantages of CFD over traditional experimental-based approaches to product development. Companies especially appreciated the possible reductions in time to market and cost of new designs due to faster design iterations and the increase in safety over traditional experimental based design.

CFD provides a numerical solution to the fundamental equations of fluid flow, discussed thoroughly in Chapter 3, Theory. To structure the problem in such a way that the computer can understand, and then view the results given by the computer, CFD codes have three main parts: a Pre-processor, Solver, and Post-processor.

The pre-processor consists of a user-friendly (hopefully) interface allowing the operator to convert the physical geometry of a design into a computational domain suitable for use with the solver. That computational domain is then converted into a mesh of small sub-domains, known as cells. The accuracy, stability, and quality of the solution is directly dependent upon the quality of the mesh, however the computational time and resources are also directly dependent on mesh quality. For this reason, there is an important trade-off to consider. Because the solution is just a numerical approximation, the more cells a mesh has the more likely the solution is to be accurate. This is known as mesh density. However, with a higher mesh density, the time it takes for the numerical approximation to converge is also increased. It is possible to create a mesh that would take months or more of computation time to converge. In addition to
setting up the initial computational mesh, the pre-processor also allows the user to select
the working fluid and relevant physical phenomena, and apply the proper boundary
conditions to the mesh. Because of the trade-offs to consider and the importance of the
mesh to overall solution accuracy, it is common to spend over half of the total time for a
CFD project on the pre-processing and mesh creation process.

For this investigation, the FLUENT grid generator GAMBIT was used in the
creation of the meshes for the stator seal portion. For the compressor rotor casing
treatment portion, a variant of the NASA TIGG-3D code was used for mesh creation.

The next main part, and truly the brain of a CFD code, is the solver. The solver
computes the numerical solution based upon the grid and conditions supplied by the pre-
processor. Solvers generally use one of two distinct solution techniques: finite element or
finite volume. This defines the way in which the flow variables are approximated and the
governing equations are discretized. The codes of interest in this investigation, Fluent
and ADPAC, use the finite volume formulation. The governing formulae and
discretization techniques of the finite volume method are discussed thoroughly in the
Theory section.

The final piece of CFD is the post-processor. The post-processor takes the large
amount of output data from the solver and converts it into a useful form such as a contour
plot, vector plot, or graph. Some post-processors have the capability to produce movies
for simpler visualization of transient results. The post-processor allows the user to
visualize the results from a CFD simulation in a clear and easy to comprehend format.
For the study of stator seals, the Fluent built-in post processor was used. For the
compressor rotor casing treatment portion, the NASA public domain post processor PLOT3D was used.

2.3 Computational Resources

Each of the computational investigations in this research used a different Computational Fluid Dynamics package. For the stator labyrinth seal study, Fluent was the solver of choice. To create the grids and perform other necessary pre-processing, the Fluent grid generator Gambit was used. For post-processing work, Microsoft Excel was used in creating graphs and the post-processing capabilities of Fluent were used in the creation of all vector and contour plots. In the compressor rotor casing treatment study, the Advanced Ducted Propfan Analysis Code (ADPAC) was used. ADPAC was purpose-written to solve problems related to turbomachinery, specifically with regards to fans and compressors. To visualize the mesh quality and post-process the results given by ADPAC, the NASA PLOT3D visualization code was used.

The hardware used in all of the casing treatment computational studies was a heterogeneous Beowulf-style cluster of Sun ULTRA 30 and ULTRA 60 workstations. The machines had either 128, 256, or 512 megabytes of RAM, and the ULTRA 30 boxes had 250 MHz processors while the ULTRA 60 boxes had 360 MHz processors. All of the machines used Sun's SOLARIS operating system.

The hardware used in the stator seal computational study was a homogeneous Beowulf-style cluster of AMD processor-based computers with the Redhat Linux version 7.3 operating system. Each machine had two 1.8 GHz processors and one gigabyte of
RAM. Hardware interconnection was done with a 100-base T Ethernet network.

2.4 Literature Survey

A great deal of work has been done in the area of computational analysis of turbomachinery flows. As mentioned in a previous section, aerospace companies began to incorporate computational fluid dynamics in their program design and development cycles as early as the 1960s. Modern methods and solution techniques upon which ADPAC is dependent such as Runge-Kutta time stepping and multigrid algorithms were developed in the 1980s by the efforts of researchers like Jameson, Adamczyk, Jorgensen, and Ameri [15] [16] [17] [18]. In addition to those developed computational models, turbulence models have been developed and refined ranging from simple algebraic models to computationally expensive large eddy simulation. Theoretical work has even been done in to direct numerical simulation, whereby the Navier-Stokes equations are solved numerically without any turbulence model at all. In ADPAC, the choices for turbulence models include a modified coefficient version of the algebraic Baldwin-Lomax [19] [20] turbulence model, a slightly modified simple one-equation Spalart-Allmaras turbulence model [21], and an advanced two-equation \( \kappa - \varepsilon \) model developed by the efforts of Baldwin and Barth [22], and Goldberg [23].

2.4.1 Stator Seals

The modeling of the stator land seal area of gas turbines is a somewhat recent step in creating a full engine computational model. Experimental studies of the area are difficult to design, especially in high-speed compressors, as the area inside the seal is so
small. At this point, no experimental data for a complex geometry high-speed axial compressor seal cavity is available. Some experimental work has been done with low-speed compressors such as the NASA Lewis Low Speed Axial Compressor, and is collected in the Ph.D. dissertation by Wellborn [24].

Another paper by Wellborn et. al. that involved experimental work looked at a case study where the prediction of compressor performance, stage matching, and primary flow path properties of a 12-stage compressor was dependent upon the accurate modeling of the hub leakage flows [25]. The compressor was a high-speed machine that included modern design features such as variable stator geometry for the first five stages and inter-stage bleed flow extraction. All stators were shrouded with multiple knife seals and had abradable material on the inner surface to reduce the volume of flow through the cavity while still allowing for some degree of seal run-in. A compressor rig was built and tested for only a short time to allow for run-in and mechanical checks. Because the initial run was short, only a limited amount of aerodynamic data was collected for that run. The compressor was then disassembled and reassembled, unfortunately without measuring or replacing the seal abradable material due to time constraints.

When build 2 was brought up for testing, three significant things were noticed. First, the flow dropped by 2.2 percent at the operating line and efficiency was off by over one percent. Second, the stages of the compressor were matched differently between the two builds in terms of the pressure rise and loading across each stage. Third, the temperatures at the front stages were considerably higher for the second build than the first. All of these changes took place with the same exact rotor and stator blading
between build 1 and build 2.

The differences between build 1 and build 2 gave the compressor designers a reason for concern. In theory with the same blading on two builds, the compressor performance should be identical between each. A computational study of the compressor was undertaken to provide some insight into what could be causing the differences between the builds. From the simulations, it was determined that the seal-tooth clearances had increased outside of nominal values between builds, and that increase had caused all of the detrimental changes between the builds. This showed the necessity of modeling the stator seal flow during the design phase, and the large effect of leakage flows in general on overall compressor performance.

In Wellborn's work, a number of geometric and aerodynamic parameters of the labyrinth seals were investigated to determine the strength of their influence upon the performance of the seals as a whole. The number of knives and the seal gap size were the only geometric factors determined to have a strong influence on the seal performance, while the pressure ratio was the only aerodynamic factor determined to have a strong influence. The knife angle, tip thickness, tip sharpness, pitch, land surface roughness and land porosity were all geometric factors determined to have a moderate influence on the seal performance. The axial Reynolds number and knife tip speed were moderate aerodynamic performance parameters. The knife taper and knife height were both determined to have a weak influence on the seal performance, and the rotational Reynolds number and Taylor number were also determined to have a weak influence. This paper also presented a simplified model of stator seal cavity flows for axial flow
compressors and showed that by linking the simple model with a verified primary flow solver, it was possible to predict the overall effect of stator seal-tooth clearances.

A paper by Heidegger et. al. reported on the creation of a parameterized model of the stator seal cavity to isolate the relationship of certain stator cavity geometric parameters to overall compressor performance [26]. From an evaluation of the relationships, design guidelines to reduce the negative impact of the stator seal flow on stator performance were created. The parameters investigated for the study included the seal tooth gap, wheel speed, seal cavity depth, radial mismatch of hub flow path, axial trench gap, hub corner treatment, and stator land edge treatment. For each case, the seal cavity leakage flow, the losses in the cavity due to windage heating, the tangential seal cavity velocity, and the stator total pressure loss were all calculated.

The investigation showed that for many of the geometric parameters tested, there was no significant deviation from the baseline test, and therefore had very little overall effect on the compressor. The seal tooth gap was the parameter that had the most effect on the measured values. One of the most significant findings from this paper was the overall structure of the seal flow through the cavity. It was discovered that in each seal-tooth trench there was a rotating “driven” quality to the flow. This discovery helped to explain why it was that the seal gap and the number of knives were the most important features to the seal, and also to help design seals that took advantage of this phenomenon. Heidegger et. al. were among the first to report on this significant flow field finding.

Computationally, Heidegger et. al. included a mesh refinement study as part of their research. They modeled the cavity with four levels of mesh density. Level 1 had
11,421 points, level 2 had 78,897 points, level 3 was the baseline with 584,991 points, and level 4 had 1,175,985 points. This was to determine at what refinement level grid independence could be obtained. It was discovered that as the mesh was refined, the predicted mass flow through the cavity increased. This was determined to be from the increased resolution of the boundary layers as the grid spacing became smaller in those areas. Level 1 was determined to be too coarse to pick out the details of the flow, and level 2 began to pick up some of those details. For levels 3 and 4, the mass flow and radial profiles of axial and tangential velocity, deviation, turning angle, diffusion factor, and loss coefficient showed good agreement. This showed the authors that their selected baseline at level 3 was sufficient to adequately resolve the flow features and keep the computational resources necessary for the study to a minimum.

A report prepared for NASA by Heidegger et. al. [27] presented many of the same results as the paper described above [26] with much more depth and also provided additional data and results. The work included a high-speed compressor seal cavity study and a seal cavity parameterized study, including a detailed rotor-stator-rotor interaction with a seal cavity.

A paper by Young and Snowsill investigated a different type of secondary flow through an axial compressor, the bleed flow from various compressor stages [28]. A computational study was conducted where bleed flow was taken from taps of circular cross section at the hub of the rotor. It is common in gas turbine engines to bleed air from the compressor in this manner for use in cooling the turbine. This presented a problem because as the flow entered the bleed port, swirl was imparted which caused a
high pressure loss. Just as with flow through the stator seals, the secondary airflow through the bleed and cooling systems provides no contribution to engine thrust and thus there is incentive to minimize the volume and make the best use of it possible. Young and Snowsill investigated alternate shapes of the bleed taps as a means of controlling the swirl velocity in the cavity and reducing the adverse pressure gradient and vortices in the main flow path.

CFD was used as the primary investigative tool, with the ICEM CFD pre-processor used as the mesh generation package and Fluent the solver. In each of the two alternate bleed tap shape designs tested, the total area of the taps was conserved. The new designs were slots with semi-circular edges as opposed to the circular holes in the initial design. One slot design had 2 times the area of the original circular tap while using half as many, and the other had 4 times the area of the original tap while using a quarter the number. Through CFD simulation, it was determined that the double area slot reduced the inlet swirl coefficient by 4% while the quadruple area slot reduced the inlet swirl coefficient by 26%. This reduction in inlet swirl coefficient also reduced the cavity pressure difference by 4% and 37%, respectively.

These prior works show much of the effort that has been put towards understanding the secondary flow in the seal areas of an axial compressor. Geometries for stator seals have included single-, double-, and triple-knife designs, and experimental analysis by Wellborn [25] shows the extreme significance that the flow into and out of these seals can have on the overall performance of an axial compressor. It is also important to understand the structure of the flow through the seals, which through the use
of CFD techniques by Heidegger et. al. has been shown to be that of a “driven cavity”. It has also been shown that the flow moves from the opening at the trailing edge of the stator through the seal cavity to the leading edge, due to the adverse pressure gradient. These works have just begun to scratch the surface of the extremely complex flow in the stator seal cavity. As compressor designs become more aggressive in the years to come, understanding of the seal cavity flow will be an important part of the prediction of the overall performance of the compressor.

2.4.2 Compressor Rotor Casing Treatments

Another area involving secondary flow in an axial turbomachine is at the outer casing around the rotors. This rotor endwall region is one of the most important and yet least understood parts of the compressor. The clearance vortex is found in this area, which leads to blockage and compressor inefficiency and also forces the surge line of the engine closer to the design or operating point. This reduces the effective mass flow operating range of the compressor. It was discovered essentially by accident that certain casing treatments could widen the operating range of the compressor with only a small penalty in efficiency and performance. The discovery came around 1970 on a transonic fan being tested at NASA Lewis [29]. Once the researchers (Moore, Kovich, and Blade) discovered the beneficial effect of a casing treatment, blade-angled slots, honeycomb sections, axial skewed slots, and circumferential grooves were all tested as potential casing treatments. Research showed a counterintuitive fact that a treatment can be equally effective for both high inlet Mach number machines and low inlet Mach number machines. Treatments have also been used as a diagnostic tool to determine the primary
stall mechanism in different rotor setups.

Not all rotor casing treatments are effective, however, and some can even worsen the stall margin of the flow. Casing treatments have been found to be most effective in areas where the local blockage and flow deviation are in a state of rapid increase such as when the blade tips are stalling. If the blade itself is what is causing the stall, then a casing treatment is unlikely to provide any benefit.

A number of experimental studies have been undertaken in order to describe the flow structure in casing treatments and understand the mechanism by which casing treatments effectively delay stall and increase the mass flow range of compressors [30], [31], [32]. With the nature of the flow somewhat understood from experimental data, it was predicted that stall margin improvement was due to flow injection from the treatment cavities and flow suction into the treatment cavities. While these experiments have accurately shown many flow features of casing treatment, they have not given full understanding into the exact mechanism of the stall margin improvement.

One of the first attempts at using Computational Fluid Dynamics to understand the mechanism behind the delay in stall inception was performed by Hall et. al. in 1994 [12]. Other computational attempts before this generally specified the flow through the treatment region based upon experimental data. Hall however attempted to simultaneously compute the flow through the blade passage coupled to the treatment region for several different types of rotor casing treatment.

Hall used ADPAC to solve the flow with a multiple-block mesh for both the airfoil blade passage and the casing treatment flow, with several different techniques to
couple the passage and treatment areas. For the circumferential groove mesh, a direct-coupled approach was used, meaning that there was a direct, point-to-point correlation and passage of data from one mesh block to another. This coupling was attractive because the grooves and passage could be modeled as contiguous mesh systems.

The two studies chosen were NASA Rotor 5, a high speed, high pressure ratio fan with experimental data available, and the Allison AE3007 fan rotor for which no casing treatment experimental data was available. In the NASA Rotor 5 study, ADPAC accurately predicted the increase in total temperature of the flow, which is an indicator of the energy input, but not the total pressure, which is representative of rotor efficiency. In the Allison AE3007 study, it was decided that circumferential grooves provided the best trade-off of enhanced stall margin and decreased efficiency, although axial slots provided the best stall margin. The authors concluded that additional developments in boundary conditions were necessary to accurately analyze the flow through the treatment area.

An update of the Hall paper in 1996 [13] added some understanding into the flow in the treatments by including flow inlet distortion as a variable. A time-dependent analysis of the NASA Rotor 5 showed an intermittent, time-dependent quality of the recirculating flow pattern in the treatment. It also showed that the recirculation injected high-energy fluid at the leading edge of the rotor tip, increasing near-tip loading and flow incidence. At the trailing edge, low-momentum fluid was drawn off into the casing treatment. This removal/injection recirculation process was compared to a “booster stage” that added energy to the flow near the rotor tip.

A recent method applying the removal/injection theory can be found in Suder et.
al. [6]. The Discrete Tip Injection method devised for the experiments extracted bleed air from the rear of the compressor and recirculated it to an earlier stage of the machine. Twelve injectors were placed at 30 degree increments around the rotor casing, just upstream of the blade. The research reached several useful conclusions. Tip injection increased the mass flow range and also proved useful as a method to recover a compressor from a fully developed stall operation. The authors surmised that the injection decreased the incidence and blade loading at the tip, which allowed increased loading along the rest of the blade span. The maximum range extension occurred when the injectors were choked, and injector effectiveness was maximized by reducing the number of injectors to increase the injector exit velocity. This suggested that recirculating casing treatments may be more effective if the air were injected through a smaller area port, as opposed to the full annulus of the compressor. The drawback of Discrete Tip Injection is that by taking flow from the bleed air, a cycle penalty is incurred because work has already been done on that flow.

Casing treatments have also proven effective in centrifugal compressors and pumps [7],[8]. So called J-grooves running parallel to the pressure gradient on the casing wall can be seen as similar to casing treatments in axial compressors. In the paper by Saha, Kurokawa, Matsui, and Imamura [7], it was shown that a J-groove is an effective method of suppressing instability in a centrifugal compressor and increasing its flow range. It was shown that the grooves aid in reducing swirl in the inlet throat of the compressor and reduces reverse flow in the critical operating range. They also determined that the optimal groove was shallow and wide and ran parallel to the overall
pressure gradient. In Saha, Kurokawa, Matsui, and Imamura’s follow-up work [8], it was shown that rotating stall can be suppressed with a J-groove. The groove effectively reduced the peak-to-peak pressure variation on a vaned diffuser by 50%.

These papers show the effectiveness of compressor casing treatments in a variety of settings, from subsonic axial and centrifugal compressors to transonic and supersonic fans. They also show that there is still a great deal of work to be done to gain understanding into what exactly the mechanism of improvement is. Numerical techniques are perhaps the best way to gain this understanding, as they are far less expensive in terms of both money and time than experimental tests. As an added benefit, it is easier to perform a complete flow field analysis computationally than it is to attempt an experimental investigation in the small areas that make up these treatments.
Chapter 3. THEORY

This chapter will detail the development of the mathematical basis behind computational fluid flow. It will give both a general overview of methods and models traditionally used for computational fluid dynamics, and also a more specific discussion of methods and models used in solving flow through turbomachinery.

3.1 Governing Equations

The basic principles of conservation of mass, momentum, and energy provide the foundation to a study of computational fluids. These governing equations lead to the Navier Stokes equations, which can be discretized into finite difference or finite volume formulations. The governing equations of computational fluid flow are, in their simplest form, mathematical representations of the conservation laws of mass, and momentum, and the first law of thermodynamics. With these equations, we are able to describe the motion of a fluid in three dimensions.

Mass conservation can be represented in words by stating that the time rate of change of mass in any given fluid element equals the net rate of mass flow into that fluid element. Mass conservation is also known as continuity. In vector notation, the mass conservation or continuity equation is represented as the following:
\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{V}) = 0
\]  

(1)

The momentum equation is essentially Newton’s second law. It can be represented in words as the time rate of change of momentum for any given fluid element equals the sum for forces on that fluid particle. Generally, there are two types of forces on fluid particles that can cause a change in momentum. The first is surface forces such as pressure and viscous forces, and the second is body forces such as gravity and magnetic forces. Each force must be accounted for in the momentum equation. Pressure is a normal stress denoted by \( p \) and viscous stresses are normal stresses denoted by \( \tau \).

The conservation of momentum equation in three dimensions can be represented as the following, with \( k_B \) being the sum of all body forces:

\[
\rho \frac{D u}{D t} = \frac{\partial (-p + \tau_{xx})}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + k_B \tag{2a}
\]

\[
\rho \frac{D v}{D t} = \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial (-p + \tau_{yy})}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + k_B \tag{2b}
\]

\[
\rho \frac{D w}{D t} = \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial (-p + \tau_{zz})}{\partial z} + k_B \tag{2c}
\]

Finally, the energy equation is derived from the first law of thermodynamics. It states that the time rate of change of energy for any given fluid element equals the rate of
heat added to the fluid element plus the rate of work done on the fluid element. The rate of change of energy is given as $\rho \frac{DE}{Dt}$. The rate of heat added to the fluid element is due to heat conduction, and the rate of work is due to surface forces. Those equations are as follows:

Rate of heat addition to a particle due to heat conduction:

$$-\nabla \cdot q = \nabla \cdot (kVT)$$  \hspace{1cm} (3)

Rate of work done on a particle due to surface stresses:

$$\left[-\nabla \cdot (p\vec{V})\right] + \left[\frac{\partial (uT_{xx})}{\partial x} + \frac{\partial (uT_{yx})}{\partial y} + \frac{\partial (uT_{zx})}{\partial z} + \frac{\partial (vT_{xy})}{\partial x} + \frac{\partial (vT_{yx})}{\partial y} + \frac{\partial (vT_{zy})}{\partial z} + \frac{\partial (wT_{xz})}{\partial x} + \frac{\partial (wT_{yz})}{\partial y} + \frac{\partial (wT_{zz})}{\partial z}\right]$$  \hspace{1cm} (4)

Fully derived Energy equation:

$$\rho \frac{DE}{Dt} = -\nabla \cdot (p\vec{V}) + \left[\frac{\partial (uT_{xx})}{\partial x} + \frac{\partial (uT_{yx})}{\partial y} + \frac{\partial (uT_{zx})}{\partial z} + \frac{\partial (vT_{xy})}{\partial x} + \frac{\partial (vT_{yx})}{\partial y} \right.$$  
$$+ \frac{\partial (vT_{zy})}{\partial z} + \frac{\partial (wT_{xz})}{\partial x} + \frac{\partial (wT_{yz})}{\partial y} + \frac{\partial (wT_{zz})}{\partial z}\left] + \nabla \cdot (kVT)$$  \hspace{1cm} (5)

For compressible flows, the relationship between density, pressure, temperature, and internal energy can be shown via equations of state. This allows the equations to be
linked as a system of five partial differential equations. For simple incompressible flow there is no variation in density throughout the flow, and therefore it can be solved with only the mass conservation and momentum equations.

The Navier-Stokes equations, so named for the two scientists who independently derived them, are a special form of the momentum equation. In this form, the viscous forces in the momentum equation are related to the fluid properties dynamic viscosity, $\mu$, and bulk viscosity, $\lambda$. Substitution of these into the momentum equation leads to the Navier-Stokes equation:

$$\rho \frac{Du}{Dt} = -\frac{\partial p}{\partial x} + \frac{\partial}{\partial x} \left[ 2 \mu \frac{\partial u}{\partial x} + \lambda \nabla \cdot \vec{V} \right] + \frac{\partial}{\partial y} \left[ \mu \left( \frac{\partial u}{\partial y} + \frac{\partial \nu}{\partial x} \right) \right] + \frac{\partial}{\partial z} \left[ \mu \left( \frac{\partial u}{\partial z} + \frac{\partial \nu}{\partial x} \right) \right] + k_{bx} \quad (6a)$$

$$\rho \frac{Dv}{Dt} = -\frac{\partial p}{\partial y} + \frac{\partial}{\partial x} \left[ \mu \left( \frac{\partial u}{\partial y} + \frac{\partial \nu}{\partial x} \right) \right] + \frac{\partial}{\partial y} \left[ 2 \mu \frac{\partial v}{\partial y} + \lambda \nabla \cdot \vec{V} \right] + \frac{\partial}{\partial z} \left[ \mu \left( \frac{\partial v}{\partial z} + \frac{\partial \nu}{\partial y} \right) \right] + k_{by} \quad (6b)$$

$$\rho \frac{Dw}{Dt} = -\frac{\partial p}{\partial z} + \frac{\partial}{\partial x} \left[ \mu \left( \frac{\partial u}{\partial z} + \frac{\partial \nu}{\partial x} \right) \right] + \frac{\partial}{\partial y} \left[ \mu \left( \frac{\partial v}{\partial z} + \frac{\partial \nu}{\partial y} \right) \right] + \frac{\partial}{\partial z} \left[ 2 \mu \frac{\partial \nu}{\partial z} + \lambda \nabla \cdot \vec{V} \right] + k_{bz} \quad (6c)$$

With the Navier-Stokes equations, conservation of mass, conservation of energy, and the equations of state, we have a system of seven equations with seven unknowns.
However, these equations alone cannot be solved for flows of any engineering significance due to the turbulence inherent in real fluid flows and the computing power it would take to solve it. Instead, the assumption is made that the random nature of instantaneous variables in a turbulent flow can be represented analytically as a steady mean value with a fluctuating component added to it. Numerically, this is accomplished by time-averaging the properties of the flow, deriving what is known as the Reynolds-averaged form of the Navier-Stokes equations. Any flow variable that can be measured as an instantaneous quantity $f(x,t)$ such as a velocity component or pressure can be decomposed into steady time averaged and fluctuating components as

$$f(x,t) = \bar{f}(x) + f'(x,t)$$

(7)

where the time average $\bar{f}(x)$ is defined as

$$\bar{f}(x) = \lim_{T \to \infty} \frac{1}{T} \int_{-T}^{T} f'(x,t) \, dt$$

(8)

The time average is by definition zero. More detailed information on the governing equations can be found in Versteeg and Malalasekera [14].

### 3.2 ADPAC CFD Code

The numerical solution of the physical flowfield in the current compressor rotor casing treatment research was performed with the available code ADPAC v1.0 (Advanced Ducted Propfan Analysis Codes – Version 1.0). The code was developed by Allison Engine Company of Indianapolis, Indiana under NASA sponsorship. A brief
description of the code is given in this section. For the detailed manual of the code, refer to Hall et. al. [33].

ADPAC solves a discretized form of the Navier-Stokes equations and has the ability to perform a multiple-block grid discretization, allowing for coupled 2-D and 3-D mesh blocks and a wide range of geometries. It was specifically developed for the aerodynamic and/or heat transfer analysis of modern turbomachinery configurations using a time-marching Euler/Navier-Stokes methodology. The code has both steady state and transient flow analysis and prediction capabilities. It is also written to allow either serial or parallel execution on either shared memory parallel computers or distributed memory clusters of workstations from a single source.

ADPAC has been thoroughly tested and the results verified for both turbomachinery and non-turbomachinery applications. Despite the fact that ADPAC was designed and developed to analyze steady and unsteady aerodynamics of high-bypass ratio, multiple blade-row ducted fans, it is capable of computing a wide array of other complicated flow configurations such as cascades, casing treatments, or seals as well.

3.2.1 ADPAC Solution Process

To perform any analysis using the ADPAC code, there are a number of steps that must be successfully performed:

Step 1 is to define the problem. This involves selecting the geometry and flow conditions and defining what results are necessary from the analysis. This includes steady-state versus transient analysis, inviscid versus viscous
calculations, and an idea of the solution accuracy versus CPU time required. For the current research, a steady-state viscous calculation was determined to be necessary.

**Step 2** is to define the geometry and flow domain. Airfoils and endwalls are the usual concern in this step, however for the current research the seal and leakage flows were significant, along with the casing treatments. The flow domain is normally chosen such that the area of interest is significantly far enough away from the external boundary conditions that it is not incorrectly influenced by those conditions.

**Step 3** is to generate the numerical grid. ADPAC requires a structured grid that can be created in any number of commercial grid creation packages such as GRIDGEN. For this analysis, a variant of the NASA TIGG-3D code was used for grid generation of the compressor passages and casing treatment.

**Step 4** is to generate a standard input file. This file is custom-tailored to each run of the ADPAC code. The input file defines the number of iterations, number of multi-grid levels, damping parameters, and input/output control during code execution.

**Step 5** is to generate a boundary data file. When used in ADPAC this is referred to as the *BOUNDATA* file. This file is custom-tailored to each computational mesh in a given calculation. This file defines boundary conditions and block patching parameters. Steps 3, 4, and 5 are generally the most difficult and time-consuming parts of the process. Section 2.3 will cover the standard input and
boundary data files and their creation in more detail.

**Step 6** is to run ADPAC to solve for the necessary aerodynamic parameters. It is common for the code to be run several times before the final result is produced. Each time it is restarted a restart file is used to update the inlet conditions to the proper values for the new beginning time step. As the code is run, it creates a convergence history file which is useful to determine if the solution is converging.

**Step 7** is to plot and post-process the results. ADPAC is bundled with a program called ADSPIN which mass-averages flow variables to simplify their interpretation. Data files are also provided in the PLOT3D format. PLOT3D was developed at NASA Ames Research Center. For more information see the PLOT3D user's manual [34].

A method used in ADPAC to promote convergence acceleration is known as the multi-grid technique. Multi-grid is a numerical solution technique that automatically coarsens the mesh and computes corrections to the iterative solution on that coarser mesh, propagating the changes to the original fine mesh by interpolation. This may be done recursively for several coarsening levels of the original mesh. Coarse meshes are created by eliminating every other mesh line in each coordinate direction from the preceding finer mesh. Because of this, meshes must be carefully created such that the mesh size is 1 greater than any number which can be divided by 2 several times and remain a whole number (e.g. 9, 17, 33, 65). The following equation ensures that the mesh index size chosen for a mesh block will be multi-grid capable:
\[ \frac{n - 1}{2^{(mg-1)}} = \text{integer} \] (9)

where \( n \) is the block or boundary condition edge and \( mg \) is the desired number of multi-grid levels. It is common to use 3 levels of multi-grid whenever possible to promote convergence acceleration. The number of multi-grid levels is specified in the ADPAC Input File. If only 1 level of multi-grid is specified in the input file, this is the equivalent of starting the solution with no levels of multi-grid, using only the fine mesh.

A so-called “full” multi-grid startup initiates the calculation on the coarsest specified mesh, performs several time-marching iterations on that mesh, and then interpolates the results at that point in time to the next finer mesh. This is repeated until the solution at the finest mesh level is converged. This approach allows a reasonable solution to be attained using a coarse mesh to conserve CPU time. Then with this approximate solution interpolated and mapped to a finer mesh, the startup is closer to the final solution, requiring fewer time-marching iterations on the fine mesh and giving a stronger tendency for the solution to converge.
Figure 5: Wireframe Image of all Solid Surfaces in Stator-Rotor-Stator Stage; With Circumferential
Groove Casing Treatment Above Rotor
3.2.2 ADPAC Procedure of Current Work

Numerical investigation of the compressor rotor casing treatment was performed using geometry based on a research compressor rig. The rig was designed to be a subsonic compressor used mostly for aerodynamic research. It has not yet been run with a casing treatment at this point in time, therefore no data is available. The configuration for the computational results was a stator-rotor-stator setup with a discretized tip.
clearance region of the rotor included in the model. Discretizing the tip clearance has been shown to greatly improve the accuracy of the model, especially in the endwall region. Because this was region was essential to the study, the extra computational effort was well worth the level of accuracy it provided. Figure 5 and Figure 6 show the Rotor-Stator-Rotor geometry with the circumferential grooves above the rotor. The areas in red (the rotor blade and hub) are the rotating components, while the areas in pink (the stator blades and hub and all of the casing) are

![Image of geometry with grid labels](image)

*Figure 7: Zoomed in View of Rotor Endwall Casing Treatment and Rotor Blade Tip Gap*
stationary. Figure 7 shows the endwall region of the rotor blade, looking circumferentially around one groove of the casing treatment. Again, the rotating blade is in red while the case and treatment are in pink. The black area between the blade and case is the blade tip gap, also known as blade clearance.

Figure 8: View From Above Rotor Passage: Blade Airfoil Profile in White, Case Mesh and Groove Mesh Visible in Pink
The mesh was a pure-H mesh, which allowed the circumferential grooves of the casing treatment to be modeled with a direct correlation between the main rotor flowpath and each individual groove. An explanation of the H-mesh is shown in Figure 8. The image is of the rotor passage mesh, taken from the top looking down. The blade profile is shown in white with the passage mesh. The areas where the mesh is tighter are where
the grooves are situated. Figure 9 shows the direct point-to-point correlation of the passage mesh to the groove mesh, with the groove in green and the main flowpath in blue. The grooves were 20 percent of the axial chord of the rotor blade in depth and 10 percent of the axial chord in width, with a spacing between each groove of 5 percent of axial chord. Notice that at the endwall of the passage the mesh spacing becomes smaller. This higher resolution aids in the calculation of the boundary layer in the passage.

Figure 10: Rotor Hub, Blade, Endwall, and Casing Treatment Mesh with Stator Passages Outlined on Either Side
Figure 10 shows a slice of the rotor mesh with circumferential grooves across the top. Each blade passage, tip clearance, and casing treatment groove was a separate mesh block, numerically joined in ADPAC with its system of patches.

Figure 11: Full Passage Mesh; Rotating Components in Red; Non-Rotating Components in Pink
The grid size was chosen based on earlier grid refinement studies of similar geometry. The inlet and outlet of each passage was 53x45, axial by circumferential. For the stator 1 and stator 2 passages, the grid was 101 points axially (101x53x45). For the rotor passage the grid was 177 points axially (177x53x45). For the discretized tip clearance region over the rotor, the grid was 137x5x13. This means that the blade itself was 137 grid points axially and the top 5 radial points of the rotor passage were in the tip clearance region. For the circumferential grooves above the rotor, a 45x21x17 mesh was created, and above the clearance region a 13x21x17 mesh was created. The total number of mesh blocks was 14, and each block met the criteria of the multi-grid concept for a full level 3 multi-grid as discussed in the previous section. Figure 11 shows the full passage mesh with rotating components in red and stationary in pink. This grid size was demonstrated in the earlier grid refinement study to be sufficient for computation and gave a reasonable level of numerical stability and accuracy.

The input and boundary data files are customized for every run of the code. A sample case.input file is given in Appendix A1 and a sample boundary data file is given in Appendix A2. The files are also described in detail in the ADPAC Users guide, [33]. The most commonly changed parameters in the input file for stability enhancement were the value of the time-step multiplier in the time-marching routine (CFL), an explicit time-step multiplier (CFMAX), and the initial flow Mach number (RMACH). In the boundata file, the most commonly changed parameter was the stage inlet profile conditions and stage pressure rise.
3.3 Stator Seals and Fluent Background

The computational study on compressor stator seals was performed with the popular commercial CFD code Fluent. Fluent has been widely tested, distributed, and marketed for fluid flow solutions of all types. The solver is generic with a myriad of options for problem definition and setup, boundary conditions, discretization techniques, solution techniques, and turbulence models. Available add-ins include combustion modeling, flows through porous media, and other computational models for special-case fluid flow problems. For these reasons along with its relative ease of use and simple-to-understand interface, Fluent seems to be becoming an industry standard across the field of computational fluid dynamics.

Fluent is packaged with its own proprietary grid generator known as Gambit. Gambit has the ability to import various solid geometry files for mesh creation, and also has the ability to do some geometry creation within the program itself. It can create 2-D and 3-D structured or unstructured meshes with a variety of cell shapes and good mesh refinement options. Gambit was used to create the stator seal 2-D meshes for this computational study.

Fluent also has an option of using its own post-processor, which is also very good. It has the ability to show visualizations of transient and steady state images. Available visualization options include contour maps, vector maps, surface plots, particle traces, and transient movies. The Fluent post-processor was used for all visualizations in the stator seal study.
3.3.1 Fluent Procedure of Current Work

As discussed earlier, the purpose of stator seals in a compressor is to provide clearance between the spinning rotor hub and the stationary stator blades, while keeping the leakage flow through this passage to a minimum. In this study, two different stator seal geometries were selected for analysis, with one component of each varied to determine its effect on the flow in the cavity. Figure 12 and Figure 14 show examples of the two stator seal geometries.

The first geometry, Stator 1, has only one narrow flow inlet and one outlet with three individual cavities divided by two seals. In this geometry, the first seal gap is varied to allow greater or smaller amounts of mass flow through. A seal such as this would be seen in the front stages or inlet guide vanes of a typical high-speed, high pressure ratio axial compressor. The second geometry, Stator 2, represents a stator at one of the back stages of an axial compressor. The seal cavity has an extremely narrow and long inlet area and narrow exit with one large cavity area. This geometry also has a second inlet that allows flow into the cavity from outside the main flow area. The Stator 2 cavity is also unique because at the back of the compressor the majority of the walls are stationary, whereas with other seal passages the outer wall is rotating. In this setup, the mass flow in from the second inlet is the varied parameter.

The geometry for these simulations was obtained from an IGES file of a meridional slice of the flow path. The flow path was that of an advanced high-speed two-stage fan with a high pressure ratio across each stage. The IGES files were imported into the Gambit mesh generation package where any geometry unnecessary to the seal study
was removed and edges and vertices were cleaned. This involved hand selecting significant portions of the geometry file and many times manually connecting the edges. This allowed for a completely closed surface, which is necessary for building the mesh. A slice of the flow path was left at the inlet and exit of the passage to simulate the flow exiting and re-entering the main area. A high-density unstructured mesh consisting of approximately 168,500 nodes was applied to the 2d area, with a high density of points around the seals, rotating walls, and at the inlet and exit of the cavity. The inlet and exit boundary conditions at the stator were taken from a previous analysis without a stator seal model. The mesh was then exported for use with the Fluent solver.

In Fluent, there are many options for solution techniques. For the purposes of this investigation, a 2d steady segregated axisymmetric solver incorporating swirl was chosen for the calculations. This allowed the rotation of the shaft to be modeled and taken into account even in two dimensions. The energy equation was also solved to resolve compressibility effects, and the K-E turbulence model incorporating viscous heating was applied to resolve the turbulence parameters. Pressure inlets and outlets were used at the main flow path, and in the analysis of the second stator, the second inlet was set as a mass flow inlet. All inlet and exit conditions were set from prior CFD data of the flow path.
Measurement stations were defined through both seal cavities in the mesh to track key flow parameters. In particular, total and static pressures, flow velocities, and total temperatures were significant to this study, along with the mass flow rate. Tracking the pressures provided a good indicator of losses through the cavity. Tracking the velocities and total temperatures provided a good means of determining the energy in the flow and efficiency as it passed through the cavity. The mass flow gives the best indication of the impact on the overall compression stage as all the flow through the seal cavity is subjected to losses and incurs a penalty on compressor efficiency and performance.
For stator one, three seal tooth clearance heights were analyzed, a baseline and plus or minus 50% of that baseline. The baseline was at 0.010", with cases at 0.005" and 0.015". This is a good approximation of the working range of the stator seals on many modern compressors. A seal gap of 0.010" may represent a nominal compressor, a gap of 0.005" may represent a compressor at the tight end of its tolerance band, and a gap of 0.015" may represent the loose end of the tolerance band. Seal gaps of greater than 0.015" may also occur if the compressor is old and has deteriorated over time or is damaged.

*Figure 13: Stator 1 Seal Tooth Gap Detail; Left: Baseline, 0.010"; Top Right: 0.005"; Bottom Right: 0.015"*
Mass flow was calculated at four points through the seal cavity for each of the three cases. These four points were Position 4, in the neck of the inlet, Position 3, the first knife gap (which was the varied gap), Position 2, the second knife gap (which was not varied), and Position 1, at the neck of the outlet. In addition to mass flow at each point, the area-averaged and mass-averaged total and static pressure, total temperature, and tangential velocity were all measured. Figure 12 shows the full cavity area including the main flow path sections (label it) and locations of each data measurement station,
while Figure 13 shows a detailed look at the mesh of the varied seal area at each of the three tooth clearance gap values.

For stator two, there was no actual knife seal separating the main inlet and outlet of the cavity. Unlike stator one, however, there was a second inlet in this cavity from the center of the compressor, and there was a knife seal at this location. The simulation was run with two values of mass flow at this inlet: 0.0 lbm/s (no flow was allowed through the seal), and 0.32 lbm/s. Figure 14 shows the full cavity area, including modeled portions of the main flow path and each data measurement station. Three measurement locations were chosen for recording values of mass flow, total temperature, total pressure, static pressure, and tangential velocity. The three points were as follows: Position 1, at the exit to the cavity; Position 2, at the inlet of the cavity; and Position 3, at the knife seal at the second cavity inlet.
Chapter 4. Computational Results and Discussion

This chapter details the results of two secondary flow related investigations. The first involves the flow through two different types of stator knife seal cavities, while the second involves a circumferential groove casing treatment at the rotor endwalls of a typical axial flow compressor.

4.1 Stator Seal Cavity Investigation

In an axial compressor, the stator seal cavity provides clearance between the rotating rotor shaft and the stationary stator geometry. This clearance allows mass flow to bypass the main flow path, and therefore is a source of leakage and potential losses in the compressor. The larger the mass flow through the seal as a percentage of the total mass flow, the larger the leakage and losses.

Because the pressure increases from the inlet of the stator to the exit, the flow in the seal cavity moves backwards with respect to the flow in the main passage. Therefore, the inlet of the seal cavity is considered to be at the exit of the stator vane, and the exit of the seal cavity is at the inlet to the stator vane. To minimize losses and inefficiency, it is important to understand the nature of the flow through each seal, and then to understand how the seal flow interacts with and alters the primary flow of the main flow path.
Because these cavities in an actual compressor geometry are physically difficult to instrument, experimental data is difficult to obtain. Computational Fluid Dynamics (CFD) provides an attractive method for the prediction of mass flow, pressure, temperature, velocity, and flow path throughout the cavity.

4.1.1 Stator One Geometry Results

Stator One represents a seal that could be found in the front of a high-speed, high pressure ratio axial compressor. Mass flow through the stator seal was measured and calculated as a percentage of the total mass flow through the main flow path. This flow quantity was calculated in a previous simulation involving the main flowpath but not the stator seal.

A plot of the mass flow through the varied seal is shown in Figure 15 and the percentage calculation is shown in Table 1. For the three values of seal clearance simulated, the mass flow ranged from 0.15 lbm/s to 0.47 lbm/s. This corresponds to between 0.17% and 0.5% of the total corrected flow through the stator.

As expected, when the seal tooth gap was increased, the mass flow through the cavity increased. For a 50% increase in gap size (from 0.010” to 0.015”), there was a corresponding 42% increase in mass flow. For a 50% reduction in gap size (from 0.010” to 0.005”), there was a corresponding 53% decrease in mass flow.
<table>
<thead>
<tr>
<th>Seal Cavity Gap</th>
<th>Cavity Mass Flow</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.005&quot;</td>
<td>0.17%</td>
</tr>
<tr>
<td>0.010&quot; (Baseline)</td>
<td>0.35%</td>
</tr>
<tr>
<td>0.015&quot;</td>
<td>0.50%</td>
</tr>
</tbody>
</table>

*Table 1: Stator 1 Seal Cavity Mass Flow as a Percentage of Total Corrected Stator Mass Flow*

*Figure 15: Mass Flow Through Stator 1 as a Function of Seal Tip Clearance*
In all three of the tip gap simulations, the inlet and exit conditions of the modeled portions of the main flow path were identical. From a previous computational study performed by Rolls-Royce Corporation, the conditions at the inlet of the stator were set as follows:

- Gage Total Pressure = 40.95 psi
- Static Pressure = 20.24 psi
- Total Temperature = 700.2 R

The inlet was set so that the flow was moving with the correct axial, radial, and tangential velocity magnitudes by setting each flow component individually. The conditions at the exit of the stator were set as follows:

- Gage Total Pressure = 36.95 psi
- Static Pressure = 28.56 psi
- Total Temperature = 700.2 R

Recall that because the flow in the seal cavity moves backwards with respect to the main flow path, the inlet of the seal cavity was actually the exit of the stator.

The computed values of area-averaged static and total pressure, total temperature, and tangential velocity are shown in Figure 17, Figure 18, Figure 19, and Figure 20. Looking at the Static Pressure at the inlet to the cavity, Position 4, it is approximately equivalent for the three knife gap values at 29 psi. As the flow is accelerated and drawn through the variable knife seal, Position 3, the static pressure dropped through all three of the seals, but by
Figure 16: Stator 1 Cavity Reference Locations

Figure 17: Stator 1 Static Pressure Throughout Cavity
Figure 18: Stator 1 Total Pressure Throughout Cavity

Figure 19: Stator 1 Total Temperature Throughout Cavity
more through the smaller seal gaps. Through the 0.015" seal the static pressure was reduced by 34%, whereas static pressure through the other two seals was reduced by 38%. Through the rest of the cavity to the exit, Position 2 to Position 1, static pressure remained fairly constant, and reentered the cavity with 6% difference between the low and high values.

The total pressure was slightly different from the static pressure. At the seal inlet, the total pressure of the smallest gap value was 5% higher that that of the largest gap. This was presumably due to some degree of back pressuring from the smaller seal. However, once the flow was accelerated through the variable seal, the total pressure of the 0.005" seal dropped below that of the two larger seals. The total pressure dropped
again going through the second seal at Position 2, and stayed fairly level up through the cavity exit. The total pressure dropped by 42% through the seals in the 0.005” case, 37% in the 0.010” case, and 32% through the seals in the 0.015” case. The assumption from these numbers was that the velocity magnitude through the seal cavity decreased as the seal gap was decreased.

The values of tangential velocity through the seal cavity provided some insight into the flow through the cavity. When the flow initially left the main flow path into the inlet of the seal area, the tangential velocity of the flow was 33% higher in the inlet of the 0.005” seal cavity than in the inlet of the 0.015” seal cavity. All were accelerated through the first seal, but the larger seal areas allowed greater tangential accelerations. As more flow passed through the seal cavity, the flow tangential velocity decreased to a value closer to the tangential velocity of the main flow path. This has some interesting implications for the flow when it is reintroduced to the inlet of the stator. The tangential velocity could be detrimental to blade performance due to added hub blockage and non-optimal incidence angles if the tangential velocity is at a poor value, or the flow exiting the cavity could potentially be tuned so that it helps ensure proper stage performance. This cannot be known for certain until a full analysis including the actual stator blade has been performed.

The total temperature through the cavity was interesting, and points out an important design consideration. The 0.010” and 0.015” seal areas had nearly identical total temperatures through the cavity. The 0.005” seal, however, had a significantly higher total temperature throughout. The total temperature rise was due entirely to work
done on the flow from the rotation of the shaft, since no other mechanism for work input existed. With less mass flow through the cavity, the temperature was allowed to increase. This implies that there is a seal gap value that will not allow enough mass flow through the seal area to properly cool the cavity. With less mass flow it would be easy to predict temperatures beyond what is considered safe for the metallurgical properties of the material.

Axisymmetrically averaged (no tangential component) velocity vectors colored by velocity magnitude are shown in Figure 21. From these images, it can be seen that the overall flow pattern of the cavity did not change a great deal with the change in gap clearance. What did change was the magnitude of the velocity through each seal. From the inlet, there was a counterclockwise rotation pattern with the flow being forced up the outer wall due to the rotation effects. After flowing through the seal there were two counterclockwise rotating “driven cavity” patterns until the next seal gap. From there the flow entered the exit side of the cavity where a clockwise rotation pattern was observed. This was once again a “driven cavity” structure with airflow forced up the rotating side from rotation effects.
4.1.2 Stator Two Seal Cavity Results

The design for Stator 2 has two mass flow inlets and one mass flow exit. Flow enters through the gap in the main cavity at the exit of the stator blade, and flow also enters from an opening at the centerline of the cavity as well. Mass flow was measured at the main inlet and outlet for both of the cases. The calculated mass flow is shown in Table 2.
When the secondary seal was allowing mass flow in to the cavity, the predicted flow in to the cavity from the main flow path was reduced by about 0.075 lbm/s, or 4.3%. Because continuity must be satisfied, the flow out of the cavity increased when flow was allowed in through the seal, even though the main inlet mass flow was reduced. The total flow in and out of the cavity increased by approximately 0.225 lbm/s, or 12.9% when the extra flow was introduced from the centerline seal. This increase in flow exiting the cavity with a small decrease in flow entering the cavity could have an undetermined effect on the stator's overall performance. More study of a model including the stator geometry is necessary to determine this potential detriment or benefit, however experience would lead one to believe that the losses would increase due to the generation of a larger passage vortex.

<table>
<thead>
<tr>
<th>Inlet 2 Flow</th>
<th>Main Inlet</th>
<th>Main Outlet</th>
<th>Inlet % of Total</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0 lbm/s</td>
<td>1.74 lbm/s</td>
<td>1.74 lbm/s</td>
<td>1.9%</td>
</tr>
<tr>
<td>0.32 lbm/s</td>
<td>1.67 lbm/s</td>
<td>1.97 lbm/s</td>
<td>1.8%</td>
</tr>
</tbody>
</table>

*Table 2: Stator 2 Cavity Mass Flow as a Percentage of Total Corrected Stator Mass Flow*

The inlet and exit conditions of the modeled portions of the main flow path were again identical. From a previous computational study performed by Rolls-Royce Corporation, the conditions at the inlet of the stator were set as follows:

- Gage Total Pressure = 84.88 psi
- Static Pressure = 44.70 psi
• Total Temperature = 890.0 R

The inlet was set so that the flow was moving with the correct axial, radial, and tangential velocity magnitudes by setting each flow component individually as defined in the previous Rolls-Royce study. The conditions at the exit of the stator were set as follows:

• Gage Total Pressure = 78.70 psi
• Static Pressure = 57.90 psi
• Total Temperature = 890.0 R

Recall that because the flow in the seal cavity moved backwards with respect to the main flow path, the exit side of the stator is actually the inlet side of the seal cavity.

The computed values of area-averaged static pressure, total pressure, tangential velocity, and total temperature are shown in Figure 23, Figure 24, Figure 25, and Figure 26. There is no major effect of allowing flow through the center seal on the flow values at the inlet and outlet of the cavity (Positions 2 and 1, respectively). Static pressure, total pressure, tangential velocity, and total temperature remain virtually unchanged at the inlet and outlet. When the second inlet allows 0.32 lbm/s to enter the cavity, this causes the static pressure at the inlet of the cavity to increase by 2%, the total pressure to increase by 0.4%, and the tangential velocity to increase by 0.9%. The total temperature at the inlet did not increase appreciably.

One problem was observed in the case with no mass flow through the center seal. This involved the predicted temperatures around the knife seal. Because there was no air flowing through that area, it allowed the viscous heating of the air to raise the
temperature to extremely high values in the seal area. In practice this should not be a concern, because air should still manage to leak through the seal, regardless of how tight its clearance may be. If there were actually very little or no air flowing through that area, a redesign of the cavity may be necessary to promote circulation there.

Axisymmetrically averaged (no tangential component) velocity vectors colored by velocity magnitude are shown in Figure 27. Allowing mass flow in from the center seal had an obvious effect on the overall predicted flow path inside the cavity. Both had a counterclockwise driven area from the flow passing from inlet to outlet, but the area is much smaller when the second inlet was allowing flow into the cavity. This was due to the interaction of the two inlet flows. As was mentioned previously, the flow in the seal area was very important to cooling in that area. With no flow coming in from the seal, there was very little circulation there and temperatures are allowed to reach very high levels.
Figure 22: Stator 2 Cavity Reference Locations

Figure 23: Stator 2 Static Pressure Throughout Cavity
Figure 24: Stator 2 Total Pressure Throughout Cavity

Figure 25: Stator 2 Tangential Velocity Throughout Cavity
Figure 26: Stator 2 Total Temperature Throughout Cavity
Figure 27: Axisymmetrically Averaged Velocity Vectors Through Stator 2 Seal Cavity; Left: 0.0 lbm/s Through Center Seal, With Detail. Right: 0.32 lbm/s Through Center Seal
4.1.3 Stator Seal Study Conclusions

This study investigated the flow through two types of stator seals using CFD analysis tools. The first stator seal geometry had one inlet and one outlet with a complicated flow path and several knife seals. The second geometry had two inlets and one outlet with a single undivided cavity. Several interesting observations were noted about the design of each of the two geometries.

In the first design from Figure 12, for every 0.005" reduction in knife seal gap, there was approximately a 50% reduction in mass flow through the seal. In the second design from Figure 14, when mass was allowed to flow in from the second inlet, the total outlet mass increased by 12.9%. The flow path in both designs exhibited driven cavity like results, with the flow forcing several tangential vortices in each cavity. Due to the pressure rise across the stator blade, flow entered both cavities downstream of the stator, and traveled backwards to reenter the main flow path upstream of the stator.

When less flow moves through the seal cavity there is less leakage, blockage, and therefore increased efficiency from the stator blade. There is a danger to consider with smaller clearances and less flow, however. The temperature in at least part of the cavity can rise to values that are considered unacceptable for the material. This is a trade off that must be considered in planning for lower leakage and higher efficiency.

A future project would be to model not only the stator seal cavity, but also to consider the stator blade. This could provide some important insight on the dynamics of the interaction of the leakage flow with the main flow and the stator. This could help in predicting features such as hub separation and altered incidence on the vane as a result of
the stator leakage flow. This will help compressor designers account for the effects of the leakage flow and more accurately model engine passages.
4.2 Compressor Rotor Casing Treatment Study

In the design of an axial compressor, the surge margin from the design/operation point is an extremely important consideration. Ample surge margin must be included to take into account degradation and wear on the compressor, unexpected operating points, inlet distortion, transient operation during throttle up and throttle down, and foreign object ingestion. For this reason, compressor designers are interested in methods that can enhance the overall stability and surge resistance of the machine, even if it may mean a trade off in another facet of the design. There are many methods for defining the surge margin of a compressor, but one of the simplest definitions assuming a constant corrected mass flow could be

\[
SM = \frac{(PR_{Surge} - PR_{Working})}{PR_{Working}}
\]  

(10)

Where \textit{PR} is the overall pressure rise at the surge line and at the working line of the compressor.

It is important to recognize that the working line of the compressor is somewhat arbitrary and defined by the designer. However, it is normal to require a surge margin of around 25% for a multistage compressor for use in a gas turbine engine. The surge margin could always be arbitrarily increased by simply lowering the working line of the compressor. This is not an attractive solution though, because lowering the working line also lowers the quantity most desired from any compressor, namely pressure rise. The best solution would maintain the design point of the compressor but move the surge line
up away from the operating line.

This best solution can be found from several methods; some of which the designer has control over and some he does not. These methods could include reducing the amount of inlet distortion allowed in to the compressor, forcing transient acceleration and deceleration to change very slowly to keep the transient from having any overshoot off of the working line, designing aerodynamically superior blades, reducing the tip clearance of the blades, or including some sort of rotor casing treatment. Each of these methods has a specific set of restrictions and limitations. A reduction in inlet distortion is generally a function of the engine installations on the airframe over which the compressor designer has no influence. Forcing the rate of transient acceleration and deceleration to be small is possible in industrial-use compressors, however for safety and performance reasons airframe manufacturers and operators require a much quicker spool up and down from the engines. Aerodynamically superior blades are good in theory, however most design are fairly well optimized for the aerodynamic and mechanical loads they must absorb. Likewise, tip clearances over the rotor blades are set as small as they can safely be for factors such as transient heating and cooling and case distortion. Many compressor rotor tracks even have an abradable coating applied to the case to allow for a small degree of rub between the blades and case. The rotor casing treatment has the drawback of reducing the compressor efficiency, but it is perhaps the most controllable and consistent option available to compressor designers.

As was discussed in Section 2.1.2, some research has been directed towards documenting the effectiveness of several types of compressor rotor casing treatments,
however little has been done to understand the exact mechanism and aerodynamics that provides an improvement in stall margin and stability. For the purpose of this study, an aerodynamic research compressor rig with available computational geometry was chosen. The rig did not have any type of casing treatment. To add the grooves to the computational H-mesh, a FORTRAN program called CircumGrooves was written. Pictures of the computational geometry and mesh along with descriptions of the procedure for working with ADPAC are given in Section 2.2.2.

### 4.2.1 Circumferential Grooves Results

The research rig analyzed was a simple stator-rotor-stator setup with an overall baseline design point pressure ratio of 1.10, 100% corrected speed of 5000 rpm, inlet corrected flow design intent of 20 lbm/s, and constant radius flowpath. It had 44 stator blades for both stator 1 and 2, and the rotor wheel had 36 blades. The flowpath was two inches high and the radius of the hub was 10 inches, making the tip speed of the rotor blade approximately 524 ft/s. This is a subsonic tip speed, therefore there is no passage shock in the design. For cases with the casing treatment, there were five grooves evenly spaced above the rotor blade. Each was 20% of the axial chord of the rotor blade in depth, and 10% of the axial chord in length. They were evenly spaced beginning at 15% of the axial chord downstream of the blade leading edge, and 5% axial chord apart from each other.

It was expected that when the casing treatment was added, the calculated values for both the pressure ratio and inlet corrected flow would drop a small amount due to the
decrease in compressor efficiency. Convergence plots of residuals, pressure ratio, mass flow rate, number of separated points, adiabatic efficiency, and number of supersonic points are given for the baseline untreated configuration in Figure 28 and the baseline treated configuration in Figure 29. The disconnects seen on the untreated convergence plots at 250 and 500 cycles represent the points at which the solver switched to a more refined level of multi-grid. The treated convergence plots show a close-up look at the final 500 iterations, after the solver had switched to the finest level of multi-grid. The plots show a reasonable level of convergence was attained in all areas after 1000 time steps.

Inspecting the values of each, it was seen that, as expected, the pressure ratio, mass flow rate, and efficiency were all reduced when the circumferential grooves were added to the compressor case. At the compressor design point, the pressure ratio was reduced by .07%, the mass flow rate was reduced by 1.21%, and the efficiency was reduced by .59%. Despite these values being reduced, the operating line was able to be extended to higher overall pressure ratios before the compressor was fully stalled or choked. This is shown graphically in Figure 30. The treated compressor was able to reach an overall pressure ratio of 1.1198 with an efficiency of 90.32% and a mass flow of 17.94 lbm/s. The untreated compressor was only able to give a pressure rise of 1.1153 with an efficiency of 91.2% and a mass flow of 19.30 lbm/s. The treated compressor was even able to go to a higher pressure of 1.1286, however at this point the efficiency had dropped off to 76% and the mass flow was reduced by over half to 10.87 lbm/s, meaning that the compressor was effectively choked, although still providing adequate pressure
Figure 28: Baseline Untreated Rotor Convergence Plots
Figure 29: Rotor Mesh With Casing Treatment Convergence Plots
rise. The treated compressor was able to reach higher up the working line before it surged. Figure 30 shows the traditional compressor overall performance plots of Efficiency versus Inlet Corrected Flow and Predicted Pressure Rise versus Inlet Corrected Flow. The treated case is shown in red and the untreated case is shown in blue.

To gain some degree of understanding into the mechanism that leads to better stability, flow visualization plots of the data, especially at the rotor endwall and casing treatment, were created using PLOT3D. Of particular interest were streamlines to see where the flow traveled and what sort of structure it had in the endwall and treatment regions. Pressure and velocity profiles along the rotor blade were also of interest. The most interesting cases were those highest along the working line, closest to surge.

The first task was to compare the amount and location of aerodynamic blockage in the rotor passage. Aerodynamic blockage is generally the result of low-momentum and low total pressure flow in endwall vortices and flow separation from the suction-surface of the rotor blade. The buildup of blockage in rotor and stator passages is thought to be one of the driving forces behind stall in axial turbomachines. Visualizations showing iso-surfaces of low-velocity flow in the passage proved to be an adequate method for viewing blockage, and are shown in Figure 31 for the untreated rotor and Figure 32 for the treated rotor. Unfortunately, in this test rig the starting point for blockage was found to be in the hub and not at the tip, meaning that the full benefit of a casing treatment may not have been realized in this compressor. This does not mean that the endwall results were any less relevant, rather it means that the full benefits of casing treatment in general were somewhat subdued in this particular axial compressor.
Figure 30: Efficiency and Pressure Ratio versus Compressor Inlet Corrected Flow
Figure 31: Contour Plot Showing Separation Bubble off Rotor Suction Surface, No Casing Treatment
Figure 32: Contour Plot Showing Separation Surface off Rotor Suction Surface, With Casing Treatment
Figure 33: Streamlines From the Rotor Tip Colored by Velocity Magnitude, No Casing Treatment
Figure 34: Streamlines From the Rotor Tip Colored by Velocity Magnitude, With Casing Treatment
Figure 35: Reverse Angle of Particle Traces Released From Rotor Tip Suction Side, No Casing Treatment
Figure 36: Reverse Angle of Particle Traces Released From Rotor Tip Suction Side, With Casing Treatment
Plots of streamlines of flow in the tip clearance endwall region colored by velocity magnitude for the untreated and treated cases are shown in Figure 33 and Figure 34, respectively. The view is from the front of the rotor passage looking back along the blade. The rotor is rotating to the right. The streamlines begin above the suction surface of the rotor blade and are relative to the blade's rotation, not absolute. Several things were immediately apparent from comparing these plots. The untreated rotor appeared to have much smoother flow, with the majority of the streamlines moving from one blade tip straight across to the other, and turning under the influence of the pressure side of the next blade. The vortex off the leading edge of the untreated rotor curved back through the passage. The flow from above the treated rotor had a very different pattern. Much of the flow moved up into the casing treatment grooves under the influence of the rotor pressure surface, and was then infected back into the flowpath beyond the rotor suction surface. This allowed the flow not drawn into the grooves to flow unrestricted back through the flowpath at a lower Mach number. Figure 35 and Figure 36 show the same streamlines viewed from the back of the flowpath. This gives a good representation of the highly complex three-dimensional nature of the flow. Also apparent from this view is the velocity difference at the trailing edge between the untreated and treated cases.

Figure 37 shows where flow enters and leaves the grooves. This gave an interesting look at some of the dynamics of the flow in the grooves. It was cyclic and wavelike, typically removing flow at the downstream half of the groove and re-injecting it at the upstream half of the groove. The majority of the flow in the groove was injected
by the pressure wake of the rotor, and the majority was returned to the flowpath midway between the two blades. It could be said that the rotor grooves acted as a sort of pressure relief to the tip of the rotor. As such, it would not allow a separation surface to form off the suction surface of the blade. Instead of flow at the blade tip curling around between the blade and the case and creating a passage vortex, the flow at the tip was injected up into the grooves where the momentum was reduced. The grooves allowed for a turbulent boundary layer at the endwall that appeared to the flow to be very thin, allowing more pressure rise capability before surge.

Figure 38 through Figure 42 show more streamlines of flow into and out of each individual groove. Figure 43 shows streamlines at the exit from the culmination of all five grooves. Notice that the amount of recirculation increased with each respective groove, while the velocity in the grooves decreased along the span of the rotor blade. Also interesting was the way in which each groove impacted the flowpath individually, with upstream grooves impacting those downstream, causing a higher volume of flow to stay in the downstream grooves, while lowering the Mach number. Figure 43 shows how, even with the grooves, there was still a passage vortex formed. However, this vortex was not as substantial in any dimension as the vortex off the leading edge and suction surface of the untreated rotor.

Another demonstration of this phenomena along with the way in which the rotor wake injects flow into the groove can be seen in Figure 44 through Figure 48. These images are looking over the rotor blade, circumferentially around each individual groove. Along the flowpath from groove one to groove five, the injection and driven cavity
structure of the flow increased, while the velocity of the flow entering the cavity decreased.

A possible reason for the increase in stall margin due to the circumferential groove casing treatment has to do with the boundary layer at the case. The way in which the grooves removed flow from the rotor wake, circulated it in a driven cavity structure in the groove, and then re-injected it slightly upstream of the removal point, it was essentially providing a moving wall at the grooves. This means that the 5% axial chord length solid areas between grooves allowed small boundary layers to build, but the 10% axial chord length grooves removed the boundary layer and provided a new sliding wall up to the point where the groove ended, drew some of the flow up, and then started another new boundary layer at the next portion of the case.

Another possible reason for the stall margin increase is the lack of a tip vortex propagating through the passage of the treated rotor. The vortex beginning at the tip of the rotor has the same effect as a large boundary layer at the endwall. A growing boundary layer closes down the passage and reduces the mass flow range of the compressor. As the pressure rise increases and the mass flow decreases, the boundary layer at the endwall plays a large part in the stall characteristics of the compressor. Controlling or even eliminating the tip vortex as in the case of the circumferential groove casing treatment can extend the mass flow range of the compressor.
Figure 39: Flow Injected in to and out of the Second Groove
Figure 40: Flow Injected in to and out of the Third Groove
Figure 41: Flow Injected in to and out of the Fourth Groove
Figure 42: Flow Injected into and out of the Fifth Groove
Figure 43: Streamlines Showing Combination of Outflow From All Five Circumferential Grooves
Figure 44-48: Grooves 1-5 (Left to Right, Top to Bottom) Showing Structure of Flow Immediately Above

Rotor Blade
4.2.2 Compressor Rotor Casing Treatment Study Conclusions

This study compared a simple compressor stator-rotor-stator stage with and without a casing treatment. The modeled casing treatment consisted of five circumferential grooves, which has proven in previous experiments to increase the surge margin of the compressor. It was hoped to gain understanding into the nature of the flow and some of the reasons that casing treatments can have a positive effect on surge margin.

Findings of note included the structure of the flow inside the grooves and the differences in flow at the endwall between the treated and untreated rotors. As expected, the mass flow rate, efficiency and pressure ratio at a speed were all reduced. However, overall pressure rise increased 0.4% before the speed line flattened out and mass flow dropped to unreasonable levels. There is another benefit in having a flat speed line characteristic before surge, and that is the predictability of the compressor over time. As the compressor degrades, the surge characteristics will stay fairly constant.

The grooves in the casing treatment act as a sort of pressure relief, allowing the flow at the endwall to have a small turbulent boundary layer, as opposed to a much larger passage vortex that reduces the flow area through the flowpath. High energy flow is injected into the grooves which reduces the efficiency, but allows for a much larger mass flow range in the compressor.

Future work involving this casing treatment could involve optimizing the size, location, and number of circumferential grooves. It may be possible to realize a similar surge margin gain to what is seen in this study with less of an efficiency hit. Another possible future investigation could be in to other ways to reduce or eliminate the passage
vortex that comes from the rotor tip. Ideas could include active flow control on each rotor blade such as airflow jets or new novel blade designs that blend the suction and pressure surface flows similar to a winglet at the end of an airplane wing. Transient simulations could also assist in the understanding of flow interactions and subtleties due to casing treatments.
Chapter 5.  Conclusions

Numerical simulation has been used to gain understanding in two areas of axial compressors: stator seals, and compressor rotor casing treatments. Both are part of the secondary flow of the compressor, and both can enhance or degrade compressor performance. Stator seals are a requirement in every compressor to provide clearance between the rotating and non-rotating components. Rotor casing treatments are not found in all compressors, but are one way to increase the surge margin of a compressor design with a small penalty on efficiency, pressure rise, and flow at a speed.

Stator seals were analyzed based on seal tooth gap and outside flow introduced into the chamber. It was shown that as the seal gap decreases, the amount of leakage through the seal cavity also decreases. It is desirable to minimize this flow to minimize losses inherent in its leaving and re-entering the main flowpath of the stator. It was also shown that the flow in the seal cavity takes the structure of a driven cavity. The work done on the air in the cavity was from the viscous shear effects of the rotating components, and caused the temperature in the cavity to rise, possibly to unsafe levels if there was a very small amount of flow moving through the cavity. Therefore, the stator seal cavity also served as a form of ventilation for the rotating and non-rotating components.
Comparing the Stator 2 seal passage from the rear of the compressor to the Stator 1 passage from the front showed that both shared a common driven-cavity flow structure through the passage. It also showed how even in the same compressor there can be many problems to overcome in something so simple as the stator seals alone. Looking at Stator 2 showed that putting a small amount of pressure in the cavity (in this case from a knife seal at the centerline) could bring some benefits due to the reduction of flow being removed from the main flowpath.

Many forms of compressor rotor casing treatments were described in the Theory section, however analysis centered on a circumferential groove setup. The compressor design was a stator-rotor-stator setup used in a research rig and had a baseline design point pressure ratio of 1.10, 100% corrected speed of 5000 rpm, and corrected flow design intent of 20 lbm/s. The flowpath had a constant radius. The treatment analyzed for this study consisted of five grooves over the rotor spanning the full circumference of the compressor case. The grooves were 20% blade chord in depth and 10% blade chord in length, and were centered over the rotor blade.

Overall, the grooves helped to even out the flow at the blade tips. Pressure pulses that without the casing treatment would have caused a passage vortex were instead absorbed into the treatment and then injected back into the flow between the two blades. In fact, the flow in the grooves was cyclical and wave-like, peaking with each pass of the blade pressure surface and then returning to a lower energy state. This leveling and reduction of the passage vortex allowed the mass flow and pressure rise range of the compressor to be extended beyond that of the same compressor with no casing treatment.
As expected, the calculations showed that this absorption of pressure pulses did come with a cost in overall efficiency.

Future work could be done in both areas of this study to gain even more understanding and insight into these significant flow components of axial compressors. For stator seals, future work could focus on refining the model of the stator passages and making them more advanced. The first step would be to create a three-dimensional geometry slice from the two-dimensional file used in this study. The next step would be to model the full passage including the full passage and stator blade. Finally, a coupled three-dimensional model of the full compressor including rotor and stator blades and stator seal passages could be considered for maximum understanding of the interactions of seal secondary flow and main flowpath flow.

In the compressor rotor casing treatment study, a three-dimensional coupled model was already used. Therefore, future study could be designed to include optimizations of the various geometric parameters of the casing treatment. Judging from the individual groove flows, it is possible that shallower grooves could lead to similar surge margin benefits with less efficiency loss. A configuration involving grooves that started deep at the blade leading edge and got shallower towards the trailing edge of the blade could also realize some of the same improvements. Some form of active flow control or blade redesign could also be a topic of future study.

To conclude, there is always more to be done in the field of gas turbine engines as aircraft customers demand more and more from their aircraft. Different markets have different requirements, but the basic challenges remain the same for all. Issues of
efficiency, noise, and thermal management will continue to drive engine and aircraft manufacturers to work together to explore and innovate in the fields of fluid mechanics and computational fluid dynamics. Research such as that presented here has the possibility of improving tomorrow’s engine designs and continuing the trend of making aircraft cleaner and more efficient to operate.
List of References
List of References

1, Boeing Selects Two 7E7 Dreamliner Engine Partners,  

2, A380 Double Deck Family - Flagship of the 21st Century,  

3, Kablitz, S., Passrucker, H., Hennecke, D., and Engber, M., Experimental Analysis of  

in Axial Flow Turbines, With Emph", ASME Journal of Turbomachinery, 123,  

5, Adamczyk, J., Celestina, M., and Greitzer, E., "The Role of Tip Clearance in High  

6, Suder, K.L., Hathaway, M.D., Thorp, S.A., Strazisar, A.J, and Bright, M.B.,  
"Compressor Stability Enhancement Using Discrete Tip Injection", ASME  

Curve Instability of a Mixed Flow Pump by Use of J-groove", ASME Journal of  
Fluids Engineering, 122, 2000, pp. 592-597.

8, Saha, S.L., Kurokawa, J., Matsui, J., and Imamura, H., "Passive Control of Rotating  
Stall in a Parallel-Wall Vaned Diffuser by J-Groove", ASME Journal of Fluids  

9, Cheng, P., Prell, M., Greitzer, E., and Tan, C., "Effects of Compressor Hub Treatment  

10, Takata, H., Tsukuda, Y., "Study on Mechanisms of Stall Margin Improvement of  


Appendix A1

ADPAC Sample Case Input

# Standard ADPAC input file: check all data
#
#CASENAME = adpac
PREF  = 2116.80
TREF  = 518.70
RGAS  = 1716.32
DIAM  = 0.083333
GAMMA = 1.400000
RPM(1) = 0.000000
RPM(2) = -5000.000000
RPM(3) = 0.000000
RPM(4) = -5000.000000
NBLD(1) = 44.0
NBLD(2) = 36.0
NBLD(3) = 44.0
FREST  = 0.00
FNCMAX = 500.00
FITCHK = 100.00
FMULTI = 3.00
FSUBIT = 3.00
FFULMG = 1.00
FCOAG1 = 3.00
FCOAG2 = 2.00
FITF MG = 250.00
FRDMUL = 0.000000
FTOTSM = 0.000000
CFL    = -5.000000
CFMAX  = 2.500000
RMACH  = 0.150000
EPSTOT = 0.100000
EPSX   = 1.000000
EPSY   = 1.000000
EPSZ   = 1.000000
VIS2   = 0.500000
VIS4   = 0.015625
DISSCG = 5.000000
VISCG2 = 0.125000
VISCG4 = 0.000000
FBCWARN = 1.000000
FINVII = 1.000000
FWALLF = 1.000000
FVTSPAC = 2.500000
FTIMEI = 1.000000
FTURBI = 1.000000
FTURBB = 20.000000
FFILT = 1.000000
FSOLVE = 1.000000
FRESID = 1.000000
FUNINT = 999999.000000
FGRAFIX = 0.000000
FGRAFINT = 1.000000
FIMGSAV = 0.000000
FIMGINT = 999999.000000
PRNO = 0.700000
PRNO = 0.900000
P3DPRT = 1.000000
Appendix A2

ADPAC Sample Boundata File

# Standard ADPAC bdata file : check all data
# Created by mkad_bdata
# Number of blade rows = 3
# Pref = 14.70
# Tref = 518.70
# Rgas = 1716.32
# Dref = 24.00000
# Gamma = 1.40000
# Omega = -5000.00000

# Overall inlet conditions
INLETT 1 1 I I P P J K 1 1 1 53 1 45 1 53 1 45
NDATA 21

<table>
<thead>
<tr>
<th>RAD</th>
<th>PTOT</th>
<th>TTOT</th>
<th>BETAR</th>
<th>BETAT</th>
</tr>
</thead>
<tbody>
<tr>
<td>10.00000</td>
<td>0.95000</td>
<td>1.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>10.06000</td>
<td>1.00000</td>
<td>1.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>10.63800</td>
<td>1.00000</td>
<td>1.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>10.78000</td>
<td>1.00000</td>
<td>1.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>10.89800</td>
<td>1.00000</td>
<td>1.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>11.00200</td>
<td>1.00000</td>
<td>1.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>11.09800</td>
<td>1.00000</td>
<td>1.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>11.18400</td>
<td>1.00000</td>
<td>1.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>11.26600</td>
<td>1.00000</td>
<td>1.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>11.34200</td>
<td>1.00000</td>
<td>1.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>11.41400</td>
<td>1.00000</td>
<td>1.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>11.48200</td>
<td>1.00000</td>
<td>1.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>11.54800</td>
<td>1.00000</td>
<td>1.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>11.61200</td>
<td>1.00000</td>
<td>1.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>11.67200</td>
<td>1.00000</td>
<td>1.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>11.73000</td>
<td>1.00000</td>
<td>1.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>11.78800</td>
<td>1.00000</td>
<td>1.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>11.84200</td>
<td>1.00000</td>
<td>1.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>11.89600</td>
<td>1.00000</td>
<td>1.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
<tr>
<td>11.94000</td>
<td>1.00000</td>
<td>1.00000</td>
<td>0.00000</td>
<td>0.00000</td>
</tr>
</tbody>
</table>
12.00000 0.95000 1.00000 0.00000 0.00000

# Overall exit conditions
EXITTT 3 3 I I M M L L 101 101 1 53 1 45 1 53 1 45
PEEXIT 1.05

# Patches up-and-downstream of blades
PATCH 1 1 K K M P I J 45 1 1 29 1 53 1 29 1 53
PATCH 1 1 K K P M I J 1 45 1 1 29 1 53 1 29 1 53
PATCH 1 1 K K M P I J 45 1 1 81 101 1 53 81 101 1 53
PATCH 1 1 K K P M I J 1 45 1 1 81 101 1 53 81 101 1 53
PATCH 2 2 K K M P I J 45 1 1 21 1 53 1 21 1 53
PATCH 2 2 K K P M I J 1 45 1 1 21 1 53 1 21 1 53

# patchfinder
PATCH 2 2 K K M P I J 45 1 1 73 93 1 53 73 93 1 53
PATCH 2 2 K K P M I J 1 45 73 93 1 53 73 93 1 53

# /patchfinder
PATCH 3 3 K K M P I J 45 1 1 21 1 53 1 21 1 53
PATCH 3 3 K K P M I J 1 45 1 1 21 1 53 1 21 1 53
PATCH 3 3 K K M P I J 45 1 1 73 101 1 53 73 101 1 53
PATCH 3 3 K K P M I J 1 45 73 101 1 53 73 101 1 53

# Clearance patches

#SSVI 1 1 I I P P J K 1 1 1 53 1 45 1 53 1 45
# RPMWALL TWALL
# 0.0 0.0

#SSVI 1 1 I I M M J K 101 101 1 53 1 45 1 53 1 45
# RPMWALL TWALL
# 0.0 0.0

SSVI 1 1 J J P P I K 1 1 1 101 1 45 1 101 1 45
RPMWALL TWALL
45
  # RPMWALL  TWALL
  #  0.0     0.0

SSVI  3 3 J J P P I K 1 1 1 101 1 45 1 101 1
45
  RPMWALL  TWALL
         0.0     0.0

SSVI  3 3 J J M M I K 53 53 1 101 1 45 1 101 1
45
  RPMWALL  TWALL
         0.0     0.0

SSVI  3 3 K K P P I J 1 1 21 73 1 53 21 73 1
53
  RPMWALL  TWALL
         0.0     0.0

SSVI  3 3 K K M M I J 45 45 21 73 1 53 21 73 1
53
  RPMWALL  TWALL
         0.0     0.0

#SSVI  4 4 I I P P J K 1 1 1 5 1 13 1 5 1
13
# RPMWALL  TWALL
#  -5000.0    0.0

SSVI  4 4 J J P P I K 1 1 1 53 1 13 1 53 1
13
  RPMWALL  TWALL
        -5000.0    0.0

SSVI  4 4 J J M M I K 5 5 1 53 1 13 1 53 1
13
  RPMWALL  TWALL
          0.0     0.0

SSVI  4 4 K K M M I J 13 13 1 53 1 5 1 53 1
5
  RPMWALL  TWALL
          0.0     0.0

# Mixing planes
MBCAVG  1 2 I I M P J K 101 1 1 53 1 45 1 53 1
45
NSEGS
  1
LBLOCK2B LFACE2B LDIR2B L2LIMB M2LIM1B M2LIM2B N2LIM1B N2LIM2B
  2  I  P  1  1  53  1  45
MBCAVG  2 1 I I P M J K 1 101 1 53 1 45 1 53 1
45
NSEG

1

LBLOCK2B LFACEx2B LDIR2B L2LIMB M2LIM1B M2LIM2B N2LIM1B N2LIM2B
1 I M 101 1 53 1 45

MBCAVG

2 3 I I M P J K 93 1 1 53 1 45 1 53 1

NSEG

1

LBLOCK2B LFACEx2B LDIR2B L2LIMB M2LIM1B M2LIM2B N2LIM1B N2LIM2B
3 I P 1 1 53 1 45

MBCAVG

3 2 I I P M J K 1 93 1 53 1 45 1 53 1

NSEG

1

LBLOCK2B LFACEx2B LDIR2B L2LIMB M2LIM1B M2LIM2B N2LIM1B N2LIM2B
2 I M 93 1 53 1 45

### Entries for tip 1 in block 4

PATCH

4 4 I I M M J K 1 1 1 5 1 13 1 5 1

PATCH

4 4 I I M M J K 53 53 1 5 1 13 1 5 1

# patchfinder

PATCH

4 2 K K M P I J 13 1 1 53 1 5 21 73 49

53

PATCH

2 4 K K P M I J 1 13 21 73 49 53 1 53 1

5

PATCH

2 4 K K M P I J 45 1 21 73 49 53 1 53 1

5

PATCH

4 2 K K P M I J 1 45 1 53 1 5 21 73 49

53

# /patchfinder

ENDDATA

### Startup inlet/exit conditions

INLETT

2 2 I I P P J K 1 1 1 53 1 45 1 53 1

45

NDATA

21

RAD PTOT TTOT BETAR BETAT

10.10000 0.99728 1.00000 0.17173 -11.98009
10.45800 0.99728 1.00000 0.14446 -11.98406
10.63800 0.99728 1.00000 0.11717 -11.96934
10.78000 0.99728 1.00000 0.08986 -11.97330
10.89800 0.99728 1.00000 0.06058 -11.97727
11.00200 0.99728 1.00000 0.03324 -11.98521
| 11.09800 | 0.99728 | 1.00000 | 0.00391 | -11.97046 |
| 11.18400 | 0.99728 | 1.00000 | -0.02544 | -11.97443 |
| 11.26600 | 0.99728 | 1.00000 | -0.05285 | -11.97840 |
| 11.34200 | 0.99728 | 1.00000 | -0.08031 | -11.98636 |
| 11.41400 | 0.99728 | 1.00000 | -0.10777 | -11.97159 |
| 11.48200 | 0.99728 | 1.00000 | -0.13334 | -11.97954 |
| 11.54800 | 0.99728 | 1.00000 | -0.15692 | -11.98353 |
| 11.61200 | 0.99728 | 1.00000 | -0.17856 | -11.96873 |
| 11.67200 | 0.99728 | 1.00000 | -0.20028 | -11.97670 |
| 11.73000 | 0.99728 | 1.00000 | -0.21803 | -11.98069 |
| 11.78800 | 0.99728 | 1.00000 | -0.23390 | -11.96986 |
| 11.84200 | 0.99728 | 1.00000 | -0.24774 | -11.97385 |
| 11.89600 | 0.99728 | 1.00000 | -0.25578 | -11.98184 |
| 11.94800 | 0.99728 | 1.00000 | -0.26178 | -11.98583 |
| 11.99800 | 0.99728 | 1.00000 | -0.26196 | -11.97499 |

**INLETT**

| 3 | 3 | I | I | P | P | J | K | 1 | 1 | 1 | 53 | 1 | 45 | 1 | 53 | 1 |

**NDATA**

<table>
<thead>
<tr>
<th>RAD</th>
<th>PTOT</th>
<th>TTOT</th>
<th>BETAR</th>
<th>BETAT</th>
</tr>
</thead>
<tbody>
<tr>
<td>10.10000</td>
<td>1.10864</td>
<td>1.03355</td>
<td>0.18753</td>
<td>-46.28689</td>
</tr>
<tr>
<td>10.45800</td>
<td>1.10816</td>
<td>1.03297</td>
<td>0.18347</td>
<td>-45.65061</td>
</tr>
<tr>
<td>10.63800</td>
<td>1.10760</td>
<td>1.03239</td>
<td>0.18142</td>
<td>-45.08957</td>
</tr>
<tr>
<td>10.78000</td>
<td>1.10641</td>
<td>1.03200</td>
<td>0.17948</td>
<td>-44.60846</td>
</tr>
<tr>
<td>10.89800</td>
<td>1.10544</td>
<td>1.03162</td>
<td>0.17861</td>
<td>-44.19051</td>
</tr>
<tr>
<td>11.00200</td>
<td>1.10476</td>
<td>1.03142</td>
<td>0.17974</td>
<td>-43.80757</td>
</tr>
<tr>
<td>11.09800</td>
<td>1.10476</td>
<td>1.03123</td>
<td>0.17989</td>
<td>-43.49229</td>
</tr>
<tr>
<td>11.18400</td>
<td>1.10476</td>
<td>1.03104</td>
<td>0.18218</td>
<td>-43.22555</td>
</tr>
<tr>
<td>11.26600</td>
<td>1.10403</td>
<td>1.03104</td>
<td>0.18250</td>
<td>-43.00863</td>
</tr>
<tr>
<td>11.34200</td>
<td>1.10408</td>
<td>1.03085</td>
<td>0.18282</td>
<td>-42.83263</td>
</tr>
<tr>
<td>11.41400</td>
<td>1.10408</td>
<td>1.03085</td>
<td>0.18320</td>
<td>-42.69768</td>
</tr>
<tr>
<td>11.48200</td>
<td>1.10408</td>
<td>1.03104</td>
<td>0.18157</td>
<td>-42.59438</td>
</tr>
<tr>
<td>11.54800</td>
<td>1.10408</td>
<td>1.03104</td>
<td>0.18066</td>
<td>-42.56536</td>
</tr>
<tr>
<td>11.61200</td>
<td>1.10408</td>
<td>1.03123</td>
<td>0.17854</td>
<td>-42.56919</td>
</tr>
<tr>
<td>11.67200</td>
<td>1.10408</td>
<td>1.03142</td>
<td>0.17504</td>
<td>-42.61626</td>
</tr>
<tr>
<td>11.73000</td>
<td>1.10476</td>
<td>1.03162</td>
<td>0.16960</td>
<td>-42.72805</td>
</tr>
<tr>
<td>11.78800</td>
<td>1.10476</td>
<td>1.03200</td>
<td>0.16206</td>
<td>-42.88441</td>
</tr>
<tr>
<td>11.84200</td>
<td>1.10544</td>
<td>1.03239</td>
<td>0.15463</td>
<td>-43.10503</td>
</tr>
<tr>
<td>11.89600</td>
<td>1.10544</td>
<td>1.03277</td>
<td>0.14303</td>
<td>-43.36951</td>
</tr>
<tr>
<td>11.94800</td>
<td>1.10612</td>
<td>1.03335</td>
<td>0.12723</td>
<td>-43.70991</td>
</tr>
<tr>
<td>11.99800</td>
<td>1.10680</td>
<td>1.03395</td>
<td>0.10921</td>
<td>-44.11499</td>
</tr>
</tbody>
</table>

**EXITT**

| 1 | 1 | I | I | M | M | L | L | 101 | 101 | 1 | 53 | 1 | 45 | 1 | 53 | 1 |

**PEXIT**

0.94762

**EXITT**

| 2 | 2 | I | I | M | M | L | L | 93 | 93 | 1 | 53 | 1 | 45 | 1 | 53 | 1 |

**PEXIT**

1.00816

ENDDATA
Vita

Charles C. Cates was born in Richmond, Virginia on February 10, 1979. He attended Springbrook High School, Silver Spring, Maryland, and graduated from Prince George High School, Prince George, Virginia, in 1997. That year, he began studying Mechanical Engineering at Virginia Commonwealth University, Richmond, Virginia. He earned a Bachelor of Science in Mechanical Engineering in May, 2002. He continued studies at VCU under Dr. Dan Cook, and was later guided by Dr. James T. McLeskey, Jr. While a graduate student at VCU he served as a Graduate Teaching Assistant for both Thermodynamics and Fluid Dynamics classes. Also while at VCU, he was accepted as a University Turbine Systems Research Fellow, a cooperative agreement between the US Department of Energy and Clemson University. As a research fellow he worked with the Compressor Aero department at Rolls-Royce Corporation, Indianapolis Indiana. He is currently employed as a Senior Engineering Associate in the System Performance department at Rolls-Royce.