2004

Modeling periodic blade stresses due to stator-rotor interactions using computational fluid dynamics

Lisa M. Barrett

Follow this and additional works at: http://scholarworks.rit.edu/theses

Recommended Citation

This Thesis is brought to you for free and open access by the Thesis/Dissertation Collections at RIT Scholar Works. It has been accepted for inclusion in Theses by an authorized administrator of RIT Scholar Works. For more information, please contact ritscholarworks@rit.edu.
Modeling Periodic Blade Stresses due to Stator-Rotor Interactions Using Computational Fluid Dynamics

by

Lisa M. Barrett

A Thesis Submitted in Partial Fulfillment of the Requirement for the

Master of Science

in

Mechanical Engineering

Approved by:

Dr. E.A. DeBartolo
Department of Mechanical Engineering

Dr. J.D. Kozak
Department of Mechanical Engineering

Dr. K.B. Kochersberger
Department of Mechanical Engineering

Dr. E.C. Hensel
Department Head of Mechanical Engineering

Elizabeth A. DeBartolo (Thesis Advisor)

Jeffrey D. Kozak

Kevin B. Kochersberger 1903-2003

Edward C. Hensel

DEPARTMENT OF MECHANICAL ENGINEERING
ROCHESTER INSTITUTE OF TECHNOLOGY

December 03, 2004
Permission Granted

Modeling Periodic Blade Stresses due to Stator-Rotor Interactions Using Computational Fluid Dynamics

I, Lisa M. Barrett, hereby grant permission to the Wallace Library of the Rochester Institute of Technology to reproduce my thesis in whole or in part. Any reproduction will not be for commercial use or profit.

Date: 01/30/05 Signature of Author: Lisa M. Barrett
Modeling Periodic Blade Stresses due to Stator-Rotor Interactions Using Computational Fluid Dynamics

by

Lisa M. Barrett
Rochester Institute of Technology
December 3, 2004

ABSTRACT

High cycle fatigue (HCF) is a common problem in military turbofan engines, resulting in engine component failure, and billions of dollars a year in repair and maintenance. HCF occurs due to engine component aerodynamic interactions, including vortical and potential field effects. In 1987, the IHPTET program was created by the U.S. Department of Defense. One of the goals of this program was to reduce high cycle fatigue in engine components. A method of reducing the amount of HCF occurring in a turbofan engine is referred to as trailing edge blowing (TEB). TEB reduces the vortical, or wake, components of a fluid flow that propagate from upstream stator blades, to impinge on the surfaces of downstream rotor blades, one of the causes of HCF. The experimental results from the study of an F109 turbofan engine showed that TEB did reduce the amount of HCF related stress on the rotor blades; however, the amount of TEB used compromised the overall efficiency of the compressor by too high an amount. Thus, it is necessary to find an optimal level of TEB to achieve reduction in HCF forces, while maintaining high compressor efficiency. Due to the absence of an engine test stand for experimental testing, the solution was simulated using computational fluid dynamics (CFD). The current investigation focuses on the necessary preliminary steps taken in optimizing TEB, using CFD. TEB was not simulated, however the following investigation paves the way for future CFD work, including a method of incorporating TEB into a CFD simulation. The work also outlines the mistakes made, so that they are not made again by future researchers, as well as the recommended methods of modeling the F109 stator-rotor system using CFD. To the knowledge of the current investigator, using CFD to model rotor blade surface pressures in the F109 turbofan engine, due to the presence of upstream stators, has not previously been attempted.
Acknowledgements

I would like to thank Dr. Elizabeth DeBartolo for being an excellent advisor and mentor over the past several years. Dr. DeBartolo has not only advised me in my academic studies, but has been a guiding force throughout this project. Without her direction and open-mindedness this work would not have been possible.

I would also like Dr. Jeffrey Kozak. His research at Virginia Tech was the basis for this work, and I appreciate his willingness to explain what he did and to guide me in my technical work. I also appreciate the work that both he and Dr. DeBartolo completed in order to receive funding for this project.

To Dr. Kevin Kochersberger, thank you for agreeing to be a member of my committee, and also for the guidance you have given me both for this project and during my studies at RIT.

I would like to thank Dr. Mark Kempski, for teaching me everything I ever needed to know about Fast Fourier Transforms. This simple operation proved to very important to my CFD simulations. I also appreciate his wisdom and encouragement when I have felt like an end to this project was unattainable.

To all of the aforementioned, thank you for the careful consideration and guidance you have given me concerning narrowing the scope of this project.

I would also like to thank a few of my RIT peers. I would like to thank Drew Walter and Jay Grow for the help in the wind tunnel lab. I would also like to thank Julie Jones, for allowing me to use her equipment that she designed and built for her own thesis work, for my wind tunnel experiments.

Thank you to Suman Basu and Taher Attari, both of whom helped me learn the CFD software. I would like to thank Taher for sitting with me for hours in the CFD lab, teaching how to use GAMBIT and Fluent. I would like to thank Suman, although I have never met him, for helping me with my Fluent problems via the online chat at 1 am, when I thought all hope was lost with my simulations.

Last of all, I would like to thank my family, Dad, Mom, Jennifer, Amy, and Mallory. Without their love and constant encouragement from Arizona I never would have made it through my years at RIT, let alone this project. I love you all!
ABSTRACT ............................................................................................................... 2

Acknowledgements .............................................................................................. 3

List of Figures ........................................................................................................ 7

List of Tables .......................................................................................................... 9

1. Introduction........................................................................................................ 10
   1.1 Background and Motivation ...................................................................... 10
   1.2 Previous Research: .................................................................................... 14
      1.2.1 Stator-Rotor Interactions ................................................................ 14
      1.2.2 CFD Analysis of HCF Issues ........................................................... 18
      1.2.3 Trailing Edge Blowing ..................................................................... 19
      1.2.4 Engine Component Fatigue Analysis .............................................. 21
   1.3 Objectives of Current Investigation ......................................................... 23

2. Software Validation ............................................................................................ 25
   2.1 Software Utilities: ....................................................................................... 25
      2.1.1 GAMBIT ......................................................................................... 25
      2.1.2 Fluent .............................................................................................. 26
         2.1.2.1 Mass Conservation Equation ............................................... 26
         2.1.2.2 Momentum Conservation Equation ................................... 26
   2.2 Validation Case 1 - NACA 0012 Airfoil ..................................................... 27
      2.2.1 Experimental Data .......................................................................... 27
         2.2.1.1 Experimental Test Setup ....................................................... 28
         2.2.1.2 Experimental Results ........................................................... 29
      2.2.2 CFD Simulation ............................................................................. 29
         2.2.2.1 GAMBIT Modeling .............................................................. 29
         2.2.2.2 Fluent Setup ........................................................................... 32
      2.2.3 Comparison Results ....................................................................... 33
   2.3 Validation Case 2 - Flat Plate Simulation .................................................. 35
      2.3.1 Experimental Apparatus .................................................................. 35
         2.3.1.1 Flat Plates ............................................................................... 36
         2.3.1.2 RIT Wind Tunnel ..................................................................... 38
         2.3.1.3 Pressure Transducer .............................................................. 40
         2.3.1.4 Velocity Inlet Generator ......................................................... 40
      2.3.2 Experimental Method ...................................................................... 41
         2.3.2.1 Uniform Velocity Inlet ............................................................ 41
         2.3.2.2 Non-Uniform Velocity Inlet ................................................... 43
      2.3.3 CFD Simulation .............................................................................. 44
         2.3.3.1 GAMBIT Modeling .............................................................. 45
         2.3.3.2 Fluent Setup ........................................................................... 49
      2.3.4 Comparison Results ....................................................................... 51
         2.3.4.1 Uniform Inlet Velocity Cases ................................................. 52
         2.3.4.2 Non-Uniform Inlet Velocity Cases ....................................... 55

3. Stator-Rotor Simulation – Preliminary Steps .................................................... 58
   3.1 Existing Experimental Results ................................................................. 58
      3.1.1 Experimental Test Equipment ......................................................... 58
         3.1.1.1 Allied Signal F109 Turbofan Engine ...................................... 59
         3.1.1.2 Inlet Guide Vanes .................................................................. 60
<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.1.1.3</td>
<td>Engine Inlet Design</td>
<td>60</td>
</tr>
<tr>
<td>3.1.1.4</td>
<td>Data Collection Method</td>
<td>61</td>
</tr>
<tr>
<td>3.1.2</td>
<td>Experimental Data</td>
<td>62</td>
</tr>
<tr>
<td>3.1.2.1</td>
<td>Experimental Setup</td>
<td>62</td>
</tr>
<tr>
<td>3.1.2.2</td>
<td>Experimental Data Manipulation</td>
<td>63</td>
</tr>
<tr>
<td>3.2</td>
<td>CFD Setup</td>
<td>64</td>
</tr>
<tr>
<td>3.2.1</td>
<td>System Geometry</td>
<td>65</td>
</tr>
<tr>
<td>3.2.2</td>
<td>Use of Experimental Data</td>
<td>66</td>
</tr>
<tr>
<td>3.2.2.1</td>
<td>Assumptions</td>
<td>67</td>
</tr>
<tr>
<td>3.2.2.2</td>
<td>Time Step and Velocity Calculations</td>
<td>67</td>
</tr>
<tr>
<td>3.2.2.3</td>
<td>Inlet Function</td>
<td>70</td>
</tr>
<tr>
<td>3.2.3</td>
<td>Initial Modeling Considerations</td>
<td>71</td>
</tr>
<tr>
<td>3.2.4</td>
<td>Initial Boundary Conditions</td>
<td>76</td>
</tr>
<tr>
<td>3.2.4.1</td>
<td>Inlet Boundary Condition</td>
<td>76</td>
</tr>
<tr>
<td>3.2.4.2</td>
<td>Outlet Boundary Condition</td>
<td>78</td>
</tr>
<tr>
<td>3.2.4.3</td>
<td>Upper and Lower Boundary Conditions</td>
<td>79</td>
</tr>
<tr>
<td>3.2.5</td>
<td>Modeling Issues</td>
<td>80</td>
</tr>
<tr>
<td>4</td>
<td>Stator-Rotor Simulation – Final Approach</td>
<td>82</td>
</tr>
<tr>
<td>4.1</td>
<td>CFD Setup</td>
<td>82</td>
</tr>
<tr>
<td>4.1.1</td>
<td>GAMBIT Modeling</td>
<td>82</td>
</tr>
<tr>
<td>4.1.2</td>
<td>Fluent Setup</td>
<td>88</td>
</tr>
<tr>
<td>4.1.2.1</td>
<td>Fluent Models</td>
<td>88</td>
</tr>
<tr>
<td>4.1.2.2</td>
<td>Fluent Boundary Conditions</td>
<td>89</td>
</tr>
<tr>
<td>4.2</td>
<td>Stator-Rotor Simulation CFD Results</td>
<td>92</td>
</tr>
<tr>
<td>4.2.1</td>
<td>Fluent Results – Steady State Solution</td>
<td>92</td>
</tr>
<tr>
<td>4.2.1.1</td>
<td>Steady State Solution – 10k, No TEB</td>
<td>92</td>
</tr>
<tr>
<td>4.2.1.2</td>
<td>Steady State Solution – 10k, Full TEB</td>
<td>95</td>
</tr>
<tr>
<td>4.2.1.3</td>
<td>Steady State Solution – 11k, No TEB</td>
<td>95</td>
</tr>
<tr>
<td>4.2.1.4</td>
<td>Steady State Solution – 11k, Full TEB</td>
<td>98</td>
</tr>
<tr>
<td>4.2.2</td>
<td>Fluent Results – Unsteady Solution</td>
<td>99</td>
</tr>
<tr>
<td>4.2.2.1</td>
<td>Unsteady Solution – 10k, No TEB</td>
<td>102</td>
</tr>
<tr>
<td>4.2.2.2</td>
<td>Unsteady Solution – 10k, Full TEB</td>
<td>105</td>
</tr>
<tr>
<td>4.2.2.3</td>
<td>Unsteady Solution – 11k, No TEB</td>
<td>108</td>
</tr>
<tr>
<td>4.2.2.4</td>
<td>Unsteady Solution – 11k, Full TEB</td>
<td>110</td>
</tr>
<tr>
<td>4.3</td>
<td>Data Processing</td>
<td>112</td>
</tr>
<tr>
<td>4.3.1</td>
<td>Processing Objectives</td>
<td>113</td>
</tr>
<tr>
<td>4.3.2</td>
<td>Processing Method</td>
<td>113</td>
</tr>
<tr>
<td>4.3.3</td>
<td>Processing Results</td>
<td>117</td>
</tr>
<tr>
<td>4.3.3.1</td>
<td>Stress Spectra – 10k, No TEB</td>
<td>117</td>
</tr>
<tr>
<td>4.3.3.2</td>
<td>Stress Spectra – 10k, Full TEB</td>
<td>120</td>
</tr>
<tr>
<td>4.3.3.3</td>
<td>Stress Spectra – 11k, No TEB</td>
<td>120</td>
</tr>
<tr>
<td>4.3.3.4</td>
<td>Stress Spectra – 11k, Full TEB</td>
<td>121</td>
</tr>
<tr>
<td>5</td>
<td>Future Recommendations and Conclusions</td>
<td>123</td>
</tr>
<tr>
<td>5.1</td>
<td>Future Recommendations</td>
<td>123</td>
</tr>
<tr>
<td>5.1.1</td>
<td>Software Validation</td>
<td>123</td>
</tr>
<tr>
<td>5.1.2</td>
<td>Rotor Blade Modeling</td>
<td>124</td>
</tr>
<tr>
<td>5.1.3</td>
<td>CFD Simulation</td>
<td>125</td>
</tr>
<tr>
<td>5.2</td>
<td>Conclusions</td>
<td>127</td>
</tr>
</tbody>
</table>

References | 129 |
List of Figures

Figure 1.1: F109 Turbofan Engine [www.aircraftengineedesign.com] .............................................. 10
Figure 1.2: Stator-rotor diagram [Kozak, 2000, Dissertation] .......................................................... 12
Figure 1.3: HCF and LCF Stress Spectra Schematic ................................................................. 22
Figure 2.1: NACA 0012 wing setup ............................................................................................... 28
Figure 2.2: NACA 0012 airfoil ......................................................................................................... 30
Figure 2.3: GAMBIT modeled boundaries ...................................................................................... 30
Figure 2.4: Edge meshing near airfoil ............................................................................................ 31
Figure 2.5: NACA 0012 mesh .......................................................................................................... 32
Figure 2.6: Coupled implicit solver solution method ........................................................................ 33
Figure 2.7: Cp comparison for angle of attack = 8 .......................................................................... 34
Figure 2.8: Schematic diagram of Plate 1 ......................................................................................... 36
Figure 2.9: Switch box ..................................................................................................................... 37
Figure 2.10: Wind tunnel diagram .................................................................................................. 38
Figure 2.11: Pitot-tube placement ................................................................................................... 39
Figure 2.12: Plate mounted in wind tunnel ...................................................................................... 39
Figure 2.13: Diagram of velocity inlet generator ............................................................................. 40
Figure 2.14: Velocity Inlet Generator .............................................................................................. 41
Figure 2.15: Measured inlet velocity profile .................................................................................... 44
Figure 2.16: GAMBIT geometry, uniform inlet, AOA=8 ................................................................. 45
Figure 2.17: Leading edge mesh, uniform inlet, AOA=8 ................................................................. 46
Figure 2.18: Mesh, uniform inlet, AOA=8 ....................................................................................... 47
Figure 2.19: GAMBIT geometry, non-uniform inlet, AOA=8 .......................................................... 48
Figure 2.20: Mesh, non-uniform inlet, AOA=8 ............................................................................... 49
Figure 2.21: Inlet velocity function if-statement ............................................................................... 51
Figure 2.22: Velocity magnitude vs. height at inlet ......................................................................... 51
Figure 2.23: Cp comparison, AOA=-4,0,4 degrees, uniform inlet .................................................. 53
Figure 2.24: Cp comparison, AOA=12-20 degrees, uniform inlet .................................................... 54
Figure 2.25: Cp, AOA=20, non-uniform inlet .................................................................................... 55
Figure 2.26: Cp Comparison, AOA=-4-20, non-uniform inlet ........................................................ 56
Figure 3.1: F109 Turbofan cross-section ......................................................................................... 59
Figure 3.2: NACA 0015 IGV with TEB holes .................................................................................. 60
Figure 3.3: IGV inlet ring .................................................................................................................. 61
Figure 3.4: Inlet traverse ring .......................................................................................................... 61
Figure 3.5: Experimental Setup - Top View ..................................................................................... 63
Figure 3.6: Blade pass pressure profile, No TEB, 10k rpm ............................................................... 64
Figure 3.7: Side view of experimental setup ..................................................................................... 65
Figure 3.8: Rotor blade cross section dimensions ............................................................................. 66
Figure 3.9: Velocity Triangle ............................................................................................................ 69
Figure 3.10: FFT Approximation of total pressure data ................................................................. 70
Figure 3.11: 3 blade configuration, rectangular boundary ............................................................... 71
Figure 3.12: 3 blade rectangular configuration, meshes ................................................................. 72
Figure 3.13: 3-blade configuration, parallel boundaries ................................................................. 73
Figure 3.14: 3-blade configuration, highly skewed corner elements ............................................. 74
List of Tables

Table 2.1 Plate pressure tap locations ................................................................. 37
Table 2.2: Uniform inlet, AOA=8 edge meshing ................................................. 46
Table 2.3: Non-uniform inlet, AOA=8 edge meshing ........................................... 48
Table 3.1: Fan Blade Dimensions ...................................................................... 59
Table 4.1: Rotor mesh details ............................................................................. 84
Table 4.2: Stator mesh details ............................................................................ 87
1. Introduction

The first section of this chapter will present the background and motivation for this work. This section will discuss the United States Air Force’s IHPTET program and high cycle fatigue in general. The extent and cost, as well as the main causes of high cycle fatigue will be explained. This section will also give a basic overview of the aerodynamic phenomena involved in high cycle fatigue (HCF), and will give a schematic of the stator-rotor system. The second part of this chapter will present a review on past progress made in the field of researching modes and reasons for HCF. A review of pertinent studies and results will be given. The last part of this chapter will present, in detail, the objectives of this research.

1.1 Background and Motivation

This work will focus on the fatigue of rotor blades in the first stage of compression in an Allied Signal F109 turbofan engine. The first stage of the axial compressor in the F109 turbofan engine is composed of a set of titanium rotor blades, followed by a set of outlet guide vanes. Figure 1.1 shows a cutaway image of the F109 turbofan engine.

![Figure 1.1: F109 Turbofan Engine](www.aircraftenginedesign.com)

In 1987, the Integrated High Performance Turbine Engine Technology (IHPTET) program was created by the U.S. Department of Defense. The ultimate goal of this program is to double the nation’s propulsion capability by 2005 [IHPTET brochure]. Another purpose of the program, among other things, is to improve the lifetime of turbine engine components. One of the major lifetime inhibitors in turbine engines is high cycle
fatigue (HCF). HCF is especially problematic when engine components have already been affected by low cycle fatigue, which occurs during start up, maneuvering, and shut down. It has been identified as the leading cause of turbine engine failures, and also has been a source of high maintenance costs over the past decade. The High Cycle Fatigue Program is part of a national effort to help eliminate HCF, and to reduce engine maintenance cost. The IHPTET and the National HCF programs respond to the needs of Army, Navy, and Air Force jet engines in development, and in the field. The technologies developed by IHPTET work to improve performance and durability, while reducing the cost of operating and maintaining turbine engines.

High cycle fatigue (HCF) is the cyclic loading caused by internal interactions between aerodynamic phenomena and engine components, and has been observed in turbofan engines. The majority of unexpected failures and premature maintenance replacements have been attributed to the cyclic loading of engine components. This damage has occurred in an unacceptable range of $10^9$ cycles or less [Ritchie, et al., 1998]. This has been a cause of great concern for the U.S. Air Force, because high cycle fatigue failure leads to costly maintenance and, in some cases, catastrophic failure.

Due to continued advancements in compressor technology, axial spacing of engine components continues to decrease. This decreasing in engine component spacing increases the interactions between engine components. Modern engines, such as the Pratt and Whitney F119-PW-100 engine, which will be used as the power plant in the F-22 fighter, also include a set of inlet guide vanes (IGVs), or stators, upstream of the fan. Inlet guide vanes are desirable for use in modern engines, because they swirl the incoming air into the direction of fan rotation, and also equalizes the static pressure rise through the rotor and downstream stator. The inclusion of IGVs results in an additional aerodynamic interaction, between the IGV wake and the rotor blades.

For the data used in the current study, the F109 test engine was fitted with a set of IGVs, to examine the aerodynamic effects present. A schematic diagram illustrating a typical IGV-Rotor setup can be seen in Figure 1.2.
The IGVs are stationary, while the rotor blades rotate to compress the air as it passes through the stage. The introduction of the inlet guide vanes introduces an aerodynamic interaction between the IGVs and the rotors. The viscous nature of the flow causes velocity boundary layers to develop on both the upper and lower surfaces of the stator blades. At the stator blade trailing edges, these boundary layers meet and separate, forming viscous wakes that propagate downstream from the stator trailing edge. As the rotor blade rotates, it experiences a lower velocity field and a lower pressure as it crosses into the wake regions, directly aft of the stator blades. This results in a cyclic loading effect on the rotor blades, because of the unsteady surface pressure distributions that result, as they alternate from regions of high pressure to regions of low pressure.

The wakes propagating from the stator trailing edges are also affected by the presence of potential forcing functions. The potential disturbances are caused by the presence of the rotor blade structures. Historically, it had been held that these potential disturbances did not affect the viscous wake profile, but in recent years it has been proven that the disturbances caused by the potential disturbances are of the same order as the wake disturbances [Fabian, et al., 1995]. The potential disturbances comprise a potential forcing function that travels upstream acoustically from the rotor blades. The potential disturbance profile acts constructively and destructively with the viscous wake profile, causing further unsteadiness in the rotor blade surface pressures.
One of the methods that have been investigated to alleviate HCF loading is referred to as trailing edge blowing (TEB). The process of TEB involves reducing the momentum difference between the viscous wake regions and the free stream regions of the flow. This is achieved by bleeding air from a downstream stage of the axial compressor, and injecting it through holes drilled in the stator blades, directly into the wakes. This effectively fills the wakes, giving a uniform inlet velocity profile. This removes the cyclic loading caused by the alternating high and low total pressure regions, and has been theorized to increase fatigue life in the rotor blades [Kozak, 2000, Dissertation]. The criteria set forth by the U.S. Air Force for using TEB is that no more than 1% of the total engine mass flow can be used for wake-filling. A limit has been set on the amount of inlet mass flow that can be utilized by TEB; thus, a scheme of TEB that uses less than 1% of the total engine inlet mass flow, and the corresponding benefit to fatigue life, must be investigated. The experimental data for TEB already shows that full wake filling is achieved using a total mass flow of less than 1%, at subsonic rotor speeds [Kozak, Dissertation, 200]. However, the transonic rotor speeds require much more than 1% mass flow to achieve complete wake-filling. Future research will be required to investigate optimal TEB for transonic cases, because most military engines operate in the transonic range for the majority of flight time.

High cycle fatigue (HCF) has been found to be problematic after crack initialization has occurred by some other mode of damage. The common modes of damage that initialize material cracks are low cycle fatigue and foreign object damage. Low cycle fatigue (LCF) is damage associated with the forces experienced during landing, takeoff, and maneuvering. The load amplitudes for LCF are much larger than those for HCF, but they occur much less frequently. Once a material crack has been initiated, crack propagation is dictated both by the range of stresses applied, and material properties. The mode of failure, LCF or HCF, depends both on the starting crack size, and the stress amplitude applied.

Thus, in preventing HCF, it is necessary either to stop the crack from reaching the HCF threshold length, or to prevent the HCF stresses from exceeding the endurance stress. The use of TEB will serve to decrease the HCF stress amplitudes, by decreasing of the influence of the aerodynamic forcing functions.
1.2 Previous Research

The following section will explore the previous research completed in the fields of stator-rotor interactions, trailing edge blowing, past CFD work done in the field, and rotor fatigue life.

1.2.1 Stator-Rotor Interactions

Wake interaction is one of the principle types of blade-row interaction. It is the effect upon the flow through a downstream blade row, of the vortical and entropic wakes shed by one or more upstream rows [Probasco, 1997]. It has been shown analytically that the vortical wake forcing function causes perturbations in the velocity profile experienced by the rotor blades, but has no effect on the static pressure profile [Johnston, 1998]. Thus, the change in velocity due to the vortical wakes causes a rise or drop in total pressure, while static pressure remains constant.

The aerodynamic forcing functions causing HCF are composed of both downstream-propagating vortical wakes, emanating from the trailing edges of the IGVs, and upstream-propagating perturbations caused by the rotors, that are potential in nature [Falk, 2001]. Potential disturbances are inviscid perturbations, and are generated by the existence of a structural member, in this case, the rotor blades. More specifically, the unsteady local acceleration and deceleration of the flow around aerodynamic components leads to the generation of potential disturbances. At subsonic engine speeds, the disturbances propagate upstream from the surface of the rotor blades at sonic speeds.

The role of the upstream propagating potential disturbance had historically not been considered to greatly influence HCF. However, in the work shown by Falk and Jumper, among others, it was shown that these disturbances can interact greatly with the wake forcing function, and must be taken into consideration during analysis. In fact, it has been shown that the magnitude of a potential disturbance can be on the same order as that of a vortical, or wake-related, disturbance.

Fabian, et al., was among the first to prove that the potential forcing function disturbances were of the same magnitude as the viscous wake disturbances. This was proven in an experiment in which circular cylinders were placed at 80% chord upstream of a compressible cascade of vanes, for forward forcing, and then 80% downstream of the
cascade, for aft forcing. The results showed that aft forcing resulted in the same order of unsteady surface pressure as forward forcing [Fabian, et al, 1995]. The data were measured by embedding pressure transducers in the vanes of the cascade. The results also showed a sufficient periodicity in the unsteady pressure, allowing for ensemble averaging. Ensemble averaging is done by phase-locking the measurements, and the summing the data at each point. This is a common practice, but it may result in the truncation of some of the higher pressure measurement amplitudes. The researchers were able to capture the character of the forcing function by decomposing it into a primary sinusoid at the forcing frequency, and a first harmonic. The forcing function was represented using a series of sine waves, as shown in equation 1.1.

\[ P = A_p \sin(2\pi f_p t + \phi_p) + A_h \sin(2\pi f_h t + \phi_h) \]  \hspace{1cm} (1.1)

where \( A_p \) and \( A_h \) are the primary and harmonic amplitudes, \( f_p \) and \( f_h \) are the primary and harmonic frequencies, and \( \phi_p \) and \( \phi_h \) are the primary and harmonic phase shifts, respectively. The values for the coefficients were found using Fourier analysis software. The data obtained during aft forcing, in which the potential was the only disturbance to interact with the cascade, was close in magnitude to the data obtained during forward forcing, in which case both the potential and convective disturbances interact with the cascade. This showed that the potential forcing function has a great impact on the unsteady vane surface pressure, and thus on HCF.

The work done by Fabian was also supported by Falk, et al., 1997. In the experimental study of unsteady forcing in the F109 turbofan engine, the researchers found that the perturbed flow consists of both convective and acoustically radiated potential disturbances [Falk, 1997]. This study was very similar, in that unsteady forcing was caused by von Karman vortex shedding of circular cylinders, placed either upstream or downstream of the vanes; however, this study was completed in the actual production compression stage of the F109 turbofan engine. The data was decomposed into the potential and vortical components in the same manner as in the Fabian work, and was again represented using a series of sinusoidal functions, as in equation 1.1. The same conclusion, that the forward and aft forcing produced similar unsteady disturbances, was reached.
Fabian, et al., 1999, worked to quantify the unsteady IGV pressure response due to rearward forcing. The rearward forcing was achieved as in the previous Fabian work, with circular cylinders placed 0.8 chord lengths downstream of the stator vanes. There was no vortical wake present upstream of the vanes, meaning that the unsteady pressure response was due solely to the potential disturbances created by the cylinders. The data showed that the unsteadiness in the surface pressure response was a direct result of the potential forcing function caused by the cylinders. The conclusion reached was similar to the conclusion reached in the transonic cascade experiments performed by Fabian, et al., 1995. The unsteady response measured due to rearward forcing was of similar order and magnitude as the unsteady response due to forward forcing. Furthermore, the potential forcing function was shown to be two dimensional with regard to span direction [Fabian et al., 1999]. This is important because the modeling of the system in this work will involve assigning the forcing function as a pressure-inlet boundary condition in the CFD problem, which will be two-dimensional.

Probasco, et al., 1997, measured the unsteady surface pressure response on the IGVs, due to the upstream traveling potential field generated by the downstream rotor. As time progresses, the pressure wave created by the rotor travels upstream in the chordwise direction. Thus, because of the rotation of the rotors the pressure peaks should interact on a diagonal pathline with the viscous wake profile. Probasco, et al., also found that the magnitude of the unsteady pressure disturbances do not decay significantly within the IGV blade passage, and have a higher harmonic content than historically believed. Experimental results showed that the potential forcing function demonstrated a much higher harmonic content, as blade spacing between the IGV row and the rotor row decreased [Probasco, et al., 1997].

Work done by Johnston, et al., 1998, investigated the effect of upstream IGV wakes on the downstream rotor wake profile. The wake profile both upstream and downstream of the rotor was measured. Johnston, et al., found that the upstream propagating potential forcing function generated by the downstream rotor is periodic in nature in the tangential direction to the rotor blade, but decays exponentially in the axial direction [Johnston, 1998]. The same work also proved analytically that the static pressure profile experienced by the rotor was dependent only on the potential forcing
function, while the velocity profile was affected by both the potential forcing function and the viscous wake. The potential forcing function was calculated analytically, and then subtracted from the measured velocity function, to find the differential velocity function, that could be attributed to the viscous wake.

Johnston et al., 1998 also showed by experimentation that in the near rotor wake region, the measured wake width was larger, and the velocity deficit smaller, than predicted by empirical correlations. However, farther away from the rotor, the experimental data demonstrated excellent agreement in comparison to the empirical correlation, suggesting that the potential forcing function has more of an effect on the vortical wake, as spacing between components decreases. Furthermore, this work suggested that the potential forcing function actually decreased the velocity deficit created by the vortical wake [Johnston, 1998]. The unsteady velocity perturbation upstream of the rotor was shown to be predominantly potential in nature.

Falk, et al., 1999, succeeded in characterizing the unsteady velocity field aft of the F109 rotor. The wake region velocity was represented using an exponentially decaying function, and the potential region was represented by a sinusoidal wave. The two functions were added to achieve forcing function due to both the wake and the potential disturbances. The results compared well to experimental data of the measured velocities downstream of the rotor. These results were important, because they demonstrate that it is possible to model the total forcing function by adding a wake function and a potential function.

The work of Kozak, 2000, Dissertation, showed high-pressure fluctuations on the surface of IGVs, due to the potential forcing function. The peak-to-peak amplitudes were measured as high as 2.5 psi, for a transonic fan case. Subsonic tests were also performed, and the total pressure in the wake region of the flow was measured. A baseline for both the subsonic and transonic cases was examined, in which the IGVs were placed far upstream of the rotor, where the upstream propagating rotor potential field would have no effect on the wake profile. Subsonic and transonic cases were then run in which the IGVs were placed 0.45 fan blade chords upstream of the fan, which is a spacing typical of modern engines. These results were then compared to the baseline cases, in which the IGVs were not affected by the potential disturbances. For this work, only the subsonic
cases will be examined. It was found that the total pressure loss coefficient in the subsonic case was 40% less than the baseline case, and therefore, the upstream traveling rotor potential forcing function was beneficial, by reducing the pressure losses created by the IGV wake [Kozak, 2000, AIAA Paper].

### 1.2.2 CFD Analysis of HCF Issues

Computational fluid dynamics (CFD) has been utilized in the past in analyzing high cycle fatigue (HCF) issues. Numerical procedures have been developed to model the blade-row interactions, in the case of the upstream stator and downstream rotor. The most commonly used procedures involve either modeling only the downstream rotor blade row, and prescribing the differences in velocity and pressure due to the viscous wakes as an inlet condition, or, modeling both the stator and rotor rows, and varying the position of one of the rows to simulate blade motion [Probasco, et al., 1997]. In the Probasco work, a CFD analysis was performed and compared to experimental data. The CFD code used was developed specifically for use in this research. It was determined that to accurately model the interactions between blade rows, it was necessary to model roughly 1/3 of the actual blades present around the circumference. A periodic boundary condition was used in the Probasco investigation, which was the same type of boundary condition used for the velocity inlet in the current study. A total of 4000 time steps were necessary in the Probasco simulation, to achieve a converged steady state solution, and 20000 time steps were required to capture an entire revolution. The CFD simulation under-predicted the unsteady aspect of the interactions. Overall, the CFD analysis and the experimental results showed a good trend-wise agreement, demonstrating that CFD can be used for this application.

Commonly, CFD analysis is used to predict the aerodynamic response of engine components in two steps. First, the steady state response is modeled. Second, the unsteady aerodynamics is predicted, by using CFD analysis to predict the aerodynamic damping together with the aerodynamic forcing functions to find the resulting gust response. These analyses generally consider an isolated blade row, and utilize both the linear frequency domain and the non-linear time-marching analyses [Fleeter, et al.].
One of the problems with existing software is that most packages are unable to handle both the fluid part and the structural part of turbomachinery analysis, using the same finite element method. One finite element model able to handle both fluid and solids has been developed at Lawrence Livermore National Laboratories, and is called ALE3D. This software is a step in the right direction as far as modeling both the fluid and structural components, but cannot be used for most turbomachinery analysis. The package does not allow for a variable inflow velocity boundary condition at the inlet of the domain, nor does it allow for the output of blade surface pressures or mass flow rate. A modified code was created to facilitate the modeling of turbomachinery, and is called TAM-ALE3D [Fleeter, et al.]

Designers have examined HCF issues with a CFD analysis of a single blade row, with the unsteady forcing due to prescribed inflow/outflow boundary conditions, or the blade motion itself [Gottfried, et al., 2002]. It was theorized that this method of modeling blade-row coupling effects was inadequate, and may have led to a number of unexpected HCF failures. TAM-ALE3D, a three-dimensional Euler solver using a finite element scheme, was examined for use in simulating the unsteady IGV-rotor interactions. The geometry of the mesh consisted of one IGV blade and one rotor blade, and the applicable equations were marched forward in time to obtain an unsteady solution. This study examined the solution of IGV surface pressure, but in the current study, rotor surface pressures are examined. The simulation overestimated the IGV surface pressures, but the difference between the simulated data and the experimental data never exceeded 0.6%. The study showed that CFD was able to accurately predict unsteady aerodynamic interactions, when applied to an IGV-rotor setup.

1.2.3 Trailing Edge Blowing

Trailing edge blowing (TEB) is a method of filling in the wakes that are shed by upstream components in a turbofan engine first stage compressor. In the case to be examined, TEB is used to fill the wakes shed from a row of stator blades, placed upstream of a row of rotor blades. The wakes from the stators impinge on the rotors, causing cyclic loading that is the source of HCF. Decreasing the velocity deficit between
the wake and non-wake region reduces the fluctuations in static pressure experienced by the rotor blades, theoretically increasing fatigue life.

The first study [Bailie, et al., 2000] examined the use of TEB in reducing vibrations in the rotor blades. TEB was shown in this case to reduce the peak-to-peak strain amplitude of the rotor forcing function near the hub of the blade by up to 69%, in the first torsional mode. This was achieved using 0.8% of the total compressor rig mass flow rate. For the second leading edge bending mode of excitement, the peak-to-peak strain amplitude was decreased by nearly 80% using 0.6% of the total mass flow rate. The tests were run at a free stream Mach number of 0.6, and the air was supplied by a compressed air tank near the facility. Due to a shortage of compressed air, the experiments were stopped before finding the optimal configuration for TEB.

The study also experimented with supplying air to only certain TEB holes in each stator blade. It was found that while reductions are generally larger when TEB is applied over most of the span, significant reductions can be attained even when TEB is only applied over a small portion of the span. Three different engine speeds were tested. For the lowest engine speed, the strain amplitude was reduced by 80%, using only 0.3% of the compressor flow for TEB, showing that TEB is more efficient at lower engine speeds.

Work by Leitch, et al., proved that in a turbofan engine, the first stage fan face distortion was significantly reduced due to trailing edge blowing [Leitch, et al., 2000]. This was achieved using TEB with a set of four stator vanes, for simplicity. The adverse pressures acting on the fan were clearly reduced, using less than 1% of the total mass flow.

Kozak, 2000, investigated the effectiveness of TEB for subsonic fan speeds. The fan speed investigated was 11000 rpm, corresponding to 79% of the maximum engine speed. It was shown that TEB was effective in reducing the total pressure deficit in the time averaged IGV wake, which was altered by wake-potential interactions. The total pressure loss coefficient was reduced by 91.2%, compared to the case without TEB. Furthermore, the unsteadiness in the wake region that existed without TEB was effectively removed. The TEB was shown to produce a uniform pressure distribution in the filled wake. This study was the first to investigate the effect of TEB, upstream of the rotor, with an axial spacing that is typical of modern engines [Kozak, 2000, Dissertation].
The total mass flow necessary to completely fill the wakes was 0.8%, which falls under the limit set by engine design engineers.

1.2.4 Engine Component Fatigue Analysis

Extensive research has been completed in determining the fatigue life of engine components, when they are subject to high cycle fatigue (HCF) loading. HCF failures are commonly caused by fatigue loading on materials with some other mode of damage accumulation. A few of the other types of damage that can lead to crack initiation and propagation are low cycle fatigue (LCF), foreign object damage (FOD), and material fretting [Nicholas, 1999]. In general, the larger the crack size, a lesser amount of time or a lower stress amplitude is required to propagate the flaw to failure.

Low cycle fatigue occurs due to changes in engine operating speeds. Low cycle fatigue differs from high cycle fatigue, in that LCF occurs during takeoff, landing, and other major power expenditures, while HCF occurs due to vibrating components in flight. LCF cracks develop early in the fatigue lifetime of a material, and methods exist to detect the presence of such cracks. HCF, on the other hand, requires a large fraction of the material fatigue life before initiation to a detectable size occurs. The longer lifetime percentage required for HCF crack initialization means that there is a shorter percentage of lifetime in which the crack can propagate, meaning that failure may occur before the crack is even detected by routine inspection. In many cases, LCF will not produce failure, but it can lower the stresses required for HCF crack initiation and growth [Nicholas, 1999].

Foreign object damage is another source of defects that may cause initialization of HCF. FOD occurs when an object is sucked into the rotating blades, impacting the leading edge or another part of the blade surface. The amount of damage done by FOD is very dependent on the object angle of incidence. The least amount of damage occurs when the object hits with a zero degree angle of incidence, in relation to the blade chord. The highest amount of damage has been shown to occur in the region of an angle of incidence of 30 degrees [Nicholas, 1999].

Figure 1.3 shows a schematic of typical LCF and HCF loading present on engine rotor blades.
Typically, LCF conditions are characterized by a lower stress ratio ($S_{\text{min}}/S_{\text{max}}$) than HCF conditions. However, for each cycle of LCF, there are between 100 and 10000 cycles of HCF [Larsen, et al., 1997]. This means that if the crack has reached the size of the HCF threshold crack length, for a given applied stress amplitude, the crack will propagate and failure will occur quickly due to HCF [Larsen, et al., 1997].

Crack propagation for a constant amplitude fatigue can be characterized by equation 1.2 [Gemma, et al., 1979].

$$\frac{da}{dN} = Af(R)(\Delta K)^n$$

In equation 1.2, A and $n$ are material constants, $f(R)$ is a function which models the R-ratio dependence, and $\Delta K$ is the range of the stress intensity factor. Equation 1.3 shows the general equation in determining the stress intensity factor.

$$K = \sigma \sqrt{\pi \cdot aF}$$

In equation 1.3, $\sigma$ is the applied load, $a$ is the crack length, and F is a function of crack geometry and loading, and has been determined empirically for many configurations [Gemma, et al., 1979]. In most materials, failure will occur by fracture if the applied stress intensity factor range is greater than the maximum stress intensity factor, as shown in equation 1.4.
if $\Delta K > K_{\text{max}} \rightarrow \text{fracture}$ \hfill (1.4)

However, titanium does not have an endurance limit, so crack propagation in titanium is determined by the threshold stress intensity factor range. To calculate the fatigue life, it is necessary to predict crack growth cycle by cycle. In general, crack propagation in titanium will occur if the applied stress intensity factor range is greater than the threshold stress intensity range, as in equation 1.5.

if $\Delta K > \Delta K_{th} \rightarrow \text{crack propagation}$ \hfill (1.5)

The approach taken with this research will involve using such a software that can make cycle-by-cycle predictions of crack growth using the equations outlined above.

The fatigue life of the rotor blades in the case of TEB will be predicted using the life-prediction code FASTRAN 3.8. FASTRAN 3.8 was created by Newman, et al., 1998, and was a modified version of FASTRAN II. FASTRAN II was used by Newman, et al., 1996, to predict crack growth for engine disc materials. The predictions were made using 3 different materials, and a variety of loading cases. Data were provided under constant-amplitude loading, with stress ratios of 0.1 and 0.7, and under a repeated spike overload sequence. Data were also provided under complex Turbistan sequences. Turbistan is a variable-amplitude loading sequence that is made up of 100 individual flights, with an average of 77 cycles per flight. The code was used to predict the number of cycles required to grow a crack from a specified initial size to a specified final size. The data obtained were compared to experimental data. For the constant amplitude loading case, the FASTRAN II model was able to predict crack-growth to within 30% of the test data. For the repeated spike overload conditions, the predicted lives to test lives ratio was between 0.8 and 1.0, showing excellent agreement. The predicted results agreed with the test data with an overall mean ratio of 1.01, and a standard deviation of 0.31, for all materials, crack configurations, and loading conditions [Newman, et al., 1996]. This shows that the FASTRAN II code is an adequate means for predicting fatigue life.

1.3 Objectives of Current Investigation

The main objective of the investigation presented in this work is to complete preliminary steps in modeling trailing edge blowing using CFD. It is unrealistic to assume that CFD is a valid means for modeling TEB without running test cases to
support the validity of the results. After running the test cases, an attempt will be made to model stator-rotor interactions, and obtain time-dependent rotor blade stress spectra. The first step in the process is acquiring the rotor surface static pressure data using CFD analysis. The data found using CFD must then be converted to a stress history, over the surface of the rotor blade, as a function of time. The rotor blades will be analyzed as simple beams. The stress analyses will yield stress spectra that are time dependent. Different methods of modeling stator-rotor interaction will be examined. It is hoped that a consistent method of modeling stator-rotor interaction will be found, such that future investigators can start at that point.

The objective of the work following that described in this thesis is to complete the preliminary steps in determining if there may be optimal level of trailing edge blowing (TEB) necessary in the F109 turbofan engine, to obtain a maximum rotor blade fatigue life, while still meeting the requirements set by the U.S. Air Force. Much more work will be necessary to achieve this goal, including further CFD work as well as fatigue life prediction work. The spectral data acquired using CFD and the following stress analysis will be imported into the FASTRAN 3.8 fatigue life prediction code, and the fatigue life for each case can then be predicted. It is beyond the scope of this investigation, but should be the objective of future investigations, to simulate various levels of wake filling, and to prove that less than a 100% level of wake filling will achieve an improvement in blade fatigue life. It is desired that future studies will yield an estimate of the optimal level of wake filling that can be proven experimentally, when a method doing so is found. The current techniques used for experimentally measuring rotor surface pressures, which involve using slip rings to allow transducers to be embedded in the rotors, provide inaccurate results with high levels of error. Thus, for an optimal level of wake-filling to be fully investigated, a reliable method of measuring rotor surface pressures must be found.
2. Software Validation

Determining the optimal level of trailing edge blowing was not possible experimentally, because the engine test setup was not available. Thus, it was concluded that computational fluid dynamics (CFD) should be used, to investigate the effects of different levels of trailing edge blowing (TEB). It was assumed by the current researcher that the resources present on campus would be sufficient in modeling TEB, because the available software is accepted for use in the aerospace industry, and can be used to model many different types of flow. To demonstrate that CFD was a valid method in predicting rotor blade surface pressures, it was necessary to compare CFD data to experimental data of surface pressure coefficients. The first section of this chapter describes the main software packages used in the simulations. The second and third parts of this chapter detail the test cases simulated in order to validate the use of CFD. The first case simulated was that of a NACA 0012 wing mounted vertically in a wind tunnel. The second case simulated was of wind tunnel tests performed on a series of flat plates, at a laminar Reynolds number, in the RIT wind tunnel laboratory.

2.1 Software Utilities

The following section describes the software used in modeling and simulating two different validation cases. The same software packages were also used to model the stator-rotor interaction, and the variation in trailing edge blowing.

2.1.1 GAMBIT

GAMBIT is a modeling and meshing software package that is distributed by Fluent, Inc. GAMBIT allows for both 2-d and 3-d geometry creation. GAMBIT provides a graphical user interface, allowing the user to create geometries using a variety of methods. Much of the geometry created was done so using a bottom-up technique, meaning that points were created first, which were then connected by lines; the lines were then stitched to form faces, and the faces could then be meshed. The density of the face mesh can be controlled by meshing edges before meshing the face, as was done in all cases. GAMBIT also offers the option of adding a boundary layer to wall surfaces,
which was utilized in the flat plate validation cases. GAMBIT allows for the assignment of a CFD solver, which in this case, was Fluent 5/6. Assigning the solver then allows the user to assign boundary conditions to edges or faces in GAMBIT. All of the cases presented were two-dimensional, and thus the boundary conditions were all assigned to edges.

2.1.2 Fluent

The data presented in this study were acquired using Fluent ©, a computational fluid dynamics (CFD) package. The Fluent package provides a wide range of capabilities for modeling many different types of flow, and it combines a broad range of mathematical models with the ability to model complex geometries. It also includes a C-compiler, allowing the user to import user-defined functions written in the C programming language that can be assigned to different boundary conditions or fluid properties. For all flows, Fluent solves conservation equations for mass and momentum. When applicable, Fluent also solves the conservation equation for energy.

2.1.2.1 Mass Conservation Equation

The equation solved in Fluent for the conservation of mass is shown in equation 2.1.

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = S_m
\]  

(2.1)

This equation is valid for compressible as well as incompressible flows. The term on the right side of the equation is the mass added, which for all of the validation and test cases, was equal to zero.

2.1.2.2 Momentum Conservation Equation

The equation used by Fluent for conservation of momentum in a non-accelerating reference frame is shown in equation 2.2.

\[
\frac{\partial}{\partial t} (\rho \vec{v}) + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p + \nabla \cdot \tau + \rho \vec{g} + \vec{F}
\]

(2.2)

The quantity \( p \) is equal to the static pressure, \( \tau \) is the stress tensor, and \( \rho \vec{g} \) and \( \vec{F} \) are the gravitational body and external body forces. The stress tensor is represented by
equation 2.3, in which $\mu$ is the molecular viscosity, $I$ is the unit tensor, and the second term on the right hand side is the effect of volume dilation.

$$
\tau = \mu \left( \nabla \tilde{v} + \nabla \tilde{v}^T - \frac{2}{3} \nabla \cdot \tilde{v} I \right)
$$

(2.3)

The discretization process used by Fluent is the finite-volume method. The control-volume-based technique that the solver uses first divides the domain into discrete control volumes using a computational grid that is built outside of Fluent and imported. Integration of the governing equations is then performed at each discrete volume, to construct algebraic equations for the discrete dependent variables, or unknowns, such as pressure and velocity. The discretized equations are then linearized, and solved to yield updated values of the dependent variables. This process is iterated until the solution is deemed converged, based on the decrease in magnitude of the scaled residuals for the equations of flow.

2.2 Validation Case 1 - NACA 0012 Airfoil

The use of the software was first validated by simulating a wind tunnel experiment in which the pressure coefficients were found by measuring surface pressure values on a NACA 0012 airfoil, at various dynamic pressures and angles of attack [Applin, 1995]. The objective of the CFD simulation was to match the 2-d data obtained to that of the experimental results, at a point about half-way along the span of the wing. This distance along the span was chosen to be 60% span, assuming that end effects due to the wind tunnel mounting could be neglected and data at this location would be the closest to that of a two-dimensional airfoil. It was necessary to simulate results using a 2-dimensional case, because the data taken in the study of trailing edge blowing was 2-dimensional. The experimental data was in the form of pressure coefficient vs. % chord.

2.2.1 Experimental Data

The first verification case performed was that of a NACA 0012 airfoil. The experimental data were gathered in the Langley 14- by 22-Foot Subsonic Tunnel [Applin, 1995].
2.2.1.1 Experimental Test Setup

An unswept, semi-span wing model incorporating the NACA 0012 airfoil was mounted vertically in the Langley 14- by 22-Foot Subsonic Tunnel, protruding from the bottom. The Langley 14- by 22-Foot Subsonic Tunnel is a closed, single return, atmospheric wind tunnel. The test section is 14.50 feet high by 21.75 feet wide by 50.00 feet long. The test-section dynamic pressure is variable from 0 to 144 psf. The wing was rectangular in shape, with a semi-span length of 116.1 inches, and a chord length of 39.37 inches. The wing is shown as mounted in the tunnel in Figure 2.1.

![Figure 2.1: NACA 0012 wing setup](image)

Pressure measurements on both the upper and lower surfaces of the airfoil were obtained using an electronically scanned pressure (ESP) system. Every port contained a 720 psf range pressure transducer, with a Manufacturer's stated accuracy of ±0.72 psf. The ESP system scans through the transducers at rates up to 20 kHz, ensuring that all pressure data is acquired at nearly the same instant. The data were passed to the tunnel data acquisition system at the rate of 1 sample per second. The system then averaged 20 of these samples for each point [Applin, 1995].
2.2.1.2 Experimental Results

Data were gathered for free-stream dynamic pressures of 15 to 60 psf, with corresponding Reynolds numbers of $2.36 \times 10^6$ to $4.71 \times 10^6$, based on the reference wing chord. Mach numbers corresponding to the above free stream conditions were 0.10 to 0.20. The angle of attack was varied from $-4^\circ$ to $20^\circ$ in 2 degree increments. The data were presented in tabular form as pressure coefficient ($C_p$) versus non-dimensional chord location ($x/c$). The chord locations were non-dimensionalized using the reference chord length of 39.37 inches.

2.2.2 CFD Simulation

The following section presents the CFD simulation of the NACA 0012 airfoil wind tunnel test. The experimental data taken was for a three-dimensional wing, while the CFD simulation was of a two dimensional airfoil. To compare data, it was assumed that near the mid-span region of the wing, the pressure coefficient would be closest to that of a two dimensional airfoil. It was also assumed that because of the size of the test section of the wind tunnel, the walls of the tunnel would not have an effect on the surface pressure coefficient.

2.2.2.1 GAMBIT Modeling

The mesh used for the simulation of the NACA 0012 test in the Langley Subsonic Tunnel was built using the software package, GAMBIT, with the solver set as Fluent 5/6. Setting the solver as such allows the appropriate boundary conditions to be defined. The geometry was built by first importing the vertex data, as obtained from the UIUC Airfoil Database [http://www.aae.uiuc.edu/m-selig/ads/coord_database.html]. This data included 66 points for each of the top and bottom surfaces of the airfoil. The airfoil is shown in Figure 2.2.
The faces of the geometry created are shown in Figure 2.3. The boundary conditions for each face were specified as shown on the diagram. The outer boundaries were specified as pressure-far-field boundaries, allowing a Mach number to be specified as the velocity condition. The airfoil was specified as a wall boundary, and was broken into an upper and lower surface, to allow for plotting of both the upper and lower surface pressure coefficients. The interior boundary condition was assigned automatically when the mesh was imported into Fluent, and meant that these edges were internal to the flow, and would be treated as a continuous boundary between adjacent cells.
The inlet was initially modeled as parabolic, but was modified such that it was a vertical edge. This was done so that the velocity field would be applied across a vertical inlet, and corrections would not have to be made for a curved inlet. This was the desired method of modeling the rotor for the TEB case, to mimic the actual test environment as closely as possible. The inlet was placed at a distance of 5 chord-lengths upstream of the leading edge of the airfoil, and was assigned a pressure-far-field boundary condition. The pressure-far-field condition was a valid boundary condition because of the assumption that the tunnel walls did not affect the data. The outlet of the flow was placed at a distance of 5 chord-lengths aft of the trailing edge of the airfoil, and was also assigned a pressure-far-field boundary condition. The upper and lower boundaries were placed 5 chord-lengths above and below the airfoil chord line, and were assigned pressure-far-field boundary conditions.

The edges radiating from the airfoil surface were meshed with a grading of 1.1, and an interval count of 100. This technique of meshing is shown in Figure 2.4. This was to allow for a denser mesh near the airfoil, because the most important results were occurring close to the airfoil surface.

![Figure 2.4: Edge meshing near airfoil](image)

The faces were all meshed using the quad-map scheme, excluding the two triangular faces at the inlet, which were meshed using a tri-pave scheme. The quad-map scheme is a structured mesh, and usually allows for faster convergence and a more
accurate solution, which was the reason for its selection. The tri-pave scheme is an unstructured mesh, which does not have any requirements of the geometry, making it suitable for meshing areas of complex geometry, and areas that are not vital to the solution accuracy. The mesh had a total of 24324 cells. This mesh was used for every case of the NACA 0012 model in Fluent. The mesh used in the simulation is shown in Figure 2.5.

![Figure 2.5: NACA 0012 mesh](image)

### 2.2.2.2 Fluent Setup

The mesh was imported into Fluent, and then scaled to convert the units to inches. The simulations were run using a coupled-implicit solver. This type of solver simultaneously solves all finite difference equations. Governing equations for scalars are solved sequentially. The coupled solver operates as is shown in the schematic diagram, Figure 2.6.
The Spalart-Allmaras viscous model was used. This model uses one equation to solve for viscosity, and is often suggested for use in aerospace applications, including external flow. It can be used with relatively crude simulations on coarse meshes where accurate turbulent flow computations are not critical. The density was set to be determined by the ideal gas law, in order to use the pressure-far-field boundaries.

The solutions were monitored by plotting the residuals for continuity, velocity, and energy. The solutions were considered converged when the residuals had all dropped by at least 2 orders of magnitude, and had stabilized. The convergence of the solutions was also verified by checking the contours of the entropy in the system. The existence of negative entropy in the system would indicate a problem with the simulation, because negative entropy is physically impossible. The simulations were found not to have negative entropy present. Furthermore, the areas of highest entropy were within the boundary layer, as well as just aft of the trailing edge, which is also physically valid.

The Mach number specified was assumed to be the same for all boundaries. The angle of attack of the airfoil was varied by specifying the horizontal and vertical components of the Mach number. Data were taken and compared for Mach numbers of 0.10 and 0.20, which corresponded to the dynamic pressures used in the wind tunnel experiment, assuming standard air density. Data was taken for angles of attack ranging between $-4^\circ$ and $20^\circ$, in increments of $4^\circ$.

### 2.2.3 Comparison Results

The data plots from the wind tunnel experiments were compared to the data plots obtained from the Fluent simulations. The surface pressure coefficient data obtained
using Fluent were then plotted using Microsoft Excel ©, and was overlaid atop the data from the NASA wind tunnel. The experimental data existed in tabular form only, and thus it was difficult to compare to the CFD data. This was eventually achieved by overlaying the CFD data on top of the plots of the experimental data, and scaling the plots so that the tick marks on the axes of the CFD data aligned with the tick marks on the axes of the experimental data plots. This method was not extremely accurate, but was deemed as valid because the trends of the surface pressure coefficient were assumed more important for validation purposes than the actual values. This assumption was made mainly because the experimental data was for a 3-d wing, and the CFD data was for a 2-d airfoil, so it was not possible to guarantee matching results. An example of the data comparison can be seen in Figure 2.7.

![Figure 2.7: Cp comparison for angle of attack = 8](image)

The complete plots of the CFD and experimental data can be found in Appendix B. It can be seen from Figure 2.7 that the trends of the two sets of data match closely. The major difference in plots is on the upper surface at the leading edge of the airfoil. Some of this error was assumed to be attributed to the fact that the CFD simulation was for a 2-d airfoil, while the experimental data was for a 3-d wing. It has also been found through past experience that Fluent has a tendency to over-predict the pressure coefficient for airfoil simulations.

Overall, the CFD simulation of the NACA 0012 wing demonstrated proficiency in modeling a 2-dimensional system. The pressure coefficient trends obtained from the CFD matched those obtained experimentally for the ranges tested.
2.3 Validation Case 2 - Flat Plate Simulation

The second validation case presented was the modeling of the surface pressure coefficient on a flat plate, at a low Reynolds number. The experimental data were gathered in the RIT wind tunnel. Five different plates were tested, and each plate had 12 or 13 spanwise pressure taps to allow for surface pressure measurements. The chordwise location of the taps differed for each plate, such that a complete static pressure profile could be measured. Each plate had pressure taps at two different chord-wise distances, giving a total of 10 different chord locations for static pressure to be measured. Each plate was tested from an angle of attack of -4 to 20 degrees, in increments of 4 degrees. For the first part of the experiment, the plates were mounted in the tunnel and tests were run at a Mach number of approximately 0.05. The surface pressure data for the upper and lower surfaces of each plate were recorded, and then compiled to obtain a plot of pressure coefficient vs. percent chord-length, for each angle of attack.

For the second part of the experiment, a wake generator, made up of 3 circular cylinders, mounted between pine boards, was placed 10 cylinder diameters upstream of the flat plates. The upstream placement of the cylinders resulted in a non-uniform velocity profile, traveling across the plate. The surface pressures were again recorded and compiled to obtain plots of the surface pressure coefficient, and the differences between the two cases were examined. The goal of the second experiment involving the non-uniform velocity profile was to gain familiarity with writing code for, and implementing user defined boundary conditions in Fluent. This experiment was necessary because the inlet velocity profile for the TEB cases were modeled as non-uniform, user-defined functions.

2.3.1 Experimental Apparatus

The experimental data were gathered in the RIT subsonic wind tunnel facility. The objects tested were five different flat plates, built to allow for surface pressure measurements at 10 different chord locations. The plates were mounted in the test section of the tunnel. The pressure measurements were made using a 10 torr differential pressure transducer, which was connected to a digital readout. The following section details the experimental apparatus used in the testing.
2.3.1.1 Flat Plates

The objects tested were aluminum flat plates, measuring 8 inches x 8 inches, with a thickness of 0.160 inches. There were two mounting holes at the trailing edge of each plate, mid-span, to allow the plate to be mounted to a sting arm in the test section of the wind tunnel. The plates had elliptical-shaped leading and trailing edges, and were designed and manufactured by Julie Jones, at RIT.

Each plate, except for the plate with the furthest downstream pressure taps, had 13 through holes drilled, 1/32” in diameter, at different spanwise and chordwise locations. The plate with the furthest downstream tap locations only had 12 spanwise holes, because of the mounting holes. One-inch lengths of copper tubing were inserted through each hole, such that the tubing was flush with one side of the plate. The copper tubes were held in place using epoxy. The copper tubes allowed for the connecting of Tygon® flexible tubing, which ran out of the wind tunnel to a switch box. A diagram of Plate 1 can be seen in Figure 2.8. The drawing is not to scale, but is meant to illustrate the method used in designing the plates, and the method in which the copper tubes were embedded.

![Figure 2.8: Schematic diagram of Plate 1](image)

The spanwise and chordwise locations of the copper tubes for each plate are shown in Table 2.1.
### Table 2.1 Plate pressure tap locations

<table>
<thead>
<tr>
<th>Point</th>
<th>( y/b )</th>
<th>Plate 1</th>
<th>Plate 2</th>
<th>Plate 3</th>
<th>Plate 4</th>
<th>Plate 5</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>.025</td>
<td>.0625</td>
<td>.250</td>
<td>450</td>
<td>.650</td>
<td>.850</td>
</tr>
<tr>
<td>2</td>
<td>.075</td>
<td>.0625</td>
<td>.250</td>
<td>450</td>
<td>.650</td>
<td>.850</td>
</tr>
<tr>
<td>3</td>
<td>.125</td>
<td>.0625</td>
<td>.250</td>
<td>450</td>
<td>.650</td>
<td>.850</td>
</tr>
<tr>
<td>4</td>
<td>.200</td>
<td>.0625</td>
<td>.250</td>
<td>450</td>
<td>.650</td>
<td>.850</td>
</tr>
<tr>
<td>5</td>
<td>.300</td>
<td>.0625</td>
<td>.250</td>
<td>450</td>
<td>.650</td>
<td>.850</td>
</tr>
<tr>
<td>6</td>
<td>.400</td>
<td>.0625</td>
<td>.250</td>
<td>450</td>
<td>.650</td>
<td>.850</td>
</tr>
<tr>
<td>7</td>
<td>.500</td>
<td>.0625</td>
<td>.250</td>
<td>450</td>
<td>.650</td>
<td>.850</td>
</tr>
<tr>
<td>8</td>
<td>.600</td>
<td>.150</td>
<td>.350</td>
<td>.550</td>
<td>.750</td>
<td>.9375</td>
</tr>
<tr>
<td>9</td>
<td>.700</td>
<td>.150</td>
<td>.350</td>
<td>.550</td>
<td>.750</td>
<td>.9375</td>
</tr>
<tr>
<td>10</td>
<td>.800</td>
<td>.150</td>
<td>.350</td>
<td>.550</td>
<td>.750</td>
<td>.9375</td>
</tr>
<tr>
<td>11</td>
<td>.875</td>
<td>.150</td>
<td>.350</td>
<td>.550</td>
<td>.750</td>
<td>.9375</td>
</tr>
<tr>
<td>12</td>
<td>.925</td>
<td>.150</td>
<td>.350</td>
<td>.550</td>
<td>.750</td>
<td>.9375</td>
</tr>
<tr>
<td>13</td>
<td>.975</td>
<td>.150</td>
<td>.350</td>
<td>.550</td>
<td>.750</td>
<td>.9375</td>
</tr>
</tbody>
</table>

The Tygon® tubing for the flat plate pressure taps were connected to a set of switches, which were labeled corresponding to the tap number. There was one tube leaving the switch box that could be connected to the front of the pressure transducer. Flipping each switch, one at a time, allowed the pressure for each pressure tap to be read. The switch box is shown in Figure 2.9.

![Figure 2.9: Switch box](image-url)
2.3.1.2 RIT Wind Tunnel

The RIT wind tunnel is a closed-circuit, low speed wind tunnel. The test section is approximately 4 ft in length x 2 ft tall x 2 ft wide. A diagram of the wind tunnel can be seen in Figure 2.10.

![Diagram of RIT Wind Tunnel](image)

**Figure 2.10: Wind tunnel diagram**

The wind tunnel was equipped with a pitot-static tube, mounted to a series of servo-motors, allowing 3-dimensional linear travel. Figure 2.11 shows the pitot-tube as mounted in the wind tunnel for testing, protruding through the top wall, with the tip of the probe facing upstream.
The tunnel test section was equipped with a mounting arm, to which the flat plates were bolted. Figure 2.12 shows a test plate mounted to the arm inside the wind tunnel test section.

The angle of attack of the plates was variable. The angle of attack was changed by turning a handle beneath the test section of the wind tunnel, which was mounted to a lead screw. Turning the lead screw caused the arm to rotate, which varied the angle of attack of the object mounted to the sting arm. The angle of attack was viewable by an electronic readout, and the angle was variable between $-20.0$ degrees and $20.0$ degrees, with minimum variation of $0.1$ degrees.
2.3.1.3 Pressure Transducer

The pressure transducer used for the experiments was manufactured by MKS Instruments, Inc. The transducer was of the differential type, meaning that the measurement obtained was of the difference in pressure between sides of the internal membrane. The transducer was wired to an MKS Instruments PDR-D readout, which displayed a value in millivolts, that was proportional to the measured pressure. The transducer had to be calibrated daily to determine the scaling factor necessary to convert the values on the readout to the actual measured pressure. The transducer had a maximum error as specified by the manufacturer of 0.1% of the total allowable pressure. It was necessary to calibrate the transducer every day to ensure to obtain a calibration slope. The calibration slope allowed the number read on the digital readout to be converted to a static pressure value.

2.3.1.4 Velocity Inlet Generator

For the second phase of the wind tunnel experiments, a piece of equipment was built that could be mounted in the inlet of the wind tunnel, to cause a variable velocity field over the plate. The apparatus was constructed of wooden dowels, wooden boards, L-brackets, and wood screws. The wooden dowels were 0.50 inches in diameter, and were spaced 2 inches apart. Figure 2.13 shows a diagram of the apparatus.

Figure 2.13: Diagram of velocity inlet generator
The velocity inlet generator was attached to the upper and lower walls of the wind tunnel using screws and L-brackets. Figure 2.14 shows a picture of the velocity inlet generator, mounted in the wind tunnel.

Figure 2.14: Velocity Inlet Generator

2.3.2 Experimental Method

The tests were run at a Reynolds number of approximately 235000, which corresponds to a Mach number of 0.05. The NACA 0012 validation cases were compared to experimental data at Mach numbers between 0.10 and 0.20, and thus it was determined that the flat plate wind tunnel tests should be conducted at similar values. Initially, the tests were attempted at a Mach number of 0.20; however, the wind tunnel was unable to achieve such a high speed. The tests were then attempted at a Mach number of 0.10, but the plates experienced fluttering at higher angles of attack, presumably because there were only two bolts mounting the plates to the wind tunnel arm. Thus, it was determined that a Mach number of 0.05 would be used. This yielded a Reynolds number that fell within the laminar flow range.

2.3.2.1 Uniform Velocity Inlet

The first step in obtaining the surface pressure coefficients on the flat plate was to record the atmospheric pressure, obtained from a barometer in the wind tunnel laboratory.
The data was recorded in kilopascals. The temperature was also recorded, using a probe that was placed in the tunnel test section, and connected to a digital readout. The next step was to calibrate the 10 torr MKS Instruments differential pressure transducer, using a Fluke 718 pressure calibrator. The calibrator was used to apply known pressures, in kilopascals, in small increments to the transducer, and the corresponding value was displayed on the MKS Instruments readout. The tube on the calibrator was attached to the front inlet of the pressure transducer. The other inlet of the transducer was left open, which meant that the transducer was yielding the difference between the pressure applied by the calibrator, and the atmospheric pressure. Pressure was slowly applied to the transducer using a dial, in increments of approximately 100 Pa. It was determined that the maximum safe pressure that could be applied to the transducer was 1 kPa, which corresponded to approximately 7.5 torr, which was 25% less than the maximum allowable value for which the transducer was rated. The pressure applied and the corresponding number on the readout was recorded for 6-8 different amounts of pressure. The data were then plotted as a function of pressure vs. readout value. The slope of the line plotted was equal to the factor that the readout had to be scaled by, in order to obtain the actual pressure, in kilopascals. It was assumed that the same scaling factor could be used to scale the readout data when the wind tunnel tests were run.

The tunnel velocity was set by calculating the necessary dynamic pressure required, corresponding to the desired free stream velocity. The dynamic pressure was set by attaching the tubes from both the static pressure port and the total pressure port to each side of the 10 torr pressure transducer.

From equation 2.4, the incompressible form of Bernoulli’s equation, it can be seen that the dynamic pressure is the difference between the total and the static pressure.

\[ p_\infty = p + \frac{\rho U_\infty^2}{2} \]  \hspace{1cm} (2.4)

In equation 2.4, \( p_\infty \) is the total pressure, \( p \) with the subscript infinity is the free stream static pressure, \( \rho \) is the free stream density, and \( U \) with the subscript infinity is the free stream velocity. The second term on the right side of the equal sign is the dynamic pressure. The value displayed on the readout when both Tygon® tubes were attached to the transducer corresponded to the dynamic pressure, because the transducer measured
the differential pressure. For this testing, the tests were all run at a Mach number of close to 0.05. The dynamic pressure necessary for a Mach number of 0.05 at standard conditions was calculated, and the necessary digital readout value for the dynamic pressure was also calculated, using the calibration curve discussed previously. The values for velocity, density, and Mach number were then corrected using the atmospheric temperature and pressure in the lab. These corrected values were later used in the pressure coefficient calculations, and the CFD simulations.

For the uniform velocity inlet data, the wind tunnel was first set to the correct Reynolds number, using the procedure described previously. The tube was then disconnected from the front of the transducer, which corresponded to the total pressure port on the transducer. This allowed for the measurement of the free stream static pressure. The total tunnel pressure could then be found by adding the static pressure to the dynamic pressure. The tube was then removed from the back of the transducer. The tube from the switch box was then attached to the front of the transducer, which allowed for the measurement of the surface pressure over the plate. Data were taken for every port on all five plates, even though the data were only compiled at one spanwise location. The experiments were quasi-static, in that the transducer values were allowed to stabilize before switching locations. The line losses were assumed to be negligible.

2.3.2.2 Non-Uniform Velocity Inlet

The first step of gathering the data in the non-uniform velocity inlet case was to measure the inlet velocity profile resulting from the velocity inlet generator. This was done using the pitot-static pressure probe in combination with the vertically actuating linear motor. The velocity inlet generator was mounted 10 cylinder diameters upstream of the leading edge of the plate, or 5 inches. The plate was then removed, and the probe of the pitot-static tube was placed 5 cylinder diameters downstream of the axial centerlines of the cylinders.

To measure the velocity profile generated by the cylinders, the plastic tubing from both the static and total pressure taps were attached to the transducer. This resulted in the measurement of free stream dynamic pressure, which could then be converted to the free stream velocity. First, the pitot-static probe was moved vertically to a point above the
cylinders, such that there was no variation in the digital readout. This would indicate that the probe was above any influence of the cylinders. Next, using the linear motor, the pitot-static probe was traversed vertically, in increments of 250 steps, or 1.53 mm, between measuring locations. The dynamic pressure was then scaled using the calibration curve, and converted to velocity. The measured velocity profile can be seen in Figure 2.15.

![Inlet Velocity Profile](image)

**Figure 2.15: Measured inlet velocity profile**

The gray circles in the figure represent the vertical position of the cylinders. The velocity profile looks as expected. Between the cylinders, the velocity is close to the free stream of the tunnel without the velocity profile generator. The areas of lowest velocity are directly downstream of the cylinders, as would also be expected. Because of the turbulence caused by the cylinder wakes, the pressure measurements of the velocity profile fluctuated +/- 0.01 on the digital readout, which corresponded to approximately +/- 0.013 kilopascals. This accounts for the choppiness seen in the inlet velocity profile.

After the velocity profile was measured, the plates were tested again, to find the influence of a non-uniform inlet velocity profile on the surface pressure response. The process used was the same as for the clean inlet case, as described in the previous section.

### 2.3.3 CFD Simulation

The next step of the process was performing a CFD analysis of the flat plate, and comparing the results to the experimental results. The following section will describe the geometry and problem setup for the CFD simulation.
2.3.3.1 GAMBIT Modeling

A series of meshes were built in GAMBIT, and then used for the flat plate wind tunnel simulation. To obtain results that were as accurate as possible, the meshes were built with the same dimensions as the wind tunnel test section. However, the sting arm that the plates were mounted to in the wind tunnel were not modeled, because it was assumed that it did not have an effect on the plate surface pressures.

For the uniform velocity inlet profile, seven different meshes were built. A different mesh was built for each different angle of attack, because the leading edge of the plate in the wind tunnel varied in relation to the inlet, with angle of attack. Figure 2.16 shows the geometry modeled for an angle of attack of 8°, as well as the dimensions of the wind tunnel test section, and numbered edges.

![Figure 2.16: GAMBIT geometry, uniform inlet, AOA=8](image)

The geometries for all of the angles of attack were modeled in a similar manner as shown in Figure 2.16. The leading and trailing edges of the plate were modeled as elliptic. The upper and lower edges of the geometry were set to wall boundary conditions. The inlet was set as a velocity inlet, and the outlet set as a pressure outlet. The upper and lower surfaces of the plate were both set to wall boundary conditions.

The edge meshing scheme for the geometry shown in Figure 2.16 can be seen in Table 2.2.
Table 2.2: Uniform inlet, AOA=8 edge meshing

<table>
<thead>
<tr>
<th>Edge</th>
<th>Interval Count</th>
<th>Left Grading</th>
<th>Right Grading</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>12</td>
<td>1</td>
<td>N/A</td>
</tr>
<tr>
<td>2</td>
<td>22</td>
<td>1</td>
<td>N/A</td>
</tr>
<tr>
<td>3</td>
<td>50</td>
<td>1</td>
<td>1.1</td>
</tr>
<tr>
<td>4</td>
<td>90</td>
<td>1</td>
<td>N/A</td>
</tr>
<tr>
<td>5</td>
<td>90</td>
<td>1</td>
<td>N/A</td>
</tr>
<tr>
<td>6</td>
<td>60</td>
<td>1.1</td>
<td>1</td>
</tr>
<tr>
<td>7</td>
<td>14</td>
<td>1</td>
<td>N/A</td>
</tr>
<tr>
<td>8</td>
<td>20</td>
<td>1</td>
<td>N/A</td>
</tr>
<tr>
<td>9</td>
<td>12</td>
<td>1.15</td>
<td>N/A</td>
</tr>
<tr>
<td>10</td>
<td>12</td>
<td>1.15</td>
<td>N/A</td>
</tr>
<tr>
<td>11</td>
<td>12</td>
<td>N/A</td>
<td>1.15</td>
</tr>
<tr>
<td>12</td>
<td>12</td>
<td>N/A</td>
<td>1.15</td>
</tr>
<tr>
<td>13</td>
<td>80</td>
<td>1</td>
<td>N/A</td>
</tr>
<tr>
<td>14</td>
<td>80</td>
<td>1</td>
<td>N/A</td>
</tr>
</tbody>
</table>

The term N/A in Table 2.2 was used when the mesh was only single sided. It was desirable to have a denser mesh near the surface of the flat plate, which was done by the edge mesh grading. A boundary layer was also added to the surface of the plate. A close-up of the edge mesh at the leading edge can be seen in Figure 2.17.

![Figure 2.17: Leading edge mesh, uniform inlet, AOA=8](image)

The faces were meshed using a quad-pave scheme. This meshing scheme provided the least amount of skewness, and yielded a dense mesh toward the surface of the plate. The final mesh can be seen in Figure 2.18. The mesh for an angle of attack of 8 degrees had a total of 8869 elements, with only 2.10% of elements having a skewness of above 0.4. The meshes for the other angles of attack were created in a very similar
fashion. All were meshed using the quad-pave scheme, to reduce skewness and concentrate elements toward the plate surface.

![Figure 2.18: Mesh, uniform inlet, AOA=8](image)

The meshes used for the non-uniform inlet velocity cases were slightly different than those used for the uniform velocity inlet profile cases. For the non-uniform inlet velocity cases, the inlet of the geometry had to be moved to the location at which the pitot-static probe was placed in the wind tunnel, which was 10 cylinder diameters upstream of the plate leading edge, when the plate was at an angle of attack of zero.

The geometry for the mesh created for the plate at an angle of attack of 8° can be seen in Figure 2.19.
The geometries for all of the angles of attack for the non-uniform inlet velocity were modeled in a similar manner as shown in Figure 2.19. The boundary conditions were set the same as in the uniform inlet cases.

The edge meshing scheme for the geometry shown in Figure 2.19 can be seen in Table 2.3.

<table>
<thead>
<tr>
<th>Edge</th>
<th>Interval Count</th>
<th>Left Grading</th>
<th>Right Grading</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>20</td>
<td>1.05</td>
<td>N/A</td>
</tr>
<tr>
<td>2</td>
<td>30</td>
<td>1.05</td>
<td>N/A</td>
</tr>
<tr>
<td>3</td>
<td>16</td>
<td>N/A</td>
<td>1.15</td>
</tr>
<tr>
<td>4</td>
<td>58</td>
<td>1</td>
<td>N/A</td>
</tr>
<tr>
<td>5</td>
<td>58</td>
<td>1</td>
<td>N/A</td>
</tr>
<tr>
<td>6</td>
<td>50</td>
<td>1.15</td>
<td>1</td>
</tr>
<tr>
<td>7</td>
<td>18</td>
<td>1</td>
<td>N/A</td>
</tr>
<tr>
<td>8</td>
<td>26</td>
<td>1</td>
<td>N/A</td>
</tr>
<tr>
<td>9</td>
<td>12</td>
<td>1.15</td>
<td>N/A</td>
</tr>
<tr>
<td>10</td>
<td>12</td>
<td>1.15</td>
<td>N/A</td>
</tr>
<tr>
<td>11</td>
<td>12</td>
<td>N/A</td>
<td>1.15</td>
</tr>
<tr>
<td>12</td>
<td>12</td>
<td>N/A</td>
<td>1.15</td>
</tr>
<tr>
<td>13</td>
<td>80</td>
<td>1</td>
<td>N/A</td>
</tr>
<tr>
<td>14</td>
<td>80</td>
<td>1</td>
<td>N/A</td>
</tr>
</tbody>
</table>

Table 2.3: Non-uniform inlet, AOA=8 edge meshing
The non-uniform inlet meshes were also constructed with a boundary layer mesh near the surface. The faces were all meshed using the quad-pave meshing scheme. The mesh for an angle of attack of 8 degrees had a total of 6511 elements, with 2.29% of elements having a skewness of above 0.4. The final mesh for the non-uniform inlet velocity, and angle of attack of 8°, can be seen in Figure 2.20.

![Figure 2.20: Mesh, non-uniform inlet, AOA=8](image)

### 2.3.3.2 Fluent Setup

The meshes were imported into Fluent, and scaled to English units. For the uniform inlet velocity cases, the upper and lower boundaries were assigned as walls, as they represent the wind tunnel upper and lower walls. The inlet was assigned as a velocity inlet, with a uniform velocity of 17.15 m/s, which was found to be the average tunnel velocity during the wind tunnel testing. The outlet was assigned a pressure outlet boundary condition, with a gauge pressure of zero. The 2-d coupled implicit solver was used, and Spalart-Allmaras viscous model was enabled. Even though the flow was laminar by looking at the Reynolds number alone, the laminar solver did not yield converged results. Thus, the viscous model was chosen. As the solution progressed, the lift, drag, and moment coefficients were monitored, as were the solution residuals. The residuals were considered converged when they had dropped by at least three orders of magnitude. Convergence was achieved for all angles of attack. Cases were run for the uniform inlet velocity for angles of attack from -4° to 20°, in increments of 4°. Because
the meshes incorporated the plate angle of attack, the inlet velocity was assigned as one-dimensional, parallel to the tunnel walls.

For the non-uniform inlet velocity cases, the upper and lower boundaries were again assigned as walls, and the outlet was assigned as a pressure outlet, with a gauge pressure of zero. The inlet was assigned as a velocity inlet, but the value of the velocity was determined by a user-defined function, written in C and imported and compiled by Fluent. The user-defined function represented the velocity inlet profile that was created by the velocity inlet generator during wind tunnel testing. This function was represented by a series of cosine waves, as shown in equation 2.5.

\[ V = a_0 + 2 \sum_{i=1}^{n} a_i \cos(2\pi f_i (y - y_o) + \phi_i) \]  

(2.5)

The term \( a_0 \) represents the mean offset, and \( y_o \) represents the height along the inlet at which the non-uniform profile begins. Inside the summation, the \( 'a' \) terms represent the amplitude of the cosine wave, the \( 'f' \) terms represent the frequencies, and the \( '\phi' \) terms represent the phases. The individual data points for the inlet profile, obtained during testing, were imported into Matlab, and then a fast Fourier transform (FFT) was performed to obtain data for amplitude, frequency, and phase. The final function consisted of a mean velocity value, plus 9 cosine terms. The function was embedded within an if-statement within the block of C code. The if-statement was necessary because the cosine function only represented the velocity profile for a range in which the free stream was affected by the velocity inlet generator. The code for the if-statement and the velocity function can be seen in Figure 2.21.

```c
y0=0.41229026; /*distance from top of tunnel-shifts function*/
if (y < 0.227221)
{F_PROFILE(f,thread_nv)= 17.15;}
else if (y >0.410954)
{F_PROFILE(f,thread_nv)= 17.15;}
else
F_PROFILE(f,thread_nv)= a0+2*a1*cos(2*pi*f1*(y-y0)+p1)+
2*a2*cos(2*pi*f2*(y-y0)+p2)+2*a3*cos(2*pi*f3*(y-y0)+p3)+
2*a4*cos(2*pi*f4*(y-y0)+p4)+2*a5*cos(2*pi*f5*(y-y0)+p5)+
```
2*a6*cos(2*pi*f6*(y-y0)+p6)+2*a7*cos(2*pi*f7*(y-y0)+p7)+
2*a8*cos(2*pi*f8*(y-y0)+p8)+2*a9*cos(2*pi*f9*(y-y0)+p9);}

Figure 2.21: Inlet velocity function

The ‘a’ terms represent the amplitudes, the ‘f’ terms represent the frequencies, and the ‘p’ terms represent the phases. The y term was the height along the inlet, which made it necessary to translate the grid to make the lower left corner or the geometry equal to y=0 in the Fluent coordinate system.

The code was then imported and compiled by Fluent, and assigned as the velocity inlet boundary condition. A plot of the function in Fluent can be seen in Figure 2.22.

Figure 2.22: Velocity magnitude vs. height at inlet

The non-uniform inlet velocity cases were run for the same angles of attack as the uniform inlet velocity cases. The 2-d coupled implicit solver was again chosen, as was the Spalart-Allmaras viscous model, for the same reasons as described above. The residuals were considered converged when they had dropped by three orders of magnitude. The final step in the CFD simulation was exporting the pressure coefficient vs. chord data for the upper and lower surfaces of the flat plate.

2.3.4 Comparison Results

The experimental pressure coefficient data obtained in wind tunnel testing were compared to the pressure coefficient data obtained in the CFD simulation. The pressure coefficient was obtained from surface pressure measurements along the flat plate, at
span-wise percentage locations of 0.4 and 0.6. Because the plates were symmetric, it was assumed that the data could be combined to obtain a surface pressure vs. chord distribution for 10 points along the chord. The tunnel static pressure was subtracted from the plate surface pressure, and then divided by the tunnel dynamic pressure, to obtain the surface pressure coefficient, as shown in equation 2.6.

\[ C_p = \frac{p - p_s}{\frac{1}{2} \rho U_x^2} \quad (2.6) \]

The data obtained in Fluent were exported as data files, and were then imported into Microsoft Excel© and plotted over the plots of the experimental data. In general, the pressure coefficient values obtained using CFD had a higher magnitude than the experimental results, for both the uniform and non-uniform inlet velocity cases. Some of this error can be attributed to the 3-d nature of the experiment, versus the 2-d nature of the CFD simulation. Because this experimental work was not the main objective of the research, there was not a lot of time spent in trying to validate that the experimental values matched the CFD values. The point of this step in the research was to obtain a pressure profile that had been influenced by upstream obstructions, and try to produce a CFD simulation that would show the trends that the experimental data showed. The losses experienced by 3-d wings, due to tip vortices, were not seen in the CFD. Even though the experimental data was taken at the mid-span region, the plates had aspect ratios of close to one, making it certain that even at mid-span, the 3-d effects were still seen. The following sections describe in more detail the comparisons between the experimental and CFD data, for both the uniform inlet and non-uniform inlet velocity cases.

2.3.4.1 Uniform Inlet Velocity Cases

The pressure coefficient trends found in the CFD simulation matched the pressure coefficient trends found experimentally, at low angles of attack, for the uniform inlet velocity cases. Figure 2.23 shows the pressure coefficient plots for angles of attack of zero, and +/- 4 degrees.
Figure 2.23: Cp comparison, AOA=-4,0,4 degrees, uniform inlet

The trends of the data are similar, but the magnitudes are very different. As was mentioned previously, this can be attributed to the fact that the CFD cases were 2-dimensional, but the experimental cases were 3-dimensional. It should be noted, however, that the profiles for -4 and 4 are the same, except for the reversal of upper and lower surfaces, for both the experimental and CFD results. The similarity in trends was less evident at angles of attack of 8 degrees and higher. The CFD results would have compared better to the experimental results, had the CFD data been corrected for 3-d affects. However, for the reasons stated previously, and the fact that correcting for 3-d affects would have been a tedious process due to the manner in which the CFD data was collected, this was not attempted. The plots for these angles can be seen in Figure 2.24.
The experimental plots for each of the angles of 12, 16, and 20 show a rise and then decrease in the magnitude of the pressure coefficient for the upper surface of the plate. This may be due to the existence of a laminar separation bubble near the leading edge of the plate. The existence of the laminar separation bubble is supported by the work of Julie Jones, 2004, done at RIT. In the presence of a laminar separation bubble, the flow separates from the surface and reattaches, creating a low pressure region, or bubble, in the separated flow area. Unfortunately, the work of Jones showed that the pressure taps in the test plate were too far downstream to accurately model the presence of the separation bubble. This experimental data found in this work shows the separation and reattachment of the flow on the upper surface, which may further support the existence of the laminar separation bubble.

The CFD results for the higher angles of attack show that the flow is separating near the leading edge; however, Fluent was unable to model the reattachment of the flow. Part of the problem with the CFD simulation was that in reality, there is turbulent flow, in the region of the detached flow. However, the Reynolds number of the flow falls within the laminar region. Fluent is equipped with a variety of solvers to model both turbulent and laminar flow, but does not have a solver that can model both types of flow.
simultaneously. Thus, the laminar solver was unable to accurately model the results because of the turbulent nature of the detached flow at the leading edge, and the turbulent solver, which was used for the results presented above, was unable to predict reattachment of the flow.

The experimental data shows that the pressure coefficient should be close to zero at the trailing edge of the plate. For the CFD data, however, the pressure coefficient is not close to zero at the trailing edge of the plate. This is because of the separation modeled by Fluent. Because the Kutta condition must be met in Fluent, the pressure coefficient of the lower surface rises to meet the pressure coefficient of the upper surface, such that the difference between the upper and lower surface coefficients is zero. The CFD results show that at the leading edge, even at high angles of attack, the plot of the pressure coefficient for the lower surface follows the trend of the experimental data, but toward the trailing edge it diverges sharply.

2.3.4.2 Non-Uniform Inlet Velocity Cases

The experimentally found pressure coefficients showed the influence of the non-uniform inlet velocity profile. Figure 2.25 shows the experimental data at an angle of attack of 20, which was where the influence of the non-uniform velocity inlet was most evident.

![Figure 2.25: C_p, AOA=20, non-uniform inlet](image)

It is evident that each of the three cylinders of the velocity inlet generator has an effect on the surface pressure coefficient of the flat plate. The plots of both the experimental and CFD results for the non-uniform inlet cases are shown in Figure 2.26.
Figure 2.26: Cp Comparison, AOA=-4-20, non-uniform inlet

The plots of the experimental data show the effect of the non-uniform inlet velocity. The separation that was seen in the uniform velocity profile results is not evident until an angle of 16. This suggests that the turbulence introduced by the velocity inlet generator causes the flow to remain attached over the plate, when in the previous case it had separated. The effect of the non-uniform velocity inlet is evident on both the upper and lower surfaces of the plate.
The CFD results produced a poor representation of the surface pressure coefficient. It was hoped that the surface pressure coefficients found using CFD would show the same oscillations on the upper and lower surfaces as the experimentally found coefficients; however, this was not the case. It is theorized that the CFD was unable to calculate the small oscillations in the pressure coefficient, because the simulations were made using the Spalart-Allmaras turbulent solver, when in reality the Reynolds number of the flow was in the laminar range.

Overall, the flat plate case was a poor choice for software validation. Fluent was unable to calculate both laminar and turbulent flow simultaneously, yielding poor comparison results. However, this validation case was useful in gaining experience with performing fast Fourier transform (FFT) analyses, which would prove necessary in the simulation of trailing edge blowing (TEB). The case was also useful in gaining experience with programming user-defined functions in C, to be imported into Fluent and used as boundary conditions.
3. Stator-Rotor Simulation – Preliminary Steps

The following chapter describes the preliminary steps that were necessary in simulating the stator-rotor interaction in the F109 turbofan engine. These steps included obtaining and manipulating experimental data, and devising a set of boundary conditions applicable to the problem. The chapter also includes the initial modeling attempts and results, as well as problems encountered and the solutions taken. Both the geometry modeling in GAMBIT and the solution modeling in Fluent are explored. The completion of the preliminary steps allowed for a final model that was robust. Much of the CFD simulations were completed on a trial and error basis. Because of the steps detailed in this chapter, future investigators will not repeat the same mistakes, and will be able to obtain a complete solution, leading to the optimization of TEB.

3.1 Existing Experimental Results

The data that were used for the CFD analysis comparison consisted of the total pressure measurements downstream of the IGV wake and were taken from the work of Dr. Jeff Kozak [Kozak, Dissertation, 2000]. The total pressure in the IGV wakes was measured at various circumferential locations, allowing for a complete 2-d wake profile to be represented by the data. Static pressure and inlet velocity measurements were also obtained. The following section details the manner and setup for which the experimental data for this investigation was gathered. The first section describes the experimental test setup, including the equipment designed for the experiments. The second section describes the experimental method including methods of data acquisition. All of the experiments were performed by Dr. Jeffrey Kozak, at Virginia Tech, as part of his doctoral research.

3.1.1 Experimental Test Equipment

The following section describes the test equipment used for the TEB experiments done by Kozak, as well as the method of data collection used.
3.1.1.1 Allied Signal F109 Turbofan Engine

An Allied Signal F109 turbofan engine was used for the experiments in this investigation. This engine was originally designed for use in the U.S. Air Force’s T-46 training jet, but was donated to Virginia Tech after the T-46 program was discontinued, due to airframe problems. A cross-sectional view of the engine is shown in Figure 3.1.

![Figure 3.1: F109 Turbofan cross-section](image)

The F109 is a contemporary designed, two-spool, medium-bypass-ratio turbofan engine. The first stage of the engine consists of a fan, providing the initial compression. The fan diameter is 18.7 inches, and consists of 30 blades. The blades vary in chord from hub to tip, and vary in span from leading edge to trailing edge. The dimensions of the fan blades are shown in Table 3.1.

<table>
<thead>
<tr>
<th>Chord at Tip</th>
<th>2.875 in (7.3 cm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Chord at Hub</td>
<td>2.25 in (5.7 cm)</td>
</tr>
<tr>
<td>Leading Edge Span</td>
<td>5.50 in (13.97 cm)</td>
</tr>
<tr>
<td>Trailing Edge Span</td>
<td>4.46 in (11.34 cm)</td>
</tr>
<tr>
<td>Stagger (Setting) Angle, Tip</td>
<td>59.2°</td>
</tr>
<tr>
<td>Stagger (Setting) Angle, Hub</td>
<td>29.7°</td>
</tr>
<tr>
<td>Leading Edge Thickness, Tip</td>
<td>0.019 in (0.048 cm)</td>
</tr>
<tr>
<td>Leading Edge Thickness, Hub</td>
<td>0.043 in (0.11 cm)</td>
</tr>
</tbody>
</table>

Table 3.1: Fan Blade Dimensions
3.1.1.2  Inlet Guide Vanes

The inlet guide vanes (IGV) designed for the study used a NACA 0015 profile, and had a chord, thickness, and wake profile similar to that of modern IGV. The vanes were designed without camber or turning, and were set to a zero angle of attack. The vanes were hollowed using EDM to create a plenum that would be injected with air during TEB. TEB holes were drilled in the trailing edge of the IGV, to serve as the jets that would fill the IGV wake. The holes were designed such that the flow velocity was subsonic, to keep the holes from choking. A schematic of the NACA 0015 IGV can be seen in Figure 3.2.

![Figure 3.2: NACA 0015 IGV with TEB holes [Kozak, 2000]](image)

3.1.1.3  Engine Inlet Design

Three separate components were constructed to allow TEB testing in the F109 turbofan engine. An IGV ring was constructed, in which an inlet guide vane could be mounted. The ring was also designed with static pressure taps on the inner diameter of the ring, to allow for inlet velocity measurements. The ring could either be mounted to the inlet traverse ring, to be described later, or to the engine inlet lip. Figure 3.3 shows a schematic of the IGV inlet ring.
An inlet traverse ring was also constructed, through which a pitot-static probe was passed. A slot was cut in the inlet traverse ring, allowing the pitot-static probe to be circumferentially indexed. This was mounted to the front engine lip. The IGV ring could then be mounted to the inlet traverse ring. Figure 3.4 shows the inlet traverse ring.

Because testing was done on the ground and not in flight, a bell mouth inlet was required, to minimize flow separation at the leading edge of the inlet. This was also designed for the experiments, and could be attached to the IGV ring.

3.1.1.4 Data Collection Method

The total pressure measurements in the inlet and downstream of the IGVs were obtained using total pressure probes, attached to pressure transducers. Measurement in the circumferential direction was achieved by indexing the pitot-static tube after each
total pressure measurement. The circumferential measurements allowed for the approximation of the IGV wake profiles.

### 3.1.2 Experimental Data

The experimental data used for the CFD simulations consisted of the unsteady total pressure profile in the IGV wakes for 10000 rpm and 11000 rpm. Data were taken for one complete revolution. The total pressure profiles were measured for 10k and 11k rpm with the IGVs far upstream of the rotor blades, to find a wake profile that was not affected by the potential field traveling upstream from the rotor blades. The pressure profiles were again measured with the IGVs close enough such that the potential field was realized by the pressure probe, to measure the unsteady wake profile. The data were then reduced such that the effect, but not the magnitude, of the potential field on the unsteadiness of the profile was incorporated.

#### 3.1.2.1 Experimental Setup

The following schematic diagram, Figure 3.5, details the test setup for gathering the experimental data. The IGVs were located 0.4 IGV chords upstream, and the pitot-static probe was located 0.25 fan chords upstream of the rotor leading edge. The IGV chord length was 3 inches. As seen in the diagram, the probe is initially directly downstream of the trailing edge of the IGV. Phase-locked data were taken for one complete revolution of the fan, and then the probe was indexed to the next circumferential location. The probe was immersed 1.1 inches into the flow, so as to make any wall interactions between the probe and the engine casing negligible.
3.1.2.2 Experimental Data Manipulation

The data were reduced by subtracting the pressure response due to the potential field traveling upstream from the rotor blades, leaving only the pressure response that was the effect of the IGVs. Data were gathered for both engine speeds, with and without trailing edge blowing. However, for the case with full trailing edge blowing, because the potential field forcing function had been eliminated, it was assumed that the pressure profile across the inlet was uniform. The data was in tabular form for each of 5 circumferential points, and was normalized by dividing by the total free stream atmospheric pressure of 1 atm. The unsteady pressure-versus-time data was averaged over each of the five points to obtain a 2-d profile representing the total pressure across the width of the blade pass. Figure 3.6 shows a plot of a typical wake profile for the case in which there was no TEB, at 10000 rpm.

For the phase locked data, the same time step between probe measurements was used for both the 10k and the 11k rpm cases. The experimental data set was not complete for the 10k data set, so the 11k data set was used to determine the size of the time step between probe measurements.
3.2 CFD Setup

The following section describes the methods used in simulating the stator-rotor setup that exists in the F109 turbofan engine. The attempted boundary conditions and assumptions are explained in detail, and different examples of the geometries used for the simulations are shown. The boundary conditions and geometries that yielded the most success are described further in Chapter 4.

The approach for incorporating trailing edge blowing was to slightly change the inlet pressure profile function to correspond to different wake-filling percentages. The rotation of the rotor blade for the initial attempts was simulated by assigning a phase-shift to the pressure inlet function, which was determined by an appropriate time step for the simulation. The CFD simulations were run using the transient solver in Fluent, with the phase-shift in the pressure inlet profile updating appropriately for every time step. The pressure coefficient was monitored and saved by Fluent on the upper and lower surfaces of the airfoil. After a reasonable result was achieved, it was manipulated and converted to a stress spectrum. This process is described in detail in Chapter 4. The end result of the data manipulation was a plot of stress vs. time, for the average stress on the blade.
This spectrum could be imported into a fatigue life prediction package, to obtain an estimate of the average number of cycles until failure.

### 3.2.1 System Geometry

The following section describes the methods and assumptions used to determine the geometry of the stator-rotor system components for modeling to use in the CFD simulations.

In order to create the geometry for the CFD modeling, it was necessary to estimate some dimensions of the blade geometry and surrounding fluid, using known values. The chords at the fan blade hub and tip were known, and can be seen in Table 3.1. The immersion of the pitot-static probe was also known, as was the tip clearance between the rotor blade and the engine casing. Figure 3.7 shows a side view of the experimental setup, with associated dimensions.

![Figure 3.7: Side view of experimental setup](image)

The radius from the engine centerline to the measuring location is noted as \( r \). This radius was necessary to determine the speed of the blade at the measuring point. The value for \( r \) was found using the dimensions given to be 8.25 inches. The spanwise location along the blade for which the measurements were taken is noted \( b \). This value for \( b \) was necessary in order to estimate the chord length at the measuring point. The value was found, using the geometry in Figure 3.7, to be 4.425 inches. Using the spanwise distance of 4.425 inches, and assuming that the increase in chord from hub to tip is
linear, the chord length was found to be 2.753 inches at the measurement location. This was important in modeling the rotor blades for the Fluent simulations.

The stagger angle was defined in the experiment as the angle of incidence between the rotor blade chord line and the azimuthal axis. The stagger angle at the hub and tip were given, as 29.7 and 59.2 degrees, respectively. It was assumed that the increase in stagger angle from hub to tip was linear over the span. Thus, the angle at the measuring location was found to be 53.4 degrees. This calculation was necessary in order to set the proper angle of attack between the rotor blade geometry and the horizontal axis in the GAMBIT model. The leading edge (LE) thickness decreased from hub to tip. Again, it was assumed that the change over the span was linear. The LE thickness at the hub was given as 0.043 inches, and at the tip was 0.019 inches. Thus, the estimated leading edge at the measuring location, at approximately 0.8b, was found to be 0.0237 inches.

Very little information was known about the actual geometry of the fan blades; Thus it was deemed necessary to make assumptions for thickness and shape, based on compressor aerodynamic theory. A common blade profile used for fans is the double circular arc. It is also common, for the double circular arc airfoil, for the maximum thickness of the blade to be equal to 2.5% chord, and to be located at 50% chord. The maximum thickness was found to be 0.138 inches. Figure 3.8 shows the blade profile, with the calculated dimensions.

![Figure 3.8: Rotor blade cross section dimensions](image)

### 3.2.2 Use of Experimental Data

The CFD simulations were completed using experimental data gathered by Dr. Jeffrey Kozak, for his Doctoral research at Virginia Tech. To use the data in the CFD simulations, it was necessary to perform some manipulations, which are described in the
following section. It should be noted that the units of the following data switch between English and metric units; this is not a factor, because in GAMBIT, dimensions are not used, and in Fluent, English or scientific units can be set for every necessary variable.

3.2.1.1 Assumptions

It was necessary to make assumptions to put the experimental data in a usable form. Because the experiments took place at Virginia Tech, which is close to sea level, it was assumed that standard values for ambient total pressure and density could be used. Thus, the total ambient pressure was assumed to be 101325 Pa, and the density was assumed to be 1.225 kg/m³.

3.2.1.2 Time Step and Velocity Calculations

The first step in determining the size of the time-step used in data collection was to find the blade passing frequency. This was done by multiplying the speed of the engine, in revolutions per second, by the number of blades in the fan, which was 30.

\[ BPF_{11k} = 11000 \text{rpm} \times \frac{1 \text{min}}{60 \text{sec}} \times \frac{30 \text{blades}}{1 \text{rev}} = 5500 \text{ blade per sec} \]  

Thus, the blade passing frequency was found to be 5.5 kHz. There were 2688 data points taken for 1 revolution of experimental data. To determine the number of time-steps per blade pass, it was necessary to divide the total number of data points by 30 blades. This yielded a value of 89.6 time-steps per blade pass, which was rounded up to 90 time-steps per pass. After the previous two steps were complete, it was possible to find the size, in seconds, of the time-step. This was done by taking the inverse of the BPF, to find the time in seconds for each blade pass, and then by dividing that answer by the number of time-steps per pass. Equation 3.2 shows the process used to determine the value of the time step for the 11k case.

\[ t_{-\text{step}} = \frac{1}{BPF} \times \frac{1}{\#\text{times steps}} = \frac{1}{5500} \times \frac{1}{90} = 2.02 \times 10^{-6} \text{ sec} \]  

The value for the time step was found to be 2.0202 microseconds. This value of the time step was used in the corresponding CFD simulation.

Experimental data did not exist for a complete revolution of the fan at 10000 rpm. This made it difficult to determine the number of time steps per blade pass for the 10000
rpm data. However, the assumption was able to be made that the value of the time-step was the same between the 10k and 11k data, because this was a characteristic of the measuring device. The blade passing frequency for the 10k case was found in the same manner as the 11k case.

\[
BPF_{10k} = 10000rpm \cdot \frac{1 \text{min}}{60 \text{sec}} \cdot \frac{30 \text{blades}}{1 \text{rev}} = 5000 \frac{\text{blade}}{\text{sec}} 
\]  

(3.3)

The BPF for the 10k case was found to be 5.0 kHz. Equation 3.4 was then used to determine the number of time-steps necessary for each blade pass.

\[
\# \text{timesteps} = \frac{1}{BPF} \cdot \frac{1}{t_{\text{step}}} = \frac{1}{5000} \cdot \frac{1}{2.02 \times 10^{-6} \text{sec}} = 99 \text{ timesteps} 
\]  

(3.4)

This showed that there were more time-steps necessary for the 10k blade pass than the 11k blade pass, which makes sense, because the 10k blade is moving the same distance in a longer amount of time. It was necessary to determine the linear distance traveled by the rotor blade for each blade pass. This was done by finding the circumference of the rotor blades at the measuring location, and dividing by 30, which was the number of blades in the entire fan. Equation 3.5 shows the equation and result.

\[
\frac{2\pi r}{\# \text{blades}} = \frac{2 \pi \cdot 8.275 \text{in}}{30 \text{blades}} = 1.7331 \frac{\text{in}}{\text{blade}} 
\]  

(3.5)

This value could then be divided by the number of time steps for the 10k and 11k cases, to find the linear distance traveled by the rotor per time step.

It was also necessary, in establishing the boundary conditions to be used for the CFD simulations, to calculate the relative speeds of the rotor blades for both the 10k and 11k cases. There were two components of the speed seen by the rotor blades. Figure 3.9 gives a schematic diagram of the components of the relative speed seen by the rotor blade. The first component, \( U \), was due to the rotation of the blade, and the second component, \( V \), was due to the velocity of the inlet flow. Resolving these two vectors gave a resultant speed, \( V_{\text{rel}} \), at some angle, \( \theta \).
Figure 3.9: Velocity Triangle

The angular speed of the blades was calculated using equation 3.6.

$$\omega_{10k} = 10000 \frac{\text{rev}}{\text{min}} \times \frac{2\pi \text{rad}}{\text{rev}} \times \frac{1 \text{min}}{60 \text{sec}} = 1047.2 \frac{\text{rad}}{\text{sec}}$$

The angular speed of the blades for the 11k case was found to be 1151.9 rad/s. The linear speed was then calculated by multiplying the angular speed by the effective radius and converting to standard units. Thus, for the 10k case, the linear speed of the blade due to its rotation was found to be 220.1 m/s. The speed for the 11k case was found in the same manner, and was equal to 242.1 m/s. The direction of the relative speed, represented by the angle theta, was calculated by taking the inverse tangent of the angular velocity divided by the inlet velocity, as seen in equation 3.7.

$$\theta = \tan^{-1}\left(\frac{U}{V}\right)$$

For the 10k and 11k cases, respectively, the resultant speed angles were found to be 70.95 and 70.8 degrees, respectively. The magnitude of the relative speed, which was the resultant of the combination of the inlet and rotational speeds, was found using equation 3.8.

$$V_{rel} = \sqrt{U^2 + V^2}$$

The relative speeds were found to be 232.9 m/s and 256.4 m/s for the 10k and 11k cases, respectively. These speeds were those being felt by the rotor blade, as it rotated through the fluid. The angle of incidence of the speed on the blade was the difference between the angle theta, and the angle of incidence of the blade, which was earlier found to be 53.4 degrees.
3.2.1.3 Inlet Function

To use the experimental data found by Kozak, 2000, it was necessary to perform mathematical manipulations to compile the data into a usable form. To use the data as a boundary condition, it was necessary to obtain a 2-d representation that was function of distance along the inlet, from the tabular experimental data. To achieve this, a Fast Fourier Transform (FFT) was performed using Matlab, as was detailed in section 2.3.3.2.

The set of data points was represented by a 2-d function in the form of equation 3.9.

\[ P = a_0 + 2 \sum_{i=1}^{n} a_i \cos(2\pi f_i y + \phi_i) \]  

(3.9)

This equation is similar to equation 2.5, where \( y \) is the vertical distance along the inlet. The Matlab function code used to find the values of the coefficients is shown in Appendix A1.

The final function representing the unsteady inlet total pressure consisted of a mean total pressure, plus 20 harmonic terms. The function was embedded within a block of C code, to allow it to be compiled and run in Fluent. Each value for amplitude, frequency, and phase was pulled from the values generated by the Matlab FFT. A plot of the resulting function for the case in which there was no TEB and the engine speed was 10000 rpm, can be seen in Figure 3.10. The dots represent the experimental data points, and the solid line represents the FFT curve-fit.

![Figure 3.10: FFT Approximation of total pressure data](image)
3.2.2 Initial Modeling Considerations

Because the effects of the viscous wakes propagating from the trailing edge of the stator in the F109 turbofan engine was represented using the inlet boundary condition, it was necessary only to model the rotor and the surrounding field for the CFD simulation. The following section details the many different approaches used for modeling the geometry, as well as associated schematic representations.

3.2.3 GAMBIT fluid modeling approach

Finding an accurate method of modeling the fluid area surrounding the rotor blades proved to be a difficult task. Because the inlet boundary condition was a linear function, it was necessary to model the geometry of the inlet of the fluid zone as a straight line. This was deemed to be most appropriate, given the manner in which the experimental data were taken. The first approach to modeling the fluid was just as a simple rectangle, with three rotor blades contained. The mesh was stationary, with the inlet boundary condition moving across the inlet boundary, to simulate rotation. The approach was to take pressure data on the center blade only. The upper and lower blades would serve to keep the effect of the upper and lower boundaries of the fluid from influencing the surface pressure data on the center blade. Figure 3.11 shows the geometry created for this configuration.

![Figure 3.11: 3 blade configuration, rectangular boundary](image)
The inlet of the fluid area was modeled 0.25 IGV chord lengths upstream of the leading edge of rotor blade, corresponding to the location of the pitot-static probe during testing. The outlet of the fluid area was initially placed 8 units downstream from the leading edge, assuming that the downstream boundary would be a pressure-outlet. The desire was to place the outlet far enough downstream such that the pressure at the outlet would not have an effect on the surface pressures on the center rotor blade. Meshes with both the tri-pave and quad-pave schemes were attempted with the rectangular boundary configuration, with differing edge meshing ratios, but a convergent solution was not achieved in Fluent. The quad-pave and tri-pave meshes can be seen in Figure 3.12.

Regardless of the meshing scheme used, convergence was not achieved. Many times the solution diverged because of mesh skewness in corners. It was thought that the problem with convergence was due to the fact that the rotor blades were at a high angle of attack, yet the outer boundaries were modeled as a rectangle. Thus, the next attempt in modeling the system was to model the outer boundaries parallel to the chord line of the rotor blades.
The second approach to modeling the system was to still use a stationary mesh, but to angle the upper and lower boundaries to reflect the angle of incidence of the rotor blades. The inlet was again placed 0.25 IGV chords upstream of the rotor leading edge, corresponding to the location of the pitot-static probe during testing. The geometry for the next set of meshes is shown in Figure 3.13.

![Figure 3.13: 3-blade configuration, parallel boundaries](image)

Meshes were built using both the quad-pave and tri-pave meshing schemes. However, due to the highly acute angles at the lower left and upper right corners of the outer boundaries, highly skewed face elements were present for all schemes used. Figure 3.14 shows a close up view of the face elements in the lower left corner of the mesh. The meshing scheme used for the mesh shown was quad-pave, with a total of 6137 elements for the entire mesh.
Figure 3.14: 3-blade configuration, highly skewed corner elements

The shaded elements in Figure 3.14 are those that have an equiangle skewness of greater than 0.5. Initially it was assumed that these elements would not have an effect on the solution, because they were far from surfaces of the center airfoil; however, when the Fluent solution was attempted, it was found that divergence occurred within the first few iterations, and the most greatly diverging values were at the lower left corner of the mesh. Thus, it was deemed necessary to model yet another configuration, which would yield a valid and convergent solution.

The next attempt in creating a mesh that would yield a convergent and stable solution was to build the geometry such that the upper and lower portions of the inlet were parallel to the inlet flow. It was thought that this attempt would also reduce the chance of the flow immediately diverging in the lower left corner of the fluid region. An example of the geometry built using this approach can be seen in Figure 3.15. Limited success was achieved with this mesh; divergence did not occur but the results were not stable, and did not converge in a timely matter.
Because the CFD solution was transient, it was important to build a mesh that would yield convergence in as few iterations as possible. The best steady state result achieved using the stationary mesh technique converged in 3000 iterations, however, this was deemed unacceptable because it took many hours to achieve. There were many meshes built using the stationary mesh technique, however, none of the meshes led to a quickly converging solution in Fluent. The corners of the boundaries were also rounded, in hopes of decreasing iterations required to achieve a steady state solution. A rectangular outer boundary was again attempted, using a quad-pave mesh. The number of elements in the mesh was limited by the computing power of the systems used, thus it was difficult to create a mesh with fine grading close to the rotor blade. The last attempt made using a stationary mesh was to create a mesh with only one rotor blade, and set the boundary conditions such that the effects of the other blades would be seen. Some examples of the different meshes used with the stationary mesh technique can be seen in Figure 3.16.
3.2.4 Initial Boundary Conditions

The following section details the boundary conditions used in the initial attempts at modeling the stator-rotor system. Because the mesh was stationary for the initial attempts at modeling the stator-rotor interactions in the F109 turbofan engine, careful consideration had to be taken for the boundary conditions. Many different types of boundary conditions were used, but none led to a quickly converging and stable solution in Fluent. The inlet and outlet boundaries were initially set as pressure-inlet and pressure-outlet boundaries, respectively. The proper conditions for the upper and lower boundaries, however, were more difficult to determine and proved to cause problems in the CFD simulations.

3.2.4.1 Inlet Boundary Condition

The inlet boundary condition was assigned as a pressure-inlet. This was the most logical solution because the experimental data was of total pressure of the flow. The values for the constants found using a Fast Fourier Transform (FFT), as described in
section 3.x, were coded into a block of c-code, which was then imported and compiled as a user-defined function (udf) in Fluent. The code for the udf can be seen below.

```c
DEFINE_PROFILE(no_teb_10k_pressure_inlet, thread, nv) {
    float x[3];
    float y;
    float pi;
    face_t f;

    /* coefficients for cosine inlet function */
    float a0, a1, a2, a3, a4, a5, a6, a7, a8, a9, a10, a11, a12, a13, a14, a15, a16, a17, a18, a19, a20;
    float f1, f2, f3, f4, f5, f6, f7, f8, f9, f10, f11, f12, f13, f14, f15, f16, f17, f18, f19, f20;
    float p1, p2, p3, p4, p5, p6, p7, p8, p9, p10, p11, p12, p13, p14, p15, p16, p17, p18, p19, p20;

    begin_f_loop(f, thread) /* a looping MACRO used to access all cells or cell faces */ {
        F_CENTROID(x, f, thread); /* a MACRO that assigns Cell positions to x */
        y = x[1];
        pi = 3.14159;

        a0 = 100962; a1 = 66.1366; a2 = 51.5334; a3 = 31.7971; a4 = 29.9774; a5 = 28.3677; a6 = 20.1083; a7 = 17.2224; a8 = 13.1943; a9 = 9.6001;
        a10 = 7.9397; a11 = 7.7289; a12 = 7.3173; a13 = 6.9766; a14 = 4.6289; a15 = 4.6108; a16 = 3.8219; a17 = 3.6834; a18 = 3.5669; a19 = 3.3131; a20 = 3.0927;

        p1 = -1.0867; p2 = -2.5083; p3 = -0.27120; p4 = -2.6752; p5 = 0.8261; p6 = -1.8428; p7 = 2.1259; p8 = 2.4438; p9 = -0.6077;
        p10 = 2.7308; p11 = 2.3181; p12 = 1.7704; p13 = 2.0378; p14 = 1.3162; p15 = 2.1592; p16 = 1.9588; p17 = 2.1030;

        F_PROFILE(f, thread, nv) = a0 + 2*a1*cos(2*pi*f1*y+p1)
    }
}
```

77
The udf showed up as an option for selection in the boundary condition panel. The code shown previously is the final code used for the inlet boundary condition. However, initially, there was another term in the cosine function to allow for the propagation of the function in the vertical direction across the inlet boundary, as this was the attempt used to simulate a transient solution using a stationary mesh. The extra term was scaled to correspond to the size of the time step, such that the function would repeat itself over every blade pass.

### 3.2.4.2 Outlet Boundary Condition

The outlet boundary condition, for most simulations, was set as a pressure outlet. This caused problems initially, because it was difficult to determine what the pressure should be at the outlet. According to the F109 engine data, the total pressure rise ratio through the fan was 1.6 at maximum engine speed. However, when this pressure ratio was used to determine the total pressure at the outlet, a converged solution was not achieved. Furthermore, upon examining the contours of static and total pressure in the
fluid region, it was realized that the back pressure was gradually moving forward in the flow, until eventually the system stabilized with the total pressure of the entire field being nearly equal to the total pressure at the outlet. In essence, this meant that the rotor blades were not experiencing the effect of the inlet pressure function, because the high back pressure had caused the profile to wash out.

Multiple outlet pressure ratios, including a value of unity, were used as the pressure outlet boundary condition. Every time the same results were achieved: the pressure of the entire flow field would eventually converge to the pressure of the pressure outlet, which is obviously inaccurate. Thus it was deemed that this assumption for the outlet boundary was inaccurate. Multiple types of outlet boundary conditions were used, including an outflow condition, and an outlet vent condition. The Fluent help manuals recommend using these types of boundary conditions when little is known about the flow as it exits the flow field. After many unsuccessful attempts, a Fluent help representative was contacted, and it was concluded that another UDF should be written to determine the values for temperature and pressure at the outlet [Basu, personal communication]. This UDF will be detailed in Chapter 4. The conclusion was made that the pressure-outlet was the proper type of boundary condition to be used at the outlet, but it was in error to assign the entire constant pressure value, because in reality, the value could vary along the boundary.

3.2.4.3 Upper and Lower Boundary Conditions

Determining the type of boundary to be used initially for the upper and lower boundaries of the fluid zone was difficult with the 3-blade mesh scheme. This was because none of the available boundary conditions truly represented what was going on in the system. Initially, these boundaries were set as pressure-far-field, with the mach number set to the inlet velocity given in the experimental data. This, however, did not yield a valid solution in Fluent. The second type of boundary that was tried with many of the different meshes was a wall boundary. It was thought that because the center rotor blade was the only blade for which data was being taken, assigning the upper and lower boundaries as walls would not have an effect on the results. However, because of the skewed corner elements, the simulations using wall boundaries diverged.
Pressure outlet boundary conditions were also used, but as with the outlet boundary condition, it was impossible to determine the values for the pressure all along the boundaries. A symmetry boundary was also attempted using the 3 blade approach, but an error was given in Fluent when trying to assign such a boundary. A symmetry boundary simply implies that the fluid has the same characteristics on one side of the boundary as it does on the other, and is commonly used in stator-rotor simulations. As the meshing approach gradually moved to a one-blade scheme, the symmetry boundary was revisited, and was eventually determined as the proper boundary condition for the problem. The added benefit of using a symmetry boundary and modeling only one blade was that the mesh could be finer over the entire area, and a solution could still be obtained in a reasonable amount of computing time. Figures detailing the symmetry boundary will be shown in Chapter 4.

3.2.5 Modeling Issues

One of the problems encountered while modeling the rotor blade geometry was determining the blade profile. Initially the blade was modeled with a very small maximum thickness, of only 0.019 inches, and was modeled with a sharp leading and trailing edge. This was a poor assumption, because in reality, the blade would have had at least a small radius at the leading and trailing edges. Convergence was not achieved while using the geometry with the sharp LE and TE, and for some cases the solution diverged at the LE; thus a different modeling approach was taken. A radius was given to the leading and trailing edges of the rotor blade, allowing for the mesh to form around the surface. The final blade profile is shown in section 3.8.

Another issue experienced was the presence of very highly skewed elements near the boundaries of the mesh and the surfaces of the rotor blades. The high angle of incidence of the rotor blade made it very difficult to produce square meshes. Divergence occurred in the corners of the fluid region because of the highly skewed meshes. This problem was solved by using a tri-pave scheme in the final meshing technique. The tri-pave scheme allowed for a mesh of more uniformly shaped elements, with less equiangle skewness overall.
Due to the limited processing power of the computers available to run the simulations on, the size and number of elements in the mesh was also limited. For a 2-d mesh, it was generally acceptable to have a mesh of 100,000 elements or less. Using the three rotor blade geometry scheme, and a tri-pave scheme, it was very difficult to create a mesh that was sufficiently fine toward the rotor blade surface. Transitioning to a one-blade modeling approach alleviated this problem. The preceding information should prove useful to future investigators, such that these mistakes in modeling can be avoided in subsequent studies.
4. **Stator-Rotor Simulation – Final Approach**

The following chapter will explain the final approach taken in modeling the F109 turbofan engine stator-rotor system using computational fluid dynamics. After many attempts to model the system using a stationary mesh scheme failed, it was finally decided that a moving mesh scheme should be used. A moving mesh scheme would more closely resemble the actual configuration of the rotating stator-rotor system. Furthermore, using a moving mesh scheme with a symmetry boundary condition at the upper and lower boundaries of the fluid allowed the specification of rotational motion. Specifying the azimuthal axis to be the axis of rotation allowed for a 3-dimensional rotational simulation, using a 2-dimensional mesh. This was the best solution considering the limited computing power that was available.

The following sections will detail the final meshes built and boundary conditions used, and will explain the model setup in Fluent. The CFD results will then be presented pictorially. The method for determining the stress spectrum on the rotor blades will also be explained. The last section of this chapter will show the final results achieved for the rotor blade stress spectra.

### 4.1 CFD Setup

This following sections explain the meshing schema and techniques used to build the final geometry used for the CFD simulations. The Fluent setup, including boundary conditions and model parameters, will also be detailed.

#### 4.1.1 GAMBIT Modeling

The final geometry used for the CFD simulations consisted of two separate meshes. The larger of the two meshes contained one rotor blade, with the upper and lower boundaries a distance of one blade pass above and below the leading edge of the rotor blade. The smaller of the two meshes was the portion that would be assigned a velocity in the y-direction to simulate the rotation of the blade. The inlet of this mesh was placed 0.25 IGV chord lengths upstream of the rotor leading edge, as was done with the previous mesh attempts. Henceforth, the smaller of the two meshes will be referred
to as the stator mesh, because it represents the fluid area just downstream of the stators, and the larger of the two meshes will be referred to as the rotor mesh, because it contains the rotor blade. Figure 4.1 shows the geometry built for both meshes.

![Final mesh geometry](image)

**Figure 4.1: Final mesh geometry**

The outlet of the mesh was placed far enough downstream from the leading edge that the outlet pressure would not have an effect on the surface pressure measurements. The profile of the rotor blade corresponds to the cross-sectional shape and dimensions detailed in Chapter 3. It was decided early in the modeling of the final mesh that a quad-pave scheme would be used for face meshing. This scheme was chosen because it yielded the least amount of skewed elements, and a sufficiently fine grade could be applied near the surface of the rotor blade without overloading the available computing resources. This was important for the transient solutions, because many time steps were required to obtain one blade pass worth of data. Figure 4.2 shows the geometry with labeled edges, and Table 4.1 shows the number of meshes and the grading, corresponding to the labeled edges.
Figure 4.2: Final mesh scheme - edge meshing

<table>
<thead>
<tr>
<th></th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
<th>10</th>
<th>11</th>
<th>12</th>
<th>13</th>
<th>14</th>
<th>15</th>
<th>16</th>
</tr>
</thead>
<tbody>
<tr>
<td>IC</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>5</td>
<td>5</td>
<td>5</td>
<td>5</td>
<td>5</td>
<td>121</td>
<td>121</td>
<td>89</td>
<td>89</td>
<td>69</td>
<td>9</td>
<td></td>
</tr>
<tr>
<td>G</td>
<td>N</td>
<td>N</td>
<td>N</td>
<td>N</td>
<td>N</td>
<td>N</td>
<td>N</td>
<td>N</td>
<td>N</td>
<td>N</td>
<td>N</td>
<td>N</td>
<td>BS-.6</td>
<td>BS-.6</td>
<td>N</td>
<td>N</td>
</tr>
</tbody>
</table>

Table 4.1: Rotor mesh details

In the table, IC stands for interval count, and the corresponding number is the number of
intervals, i.e. the number of meshes along the edge; G stands for grading, and the letter N
means that no grading was used, while BS-.6 means that a bell-shaped edge mesh was
used, with a grading value of -0.6. The bell-shaped mesh with the negative grading was
used on the edges of the rotor blade, to allow for a denser mesh toward the leading and
trailing edges. Because all of the edges were meshed separately, a face mesh size was not
specified when the face was meshed. Figure 4.3 shows the face mesh for the entire fluid
region, as well as enlarged views of the mesh at the leading and trailing edge of the rotor
blade.
The mesh created had a total of 17043 elements, with only 0.31% of elements having an equiangle skewness value of greater than 0.4. By CFD terms, this is a very low value, and thus it is reasonable to assume that the skewness will not affect the solution. This could further be assumed because of the location of the highly skewed elements. Figure 4.4 shows the elements of the rotor mesh that have a skewness value of greater than 0.4.
As is seen in the above figure, the most highly skewed elements are not near any of the boundaries, nor are they near the surface of the rotor blade. This decreases the chance that these skewed elements will affect the results of the simulation. The most highly skewed element in the entire mesh has a value of 0.49, which leads to the conclusion that the mesh quality is high.

The stator mesh was also meshed using a quad-pave scheme. Figure 4.5 shows the meshed fluid region, as well as numbered edges. The mesh details can be seen in Table 4.2. The quality of the stator mesh was very high, because of rectangular shape of the boundary. The most highly skewed elements had an equiangle skewness of 0.01, which means that the element is very close to square. The elements of this mesh were sized such that they were similar in size to the majority of the elements in the rotor mesh, yet were sufficiently small enough to show the resolution of the user defined inlet boundary condition across the inlet of the mesh. This mesh had a total of 2871 quad-pave elements.
To use the moving mesh technique in Fluent, it was necessary to merge the stator and rotor meshes to form one mesh. The meshes were merged based on direction from a Fluent tutorial on modeling stator-rotor interactions. The tmerge command was used in the GAMBIT command line, and the names of both meshes, as well as the name of the final mesh, were specified. The merged mesh did not create a new GAMBIT .dbs file, so it could not be manipulated in GAMBIT, but it did create a .msh file, which is the format used when importing into Fluent. Because the merged mesh could not be changed, any time either the stator or rotor mesh was changed, they had to be re-merged for importing into Fluent. Merging the meshes using the tmerge command allowed both meshes to be shown in Fluent, yet they could be controlled independently. This was necessary because for the simulations, the stator mesh was assigned a vertical velocity, while the rotor mesh was held stationary. Figure 4.6 shows the merged meshes in Fluent.
4.1.2 Fluent Setup

The following section describes the steps taken to model the stator-rotor problem in Fluent. This section will detail the solvers used and the assigned boundary conditions. The cases were started by importing the merged meshes into Fluent, and then checking and scaling them to match the dimensions of the problem.

4.1.2.1 Fluent Models

This section will explain the solvers chosen in Fluent, and the reasoning behind their selection. Upon initialization of Fluent, the 2-d, double precision solver was selected. Cases were run using both the single precision and double precision solver, and it was determined that the double precision solver should be used, because it resulted in the solution converging in fewer iterations. The downside of using the double precision solver was that it required a higher computing time per iteration, but because of the decreased number of iterations required to achieve convergence of the solution, this was deemed acceptable.

The coupled explicit solver was used to solve the problem. This solver was chosen after some trial and error using the other solvers. This was also the solver that
was used in the Fluent tutorial case. One of the benefits of the coupled explicit solver is that it couples the necessary equations, resulting in faster converging and more accurate results than a segregated solver. The explicit solver was chosen over the implicit solver because although the implicit solver converges in fewer iterations, each iteration takes much longer than when the explicit solver is used, especially in the case that the double precision option is enabled.

The K-Epsilon model was selected for viscous calculations. This model was selected for its robustness and common use in aerospace applications. To verify that this model was the best to use, the simulations were attempted using the other viscous models. The inviscid model diverged when attempted, and the laminar model did not converge. The Spalart-Allmaras model was also attempted, and it gave results that looked reasonable, but the K-E model was selected over the S-A model because the S-A model is relatively new, and only uses one equation, while the K-E model uses two. This was also the model used in the stator-rotor interaction tutorial. Because the solution was not inviscid, it was necessary to enable the calculation of the energy equation. The material assigned in the materials panel for the fluid region was air, as expected, with the ideal gas law selected to calculate the density.

4.1.2.2 Fluent Boundary Conditions

One of the most important aspects of the stator-rotor simulation was assigning the proper boundary conditions in Fluent. The upper and lower boundaries of the stator and rotor meshes were both assigned as periodic boundaries. This would accurately represent the problem and allow continuity to be conserved across the boundary. The actual engine set up consisted of circumferential stator and rotor blades. To represent the system in 2-d, the blades were essentially unrolled, to form a cascade in Fluent. The assignment of the periodic boundary allowed the use of the rotor mesh with only one blade, yet the effects of the blades above and below the rotor blade would still be realized in the solution. Figure 4.7 shows a view in Fluent of 3 periodic fluid zones, which were displayed by increasing the periodic repeats in the views panel.
The periodic boundary condition cannot be set in the Fluent boundary condition panel, as can the other types of boundaries. This boundary condition must be set using text commands in the Fluent GUI. The upper and lower boundaries of the stator mesh were called upper-stator and lower-stator, and the upper and lower boundaries of the rotor mesh were called upper-rotor and lower-rotor. The following text shows the command used to assign the upper and lower boundaries as periodic.

/define/boundary-conditions/modify-zone> make-periodic

Upon entering the previous command, the user is prompted to enter the name of the edge that is to become periodic, which in this case was named upper-stator. The user is then prompted to enter the name of the “shadow” edge, which in this case was named lower-stator. The order of the specified edges is arbitrary. The user is then asked if the zone is to be translationally or rotationally periodic; from Figure 4.7, it can be seen that the zone is translationally periodic in the vertical direction. This can be a source of confusion, because the case is representing rotational motion, however, setting the zones as rotationally periodic would have caused them to multiply in the plane of the rotor blade, which is not an accurate representation of the system. The user is then prompted to enter a translation vector, or to allow Fluent to automatically calculate it. The second option was used for this case. The entire process described above was repeated for the rotor mesh.
The next step in assigning the boundary conditions was to assign the outlet of the stator mesh, which aligned with the inlet of the rotor mesh, as an interface boundary. If this step were not taken, Fluent would automatically assign these boundaries as wall boundaries, which would not allow fluid to pass between the two regions. The assignment of the interface boundary ensured that there was continuity between the stator region and the rotor region of fluid. Initially, when the attempt was made to assign these boundaries as interfaces, an error was received, and the assignments were unsuccessful. A Fluent help representative was contacted and it was realized that extra commands were required to allow the assignment of an interface boundary that was also periodic [Basu, personal communication]. The following commands were entered into the Fluent command line, after which the interfaces could be created.

(rpsetvar 'nonconformal/allow-interface-at-periodic-boundary 0)
(rpsetvar 'nonconformal/cell-faces 0)

One very important thing to note, which was discovered after the above commands did not initially work, is that for the interface boundaries to be set as periodic, the periodic upper and lower edges must be meshed in the same direction, that is, the arrows must point the same direction in GAMBIT when the mesh is created.

The outlet boundary condition, was set as a pressure-outlet. It was difficult to determine the values that should be set for temperature and pressure, as was described in Chapter 3. However, the values for total pressure and total temperature were eventually calculated using two different user defined functions. The C code for these functions are shown in Appendix A2.

The above code simply takes the value of the temperature or pressure found during the previous iteration from the cell adjacent to the boundary, and assigns it to the value at the boundary. This allows the pressure and temperature at the boundary to change with time, and prevents the back pressure build up that was seen in the initial attempts at modeling the system. This assignment to the outlet boundary condition was a significant step in achieving a converged solution. It is important to note, however, that these udfs cannot initially be assigned to the outlet boundary, because there is no data known for the adjacent cells upon solution initialization. Thus, it is necessary to first
assign a constant pressure value to the outlet boundary, and then reassign the values to be calculated by the user defined functions after one iteration of the solution.

The inlet of the mesh was assigned as a pressure-inlet. The value across the boundary was calculated by a user defined function, as described in Chapter 3. The rotor blade was given a wall boundary condition, with the motion set to stationary and the shear condition set as no-slip.

4.2 Stator-Rotor Simulation CFD Results

The following section will present the both the steady and unsteady CFD results. The steady results are presented not to draw conclusions, but to explain the process taken in modeling the unsteady system. In reality, there is no steady state solution for the stator-rotor interaction, however, it is common practice in CFD to first simulate a steady-state solution, so that the unsteady solution has a set of beginning conditions. This increases the chance of convergence and the accuracy of the results for the unsteady solution.

4.2.1 Fluent Results – Steady State Solution

When modeling an unsteady CFD simulation, it is common practice to begin with the results of a steady state simulation. This section describes the results found for the steady state solution. For each case, a converged steady state solution was found before moving on to the unsteady simulation. For the steady state solutions, the Courant number was left at the default value of unity, and the multigrid levels were increased from 0 to 5. This was done based on the suggestion of the Fluent help guide, to increase the stability of the system as iteration took place. The steady state solution was iterated and the residuals for continuity, x and y velocity, energy, and k and epsilon were monitored. The solution was considered converged when every residual had dropped below 0.001. There were four different cases simulated, representing engine speeds of 10k and 11k rpm, and with or without trailing edge blowing.

4.2.1.1 Steady State Solution – 10k, No TEB

The first set of results that will be shown is for the case in which the engine speed was 10000 rpm, and TEB was not used. The solution was iterated until it converged,
after 1883 iterations. The residuals for the 10k, no TEB solution, can be seen in Figure 4.8.

![Residuals graph](image)

**Figure 4.8: 10k, no TEB scaled residuals**

As can be seen in the figure, residuals continuously drop until the solution is converged. Figure 4.9 shows the total and static pressure contours, respectively, of the steady state solution.

![Pressure contours](image)

**Figure 4.9: Total and static pressure contours, 10k, no TEB**
The total pressure ranges between 91.6 and 102 kPa. There is an area of low static pressure on the lower side of the rotor blade, which is expected because of the negative angle of attack of the blade. By looking at the static pressure contour plot, it can be seen that there is an area of very high pressure toward the inlet of the mesh. The only velocity that the blades are experiencing is in the x-direction, and fluid cannot flow freely through the system because it is essentially experiencing blockage by the blades.

It can also be observed that continuity is conserved across the interface boundary, and that the periodic boundaries are serving their purpose. Upon observation, it can be seen that the total pressure varies across the inlet. Figure 4.10 gives a better idea of the variance in pressure across the inlet. The values in the contour plot were clipped to a smaller range to show the difference across the inlet, and to further illustrate that the solution is as expected, the plot of the user defined pressure inlet function is also shown.

![Figure 4.10: Total pressure across inlet, 10k, no TEB](image)

Examining the velocity contours and vectors also shows an interesting response to the pressure inlet boundary condition. Figure 4.11 shows the velocity contour plot, and an enlarged view of the velocity vectors at the leading edge of the rotor. The units of the velocity are in m/s.
It can be seen from the contour plot that the maximum speed occurs between the rotor blades. Upon examining the velocity vectors at the leading edge of the rotor blade, it is apparent that separation is occurring, due to the large negative angle of attack of the rotor blade. The separated, turbulent flow is acting as a boundary, and causing a nozzle effect between the blades. This leads to the accelerated flow, which is represented by the red and orange coloring in the contour plot.

4.2.1.2 Steady State Solution – 10k, Full TEB

For the case in which full TEB was used, it was assumed that the total pressure deficit across the inlet had completely compensated, resulting in a uniform pressure across the inlet. The total pressure across the inlet was set at 101325 Pa, and the temperature was set to 286.5 degrees K, which was the ambient temperature for the experimental results. The steady CFD results for this case will not be shown, but the unsteady results will still be detailed for comparison to other cases.

4.2.1.3 Steady State Solution – 11k, No TEB

The next set of results that will be shown is for the case in which the engine speed was 11000 rpm, and TEB was not used. Again, the solution was iterated until it converged, after 1947 iterations. The residuals were monitored, and the solution was
deemed converged when the residuals for continuity, x velocity, energy, k, and epsilon dropped below 0.001. For this case, however, the residuals for the y velocity were approximately 50% higher than the other residuals, and did not drop to less than 0.001. It was eventually determined that because all the other residuals had dropped to 0.001. In all cases, the residual for the y-velocity was the highest throughout the solution. It is thought that this is because the incoming flow was at a high angle of attack with respect to the rotor blade. The residuals for the 11k, no TEB solution, can be seen in Figure 4.12.

![Figure 4.12: Scaled residuals, 11k, no TEB](image)

The residuals behave in a similar manner to those for the 10k solution. Over the first 200 iterations, there is a large drop in residuals. They then begin to increase, but begin to decrease again around 600 iterations. The residuals alternate between increasing and decreasing, but the general trend over the course of the entire solution is that the residuals decrease on the order of 3 degrees of magnitude. The residuals for the 11k solution were less stable over time in decreasing, most likely because the total velocity was higher. A contour plot of the total pressure of the steady state solution for 11000 rpm can be seen in Figure 4.13.
Figure 4.13: Total pressure contours, 11k, no TEB

The total pressures lie between 89.4 and 101 kPa. Again, there is an area of low pressure on the lower side of the rotor blade. Figure 4.14 shows the total pressure in the vicinity of the inlet, with the inlet total pressure function shown to its right. The values in the contour plot were clipped to a smaller range to show the difference across the inlet.

Figure 4.14: Total pressure across inlet, 11k, no TEB

The total pressure inlet function from the 10k case does not show the same characteristics as the total pressure inlet function from the 11k case. The reason for this is that the data points taken to create both functions were from highly unsteady experimental data for one revolution of the fan. The pressure profile was very different for every blade pass, so for the 10k and 11k case, the blade pass taken for use in the simulations was the worst case measured scenario, i.e., the case in which the difference
between the highest measured pressure and the lowest measured pressure was the greatest.

4.2.1.4 Steady State Solution – 11k, Full TEB

For the 11k case in which there was full trailing edge blowing, the total pressure at the inlet was set to 101325 Pa. The system was driven by the setting of the supersonic initial total pressure, which corresponded to the engine inlet speed. The inlet speed measured for the 11k case, from the experimental results, was 84.3141 m/s. The initial gage pressure was determined using equation 4.1.

\[
\frac{P_{\text{atm}}}{P_i} = \left[1 + 0.2M^2 \right]^{\frac{\gamma-1}{\gamma}}
\]  (4.1)

The Mach number for the steady state, in which the rotor blades were stationary, was found to be 0.224. The standard value of 1.4 for the specific heat ratio was used. Thus, the initial gage pressure was found to be 97063.95 Pa. The same value for the pressure was initially set as the outlet condition, causing fluid to flow through the system.

The residuals were monitored for this case, as in the previous case, and were considered converged when every value was less than 0.001. The steady state solution required 2041 iterations to converge. The plot of the scaled residuals can be seen in Figure 4.15.

![Figure 4.15: Scaled residuals, 11k, full TEB](image)
There was a brief increase in residuals around 1750 iterations, but the trend quickly reversed itself, and converged soon after. The total pressure contours are shown in Figure 4.16.

![Figure 4.16: Total pressure contours, 11k, full TEB](image)

The total pressures range between 88.6 and 102 kPa. As expected, the pressure is uniform across the inlet of the mesh, and again there is a low pressure region attached to the lower region of the rotor blade. These results do not represent the actual F109 engine characteristics, because in reality, fluid would not be flowing through engine if the rotor blades were not rotating.

The focus of this chapter will now switch to the presentation of the unsteady data acquired in the simulations. The previous data was presented only to show the procedure taken in modeling the unsteady system, and not to draw any conclusions about the success of modeling trailing edge blowing using computational fluid dynamics. However, the data previously presented was successful in proving that a complicated inlet profile can be assigned as a user defined function, and that the periodic and interface boundary conditions are useful in modeling stator-rotor interactions.

### 4.2.2 Fluent Results – Unsteady Solution

This section describes the results found for the unsteady state. For the unsteady state solutions, the Courant number was left at the default value of unity, and the multigrid levels were increased from 0 to 5. The discretization schemes were all set to
second order, to increase solution accuracy. The residuals were monitored for the same variables in the unsteady solution, as was done for the steady solution. However, due to the behavior of the unsteady solution, the convergence criteria of 0.001 was lowered to 0.0001 for each residual. This was to ensure that solution iterated multiple times for each time step, because the residuals have a tendency to spike and then drop very steeply with each time step update. The same four cases were examined for the unsteady solution: 10k and 11k, both with and without TEB. The solution was considered converged when every residual had dropped below 0.0001. For the unsteady solutions, the ‘Time’ parameter was set to ‘Unsteady’ in the solver panel.

The same boundary conditions for the mesh boundaries were used for the unsteady solutions as for the steady solutions. The only difference between the unsteady solution and the steady solution is that for the unsteady solution, a velocity is assigned to the rotor mesh. In the boundary conditions panel, the attributes for the fluid can be set. To do this, the fluid zone must be selected, and then the ‘set’ button must be selected. Figure 4.17 shows the boundary conditions panel, and the window that appears after ‘set’ is selected.

![Boundary conditions and fluid panels, 10k rpm](image)

Figure 4.17: Boundary conditions and fluid panels, 10k rpm

The panel shown in Figure 4.17 is for one of the 10k cases. As calculated in Chapter 3, the equivalent linear velocity representing an engine speed of 10000 rpm was found to be 220.1 m/s. Thus, for the rotor mesh, the fluid was assigned a velocity of 220.1 m/s. The negative sign represents that the mesh is moving in the negative y direction, as shown in Figure 3.9. For the 11k case the velocity in the negative y
direction was set to 242.1 m/s, as calculated previously. The stator mesh was not assigned a velocity. The assignment of a velocity to the rotor mesh only best represents the actual case, because in reality, the only moving components are the rotor blades.

For each simulation, the unsteady solution was initialized using the values found for the steady state solution. The time step was the same for both the 10k and 11k rpm solutions, and was found in Chapter 3 to be 2.0202e-06 seconds. The difference between the blade passes for the 10k and 11k cases was that for the 10k case, 99 time steps were required for one complete blade pass of data, and for the 11k case, only 90 time steps were required for one complete blade pass of data.

When these simulations were initially run, it was believed that the solution for the blade surface pressure response would repeat itself every blade pass. A periodic surface pressure response also leads to the belief that the rotor blade total stresses should be periodic, and repeatable. Thus, initially only one blade pass of data was gathered for each case. However, to check the repeatability of the solution, data for two consecutive blade passes were gathered, and it was discovered that the results were quite different for the surface pressure response on the rotor blades, between blade passes.

Upon further investigation, it was in the Fluent tutorials that there should to be at least 12 blade passes worth of data, before the solution begins to become periodic. It was stated in the tutorial that this fact was “known from past experience”. Each case was run for at least 12 blade passes, before the surface pressure data was gathered. However, even after 12 blade passes, the blade stress responses were still not periodic, after the data had been input and manipulated in a spreadsheet tool. In the Fluent tutorial that examined stator-rotor interactions, the rotational velocity of the rotor blades was much lower than in the present cases. Thus, it was thought that more blade passes might be required for the data to become oscillatory, for higher rotational speeds. However, no information was found explaining the correlation between number of blade passes, and data repeatability.

It was determined then, that much more data would be required to analyze the utility of using CFD to model trailing edge blowing effectively. Unfortunately, due to time constraints because of the limited computing power available, not all cases were able to be fully investigated. The most completely investigated case was with no TEB, at
10000 rpm. For this case, there were over 60 blade passes solved, with data taken for blade passes 61 and 62. The surface pressure responses will only be described qualitatively in this section, and will be described further, with supporting calculations, in section 4.3, which describes the mathematical manipulations required to find the time dependent blade stresses.

4.2.2.1 Unsteady Solution – 10k, No TEB

The residuals are shown for the last iterations performed in this case. They are shown only to illustrate the trends for the unsteady cases. Each spike corresponds to the updating of the time step. As the solution progresses, less iterations are required for each time step, until convergence is reached. When the convergence criteria are met, the time step is updated, causing the next spike in the values of the residuals. Figure 4.18 shows the residuals for the iterations up to the 18th blade pass. This plot is meant only to show the manner in which the residuals spike, and then drop, with every time step. The plot illustrates that for each time step update, the residuals dropped by four orders of magnitude. Like the steady state update, the y-velocity residual is on average the highest. The plot also shows that for just 18 blade passes, nearly 37000 iterations were required.

![Figure 4.18: Scaled residuals up to 18 blade passes, 10k, no TEB](image)

For the other cases that will be shown in the following sections, data is shown for around the 18th blade pass. Thus, for purposes of comparison, the 18th blade pass will be shown for this case, as well as later blade passes. Figure 4.19 shows the total and static pressure profiles for the case in which the engine speed was 10000 rpm, and trailing edge blowing was not used.
Figure 4.19: Total and static pressure, blade pass 18, 10k, no TEB

A plot of the static pressure over the surface of the rotor blade can be seen in Figure 4.20.

Figure 4.20: Surface pressure, blade pass 18, 10k, no TEB

The points forming the plot of the surface pressure are exported to a data file, and can then be imported into a spreadsheet program. The calculations performed on the data will be detailed in section 4.3.
When the static pressure profile over the surface of the blade was imported into the spreadsheet tool, it was found that 18 blade passes were not enough to establish a periodic profile. Because information could not be found on how many blade passes were required to establish periodicity, the solution was left to iterate for 4 days, totaling 60 blade passes. Surface pressure data were taken and exported for blade pass numbers 61 and 62.

Figure 4.21 shows the total and static pressure contour plots, respectively, for beginning of blade pass 61.

![Figure 4.21: Total and static pressure, blade pass 61, 10k, no TEB](image)

The pressure contours look very similar and have similar ranges to those for blade pass 18, however, the lower end of the static pressure range differs by quite a bit between blade passes 18 and 61, leading to the conclusion that the data was not periodic by 18 blade passes.

A plot of the surface pressure over the rotor blade at the beginning of blade pass 61 can be seen in Figure 4.22.
The surface pressure profile looks notably different, comparing the 18\textsuperscript{th} blade pass to the 61\textsuperscript{st} blade pass, further showing that periodicity had not been established after 18 passes.

### 4.2.2.2 Unsteady Solution – 10k, Full TEB

There were 18 blade passes worth of data taken for the case in which there was full TEB, and the engine was running at 10000 rpm. Pressure and velocity contours will be shown for blade passes 17 and 18, to compare the differences and comment on what the CFD is showing.

Figure 4.23 shows the total and static pressure contours at the beginning of the 17\textsuperscript{th} blade pass.
The total pressure ranges between 46.3 and 169 kPa, and the static pressure ranges between 29.2 and 137 kPa. By looking at the figure for the static pressure, it becomes apparent that the rotor blade is now at a positive angle of attack, with respect to the direction of the total velocity, because of the low pressure region, now on the upper half of the rotor blade. The positive angle of attack was calculated in Chapter 3. This also shows that the assignment of a downward velocity to the rotor mesh causes the expected response. These plots also show that there is a rise in both total and static pressure as the fluid flows through the system. The rise in total pressure is supported by the engine data presented by Kozak, (dissertation, 2000), and this supports a corresponding rise in static pressure.

Figure 4.24 shows the static pressure contours for the 17th blade pass, for one complete pass. Recall that there are 99 time steps required per blade pass. To conserve space, only 1 periodic repeat is shown.
The rotor mesh moves downward with each time step, as is expected from the assigned velocity to that portion of the mesh. A side-by-side comparison of the static pressure contours at the beginning of the 17th and 18th blade passes is shown in Figure 4.25.
The plots look nearly identical to the eye; however, the scales are slightly different. The upper boundary of pressure is the same, at 137 kPa, but the lower boundary is slightly different. It would not be known if the slight difference in contour plots would influence the blade stress response, until the calculations explained in the next section were performed.

**4.2.2.3 Unsteady Solution – 11k, No TEB**

Twenty blade passes worth of data were gathered for the case in which the engine speed was 11000 rpm, and trailing edge blowing was not used. Figure 4.26 shows the contour plots of the total and static pressures at the beginning of the 18th blade pass.
The maximum values for both the total and static pressure are higher for the 11k case than for the 10k case, when compared. This makes sense, as the engine speed increases, so does the compression ratio over the stage.

Figure 4.27 shows the plot of the static pressure on the surface of the rotor blade, at the beginning of the 18th blade pass.

The results for the pressure profile are slightly different at the beginning of the 19th blade pass. Figure 4.28 shows the total and static pressure profile on the surface of the rotor blade at this time.
Furthermore, the results for the maximum total and static pressure values, 182 kPa and 143 kPa, are just one kPa above the maximum values shown for blade pass 18. However, it will be shown in the data processing section of this chapter, that even this small amount is enough to prevent the stress profile from behaving periodically.

4.2.2.4 Unsteady Solution – 11k, Full TEB

There were 21 blade passes of data taken for the case in which the engine speed was at 11000 rpm, and the pressure deficit was completely compensated for, simulating full TEB.

The contour plots for the total pressure and the static pressure for this case can be seen in Figure 4.29. The stator and rotor mesh are no longer perfectly aligned, as they were for the 10 k case, because of the small amount of round-off error present in the time-step. This error was deemed not to be significant, because when the pressure data was processed, as described in the next section, data was processed for every single time step. The round-off error in the time step should not effect the determination of whether the stress profile shows periodicity.
The maximum values for the total and static pressures for this case are very similar to the maximum values found for the case with no TEB. This makes sense because the amplitude of the inlet pressure function was only a small percentage of the total pressure, on the order of 5%. Thus, there should not be a large difference in maximum pressure values between the TEB and no TEB cases. The maximum values are again higher than the values found for the 10k cases, showing consistency in the method used.

Figure 4.30 shows the last surface pressure distribution plot, for the beginning of the 18th blade pass. This profile looks similar to the profile found when trailing edge blowing was not present; thus, the data must be processed further to determine the sensitivity of the system to the inlet pressure function.
4.3 Data Processing

To determine if the CFD simulation scheme was sensitive enough to respond to the variable inlet pressure function assigned to the inlet boundary, it was necessary to further process the rotor blade surface pressure data. The following section describes the objectives and processes used in manipulating the data that was gathered throughout the CFD simulations. The data processing was performed using the commercially available spreadsheet tool, Microsoft Excel. The surface pressure data coordinates were exported to a data file for every single time step. There were 378 surface pressure data points for every time step, to fully describe the pressure profile on the upper and lower surface of the rotor blade.

To begin the processing, the exported surface pressure data sets were imported into the spreadsheet. A macro was then run to properly format the raw data, such that it could be copied and pasted into the calculation spreadsheet. There was a separate calculation spreadsheet for each time step. The process of importing, formatting, copying, and pasting was performed for every single time step of two complete blade passes. For the 10k case, this was equivalent to 198 time steps, and for the 11k case, was equivalent to 180 time steps.
4.3.1 Processing Objectives

The initial end goal of this work was to determine the fatigue life of the F109 rotor blade, when it was subjected to different amounts of trailing edge blowing. It was required to compile a stress versus time spectrum of discrete points, to be imported into the fatigue life prediction code. Thus, it was necessary to take the pressure data, and from that, calculate the corresponding blade stresses. The discrete pressure values from each time step had to be manipulated in a way such that the end result was one value for average blade stress. This discrete value could then be used in plotting the stress vs. time step plot, which could easily be transformed to a stress vs. time plot.

4.3.2 Processing Method

The data exported by Fluent was a set of discrete points, including the absolute surface pressure, and the x-location from the global coordinate system. The first step necessary in using the imported surface pressure data to find the blade stress was to resolve the upper and lower surface pressures to an equivalent concentrated load. This step is shown schematically in Figure 4.31. This figure does not represent actual data.

![Figure 4.31: Concentrated loads on rotor](image)

The method used to resolve the distributed load to a concentrated force was the trapezoidal method. The trapezoidal method can be used to find the area under a curve, when the data points forming the curve are discrete. To use the trapezoidal method, the total area under the curve is divided into trapezoids, and each individual area is found. The trapezoids are then summed to find the total area under the curve. This method is illustrated in Figure 4.32.
The area of each individual trapezoid was found using equation 4.2, and the total area under the curve was found using equation 4.3. The subscripts 1 and 2 correspond to the left and right sides of each trapezoid.

\[ dA = \frac{P_1 + P_2}{2} (x_2 - x_1) \]  
\[ A = \sum dA = F \]

Figure 4.33 shows a snapshot of the spreadsheet used to calculate the concentrated loads on the upper and lower surfaces using the trapezoidal method.

<table>
<thead>
<tr>
<th>x (m)</th>
<th>P (N/m)</th>
<th>P gage</th>
<th>dA</th>
<th>ydA</th>
<th>P (N/m)</th>
<th>P gage</th>
<th>dA</th>
<th>ydA</th>
</tr>
</thead>
<tbody>
<tr>
<td>7.5E-05</td>
<td>132,828</td>
<td>31,503</td>
<td>5.2915</td>
<td>0.000841</td>
<td>0.000202944</td>
<td>8532.5</td>
<td>-15943</td>
<td>-0.5288</td>
</tr>
<tr>
<td>0.000243</td>
<td>135,795</td>
<td>31,470</td>
<td>4.9622</td>
<td>0.00164</td>
<td>-0.000182817</td>
<td>6,47249</td>
<td>-36,500</td>
<td>-0.3166</td>
</tr>
<tr>
<td>0.000415</td>
<td>126,807</td>
<td>25,262</td>
<td>3.2362</td>
<td>0.001114</td>
<td>-0.00162355</td>
<td>10,6781</td>
<td>5.465</td>
<td>-1.3953</td>
</tr>
<tr>
<td>0.000579</td>
<td>116,328</td>
<td>15,003</td>
<td>1.345</td>
<td>0.00067</td>
<td>-0.000104323</td>
<td>49,676.7</td>
<td>-52546</td>
<td>-0.567</td>
</tr>
<tr>
<td>7.66E-04</td>
<td>107,496</td>
<td>617</td>
<td>0.7722</td>
<td>0.00005</td>
<td>-0.57E-05</td>
<td>124,589</td>
<td>23,264</td>
<td>-2.0589</td>
</tr>
</tbody>
</table>

The next step in resolving the surface pressure to a point load was to find the distance along the chord that was necessary to make the effects from the concentrated load equivalent to the distributed load. The equivalent distance for both the top and bottom force was found using equation 4.4.

\[ -\frac{x}{\sum dA} = \sum x \cdot dA \]

After finding the concentrated loads and the locations along the x-axis upon which they acted for both the upper and lower surfaces of the rotor blade, it was possible to combine the two forces to find one total force, equaling the force felt by the blade due to the surface static pressure. On the upper surface of the airfoil, the gage surface...
pressure was negative, tending to lift the surface in the positive-y direction. On the lower surface of the airfoil, the gage pressure was positive, meaning it acts to push on the lower surface, again acting in the positive y-direction. Thus, to find the resultant force, it was necessary to subtract the upper gage surface pressure from the lower gage surface pressure, to find the total pressure acting on the airfoil, as shown in equation 4.5.

\[ F_T = F_L - F_U \]  \hspace{1cm} (4.5)

The acting point along the x-axis for the total force was then found using equation 4.6.

\[ F_L \left( \frac{x_L}{c} \right) - F_U \left( \frac{x_U}{c} \right) = F_T \left( \frac{x_T}{c} \right) \]  \hspace{1cm} (4.6)

The term in the denominator, c, represents the rotor chord length.

After finding the resultant force due to the surface pressure on the rotor blade, an attempt was made to estimate the stresses experienced by the blade due to this force. It was concluded after the initial calculations that the largest stress that would be seen by the blade in bending, as described by equation 4.7.

\[ \sigma = \frac{Mc}{I} \]  \hspace{1cm} (4.7)

The Greek letter Sigma represents the bending stress, M represents the bending moment due to the resultant force, c represents the distance from the edge of the cross-section of the blade to its centroid, and the letter I represents the area moment of inertia. To find the moment, it was necessary to estimate the location of maximum stress along the blade. This was determined to occur at the hub of the blade, as it is the farthest from the hub. Thus, the moment arm was taken to be the distance from the hub of the blade to the measuring point, or 4.425 inches. The distance from the location of maximum stress to the centroid, represented by c, was found using equation 4.8.

\[ c = \frac{3}{8} t \]  \hspace{1cm} (4.8)

The letter t represents the maximum thickness of the blade. This method of computing c was crude, in that this equation is actually for a semi-elliptical area. However, because there was not specific information known about the actual cross-section of the blade, this was deemed acceptable at the time. The area moment of inertia
was found using equation 4.9, which also describes a half ellipse. Again, this equation was deemed acceptable at the time of calculation, because so little was known about the actual blade geometry.

\[ I = 0.1098ct^3 \] (4.9)

The moment was then able to be calculated, using the results found from the previous equations. The magnitude of the stresses calculated for this study were on the order of 1 GPa, which was assumed to be much higher than what is actually occurring, especially considering that this value is much beyond the fracture strength of most materials, including the titanium alloys generally used in turbine engines. There were multiple reasons that the calculated stresses on the blade were so high. One reason is that the blade was analyzed as a simple beam, with no twist. Had twist been incorporated, the effective moment in pure bending would probably be higher, yielding a more rigid structure. There also would have been forces acting in multiple directions, which may have caused in a lower resultant bending moment. Another reason that the stresses may have been so high was because there just was not enough available information about the rotor blade geometry. In reality, the blade might have had a higher area moment of inertia, resulting in a lower bending stress. There is little published data describing actual high cycle fatigue type stress magnitude, so the choice was made to continue calculating the stresses as outlined above, but scale the resulting stress forcing function by the maximum stress value. This would allow the function to be later scaled by an appropriate HCF stress magnitude.

The equations described above were inserted into a spreadsheet, so that the only necessary actions were to copy and paste each formatted data file into each spreadsheet. The final step in finding the stress forcing function was to create a series of points representing the entire blade pass. For each time step, the bending stress was pulled from its corresponding spreadsheet, and copied to a separate sheet. The values were then normalized by dividing each bending stress by the absolute highest bending stress from all time steps. The spectrum was then found by plotting the discrete points, with the time step number on the x-axis, and the normalized bending stress on the y-axis. The end product was a non-dimensional function, in a form for which it could be later scaled and imported into the fatigue life prediction code.
4.3.3 Processing Results

This section describes the results of the CFD data processing. The normalized stress spectra will be shown to prove that CFD can successfully model the surface pressure response of the variable inlet pressure profile. Spectra for the four cases described previously, for engine speeds of 10000 and 11000 rpm, with and without trailing edge blowing, will be shown. The stress spectra obtained never showed completely periodic results, however, they did point to the conclusion that CFD model is able to simulate the changing response in blade stress due to the unsteady pressure at the inlet.

4.3.3.1 Stress Spectra – 10k, No TEB

The case that the most data was gathered for was the 10k case, where TEB was not used. As mentioned in earlier sections, it was found that the data did not become periodic after 12 blade passes. For purposes of visualization only, the inlet pressure profile has been superimposed on the following plots. This will better illustrate that the shape of the stress spectrum mimics the shape of the pressure profile as the solution progresses. The values for the normalized stress spectra correspond to the scale on the left side of the plot, and the values for the normalized pressure profile correspond to the scale on the right side of the plot. The number along the x-axis corresponds to the time step number in the blade pass. Figure 4.34 shows the stress spectra for blade passes 18-19. The data does not appear to be periodic, and does not appear to follow the shape of the inlet pressure profile.
The difference between the highest point on the plot and the lowest point is over 7%. This number is extremely high, considering the normalized pressure profile differs by less than 1%.

Figure 4.35 shows the stress spectrum for blade passes 20 and 21.

There are notable differences between the plots for passes 18-19 and plots 20-21. The two different stress spectra in Figure 4.35 have very different magnitudes, although they have similar shapes. The difference between the local maxima is approximately three percent. This leads to the suspicion that the spectra are becoming more periodic in nature as the blade passes increase. The overall difference between the highest and
lowest point has decreased to approximately 6%, which also shows that the solution is increasing in accuracy as it progresses.

After the completion of 21 blade passes, and the realization that the spectra were still not exhibiting periodicity, the solution for this case was left to run for 40 more blade passes. It was hoped that after this many blade passes, the spectra would be periodic, with a variation in amplitude of less than 1%. Figure 4.36 shows the stress spectra for blade passes 61 and 62. After 60 blade passes, the data is showing periodicity. The amplitudes of the peaks for each blade pass differ by just 0.4%, and the troughs differ by 0.3%.

Furthermore, the shape of the stress spectrum strongly mimics the shape of the pressure profile, suggesting that the CFD model can adequately predict the surface pressure response when a variable pressure inlet is assigned, as would be the case when using trailing edge blowing. The difference between the absolute minimum and absolute maximum value has decreased to 1.5%, which is more reasonable, considering the small amplitude of the inlet total pressure profile. The difference between peaks of the stress spectrum versus the inlet pressure profile is due to the fact that the inlet is upstream of the point of calculation of the blade surface stress. Thus, the two profiles should not be in phase with each other.

Figure 4.36: Stress spectra, blade passes 61-62, 10k, no TEB
4.3.3.2 Stress Spectra – 10k, Full TEB

Figure 4.37 shows the stress spectrum of the 18th blade pass of the case in which the engine speed was 10000 rpm, and 100% wake-filling was used to compensate for the inlet pressure deficit. The expected results were that the stress spectrum should be a constant value, however, this was not so, as seen below.

![Stress Spectrum](image)

Figure 4.37: Stress spectrum, blade pass 18, 10k, no TEB

The difference between the absolute maximum value and absolute minimum value was 0.5%. It would be desirable to gather surface pressure data for enough blade passes such that the difference between the absolute maximum and minimum value was less than 0.1%. This would give a good estimation of how many blade passes were required for the system to stabilize. The number of required blade passes found could then be used as a starting point for gathering data for the cases in which trailing edge blowing was not used. This step would save some time, and increase the sensitivity of the spectrum to the inlet pressure function.

4.3.3.3 Stress Spectra – 11k, No TEB

Figure 4.38 shows plot of the stress spectra for blade passes 19 and 20, for the 11k, no TEB case. The plot clearly indicates that more blade passes are needed. The difference between the absolute minimum and absolute maximum was over 7%, and there is an increasing trend over the two blade passes.
Figure 4.38: Stress spectra, blade passes 19-20, 11k, no TEB

The good news that can be seen in the previous figure is that there are two inflection points, where the spectrum changes slope, approximately one blade pass apart. These inflection points are shown with vertical lines crossing through them on the plot. This leads to the conclusion that although the stress spectrum is not periodic, it is progressing toward becoming such. The stress spectra found for this case were very different from those found from the 10k case. After 19 blade passes, the spectra for the 10k cases were beginning to exhibit periodic characteristics. This leads to the conclusion that the as the speed of the engine increases, more blade passes may be required to acquire periodic data. Thus, more than 62 blade passes may be necessary to show data with as little variance as that shown in Figure 4.36.

4.3.3.4 Stress Spectra – 11k, Full TEB

Figure 4.39 shows the stress spectrum for blade passes 20 and 21, for the case in which the engine speed was 11000 rpm, and there was full TEB. Full TEB is shown in the figure by the straight line corresponding at a y-value of 1, when looking at the right-hand scale. The stress spectrum should also be a straight line, meaning it is constant, as there is no forcing function acting upon the rotor blade; it is rotating at a constant speed.
Figure 4.39: Stress spectra, blade passes 20-21, 11k, full TEB

The stress spectra do not show a periodic trend, leading to the conclusion that many more blade passes of data are required to accurately portray the system. The difference between the high and low value of the blade stress varies by more than 2%, further supporting the need for more data, as there should be no variance in the stress spectrum.

The preceding data support the hypothesis that Fluent is an adequate method of modeling stator-rotor interaction and trailing edge blowing. However, the results show that more blade passes for each case are necessary to come to the absolute conclusion that the software is sensitive enough to model this problem. It can be concluded with a high degree of confidence that the moving mesh model, as presented throughout this chapter, is the best method of modeling the 2-d stator rotor interaction problem. The data presented show that the normalized stress spectra will become more periodic as the data for more blade passes are gathered. Full periodicity will be reached, when the correct number of blade passes required is established.
5. Future Recommendations and Conclusions

The first part of this chapter will give recommendations and lessons learned, to any researcher that may take on this task in the future. An explanation will be given as to how the work outlined in this thesis saves much time and energy, for future researchers. This second part of this section will first give a summary of what was accomplished with this work, and will then give a broad statement of the conclusions that can be drawn from this study.

5.1 Future Recommendations

One of the biggest accomplishments of this work is that it gives a good starting point for the next researcher to take on the task of optimizing the use of trailing edge blowing, using computational fluid dynamics. The following section will give future recommendations for different steps in the process of modeling TEB, such that the mistakes that were made in this work are not made again.

5.1.1 Software Validation

One of the mistakes made in this work was that the cases chosen for validating the CFD software were not able to be effectively modeled using CFD. The purpose of validating the software was to prove user proficiency in using CFD, thus, the cases chosen should have been simple. The first case chosen, in finding the surface pressure on the surface of a NACA 0012 airfoil, was a good case for software validation. However, the low Reynolds number, flat plate case was a poor choice, because of the inability of the software to model the occurring aerodynamic phenomena occurring. The point of this case was to prove that the CFD software could model the effects of the viscous wakes on the airfoil surface pressure, and it was not known until attempting to model the case using CFD that it was not possible to acquire accurate data. Thus, time that could have been spent modeling the stator-rotor interactions in CFD, was spent on wind tunnel testing.

The CFD work completed on stator-rotor interactions for this study proved that the software is sensitive enough to register the changes in surface pressure due to the viscous wakes. This makes it unnecessary for a future researcher to complete wind
tunnel testing to gather surface pressure data in the presence of a viscous wake. However, if this step were to be taken again, it is recommended that the researcher run tests at a higher Reynolds number, so that there was no chance of the presence of a laminar separation bubble. This would require the fabrication of new test plates, because those used for this study were not rigid enough to withstand higher Reynolds number testing.

The CFD work done in modeling the stator-rotor interaction in the F109 turbofan engine showed that the software could model the effects of the stator wakes on the surface pressures of the rotors; however, there was no experimental data for which to compare the results of the simulation. Fortunately, there is much experimental data that exists for the static surface pressure on the surface of the stators, as affected by the upstream propagation of potential fields from the rotors, in the F109 turbofan engine [Fabian, 1999]. A suggestion for modeling this system would be to use a moving mesh technique, but to actually model the stator blade and the rotor blade. The rotor mesh could then be assigned a velocity in the vertical direction, as was detailed in Chapter 4. The surface pressure could be monitored on the surface of the stator blade, and compared to the experimental results. If the CFD data for the stator surface pressures compared well to the experimental results, the certainty in the accuracy of the results for the rotor surface pressures would increase, because of the similarities in modeling techniques.

5.1.2 Rotor Blade Modeling

Another problematic area in this work was that very little data was known about the geometry of the F109 rotor blades. This lack of knowledge led to many assumptions, and both the CFD work and the following stress analysis work were affected. The CFD work was affected because the assumptions about the rotor blade cross-sectional profile may not have been accurate. Even a small amount of error in the design of the cross-section could have had a large effect on the CFD results. It would be absolutely necessary to obtain an accurate representation of the cross-section, to certify that the CFD results are valid.

The stress analysis of the blade was also affected by the assumptions made about the blade geometry. It was known from previous work that the F109 rotor blades were
twisted. The angles between the blade chord and the azimuthal axis were known at the hub and tip of the blade, but nothing was known about the rate of twist. This made it necessary to assume a rate of twist, to determine the blade angle of attack for the CFD simulation.

Instead of calculating the bending stresses using a spreadsheet, and assuming that the bending stress was the driving factor, the system could be modeled and stresses could be found using a commercial stress analysis software such as ANSYS. The surface pressure data found using CFD could be imported into the stress analysis software, presumably giving a more accurate representation of the actual stresses in the rotor blade. Furthermore, the effect of all stress modes would be seen, instead of the bending stress alone.

The 2-d CFD surface pressure data was assumed to be constant along the span of the rotor blades, which is not realistic, because of the twist along the span. Because of this assumption, the rotor blades were modeled as simple beams, without any twist. This led to the stresses in the beams being much higher than the ultimate strength of the material. Incorporating twist into the beam would have decreased the bending stresses, and would have given a more accurate representation of the system. To achieve an accurate representation of the system, a 3-d model would have been necessary for both the CFD simulation and the stress analysis. Unfortunately, the experimental TEB data is only 2-d, making this impossible.

5.1.3 CFD Simulation

For the previous simulations, the stress spectra were beginning to show periodic characteristics, but were not yet fully mature. The periodic characteristics can be improved with more iterations of the solution, requiring more time. As system computing power increases, the number of iterations required for periodic solutions will take less time, allowing for further exploration of different cases, including those using variable TEB.

Different amounts of TEB can be easily incorporated into the inlet boundary condition user defined function, by specifying a minimum pressure inside the wake. This value can then be specified within the UDF code as a variable, and the values within the
function falling below this limit will be truncated. The process for determining the cutoff value is as follows. First, the inlet pressure profile must be integrated using the trapezoidal method, to find the total area under the curve that corresponds to a pressure deficit. Then, a truncation value must be established, and the area under the curve must again be found. The percentage that the resulting area under the curve is to the total percentage of area under the curve, subtracted from the original area, can be considered the amount of trailing edge blowing that is used. This means that if the user wishes to simulate 25% trailing edge blowing, the truncation value must be set such that the new area under the curve is equal to 75% of the original area under the curve. For clarification, this process is illustrated in Figure 5.1.

![Figure 5.1: TEB pressure inlet function](image)

The free stream pressure corresponds to the atmospheric pressure of 101325 kPa. The recommendations outlined in this section will save future researchers much time and frustration, because a further starting point for the next round of work has been established.
5.2 Conclusions

The original objective of this work was to use computational fluid dynamics to model stator-rotor interactions, including those incorporating variable amounts of trailing edge blowing, in the F109 turbofan engine. A time-dependent, periodic, rotor blade stress spectrum would be found using the CFD data, which could then be imported into a commercial fatigue life estimation code. This would allow the determination of the rotor blade fatigue life, for variable amounts of trailing edge blowing, with the hope that small amounts of trailing edge blowing would yield large increases in fatigue life.

After much work was completed, the scope was narrowed to focus only on modeling the stator-rotor interactions, without trailing edge blowing, and then with full trailing edge blowing, as if the upstream stators were removed. The determination of the blade fatigue life was removed from the scope, and left for future investigators. The purpose of the narrowed scope was to examine if the CFD software was sensitive enough to detect the slight changes in rotor blade surface pressure, due to the assignment of a periodic inlet pressure function. The inlet pressure function represented the pressure profile created by the viscous wakes propagating downstream from the stators in the actual engine. Another purpose was to devise a method of calculating rotor blade stress, using the exported blade surface values.

It can be concluded from the research done in this study that the CFD can in fact model the slight changes in the surface pressure of the rotor blade, caused by the variable inlet pressure function. The data presented in Chapter 4 for blade passes 61 and 62 the case in which the engine speed was 10000 rpm and there was no TEB, show that the time-dependent blade stress spectra are becoming periodic. It can be assumed that with enough iterations of the solution, the blade surface pressure would become completely periodic, with the same period and amplitude for each blade pass. Furthermore, the spectra mimic the characteristics of the inlet pressure function, showing that the CFD can model the effects of the viscous wake profile caused by the upstream stators.

It can be concluded with reasonable certainty that if the inlet pressure profiles were modified to incorporate various amounts of trailing edge blowing, the effect would still be seen in the surface pressure data of the rotor blade. This would allow the calculation of the stresses in the blade, and the benefit of using less than 100% TEB
could be explored further. The fatigue life predictions obtained from future work would establish a guess at the optimal level of trailing edge blowing. However, in the opinion of this researcher, the only way to obtain a true representation of the stator-rotor interactions in the F109 turbofan engine, using CFD, would be to model the system in 3-dimensions, using 3-dimensional experimental data.
References


Basu, Suman, Personal communication.


Fleeter, S., Zhou, C., Houstis, E., Rice, J., “Fatigue Life of Turbomachine Blading”.


http://www.aircraftengineinedesign.com/F109pics.html
Appendix A

A.1 Matlab Code

Code for 10k, no TEB

```matlab
y = no_teb_10k_pressure_profile(:,1);
v = no_teb_10k_pressure_profile(:,2);
figure(1)
plot(v,y)

n_points = length(y)
samp_freq = abs(1/(y(2)-y(1)))
high_freq = 0.5*samp_freq
low_freq = abs(1/(y(length(y)-1)-(y(1))))

vv = fft(v); % Compute DFT of x
phase = angle(vv);

f = (0:length(vv)-1)'/length(vv)*samp_freq; % Frequency vector
figure(2)
plot(f,phase)
m = abs(vv); % Magnitude
figure(3)
m_scaled = m/length(v);
plot(f,m_scaled)

sort(m_scaled);
[Y,I] = sort(m_scaled)

y0 = 0;

profile = m_scaled(1) + 2*m_scaled(2)*cos((2*pi*f(2)*(y-y0))+phase(2))...
       + 2*m_scaled(3)*cos((2*pi*f(3)*(y-y0))+phase(3))...
       + 2*m_scaled(4)*cos((2*pi*f(4)*(y-y0))+phase(4))...
       + 2*m_scaled(5)*cos((2*pi*f(5)*(y-y0))+phase(5))...
       + 2*m_scaled(6)*cos((2*pi*f(6)*(y-y0))+phase(6))...
       + 2*m_scaled(7)*cos((2*pi*f(7)*(y-y0))+phase(7))...
       + 2*m_scaled(8)*cos((2*pi*f(8)*(y-y0))+phase(8))...
       + 2*m_scaled(9)*cos((2*pi*f(9)*(y-y0))+phase(9))...
       + 2*m_scaled(10)*cos((2*pi*f(10)*(y-y0))+phase(10))...
       + 2*m_scaled(11)*cos((2*pi*f(11)*(y-y0))+phase(11))...
       + 2*m_scaled(12)*cos((2*pi*f(12)*(y-y0))+phase(12))...
       + 2*m_scaled(13)*cos((2*pi*f(13)*(y-y0))+phase(13))...
       + 2*m_scaled(14)*cos((2*pi*f(14)*(y-y0))+phase(14))...
       + 2*m_scaled(15)*cos((2*pi*f(15)*(y-y0))+phase(15))...
```

131
\[ +2 \cdot m_{\text{scaled}}(9) \cdot \cos((2 \cdot \pi \cdot f(9) \cdot (y-y_0)) + \text{phase}(9)) \]
\[ +2 \cdot m_{\text{scaled}}(15) \cdot \cos((2 \cdot \pi \cdot f(15) \cdot (y-y_0)) + \text{phase}(15)) \]
\[ +2 \cdot m_{\text{scaled}}(28) \cdot \cos((2 \cdot \pi \cdot f(28) \cdot (y-y_0)) + \text{phase}(28)) \]
\[ +2 \cdot m_{\text{scaled}}(17) \cdot \cos((2 \cdot \pi \cdot f(17) \cdot (y-y_0)) + \text{phase}(17)) \]
\[ +2 \cdot m_{\text{scaled}}(27) \cdot \cos((2 \cdot \pi \cdot f(27) \cdot (y-y_0)) + \text{phase}(27)) \]
\[ +2 \cdot m_{\text{scaled}}(18) \cdot \cos((2 \cdot \pi \cdot f(18) \cdot (y-y_0)) + \text{phase}(18)) \]
\[ +2 \cdot m_{\text{scaled}}(14) \cdot \cos((2 \cdot \pi \cdot f(14) \cdot (y-y_0)) + \text{phase}(14)) \]
\[ +2 \cdot m_{\text{scaled}}(19) \cdot \cos((2 \cdot \pi \cdot f(19) \cdot (y-y_0)) + \text{phase}(19)) \]
\[ +2 \cdot m_{\text{scaled}}(26) \cdot \cos((2 \cdot \pi \cdot f(26) \cdot (y-y_0)) + \text{phase}(26)) \]

```matlab
figure(4)
plot(profile,y,'r')
hold on
plot(v,y,'.')
```

**Code for 11k, no TEB**

```matlab
y=no_teb_11k_pressure_profile(:,1);
v=no_teb_11k_pressure_profile(:,2);
figure(1)
plot(v,y)

n_points=length(y)
samp_freq=abs(1/(y(2)-y(1)));
high_freq=0.5*samp_freq
low_freq=abs(1/(y(length(y)-1)-(y(1))))

vv = fft(v);
phase = angle(vv);

f = (0:length(vv)-1)'/length(vv)*samp_freq;

m = abs(vv);

sort(m_scaled);
[Y,I]=sort(m_scaled)

y0=0;

profile=m_scaled(1)...
\[ +2 \cdot m_{\text{scaled}}(2) \cdot \cos((2 \cdot \pi \cdot f(2) \cdot (y-y_0)) + \text{phase}(2)) \]
\[ +2 \cdot m_{\text{scaled}}(3) \cdot \cos((2 \cdot \pi \cdot f(3) \cdot (y-y_0)) + \text{phase}(3)) \]
\[ +2 \cdot m_{\text{scaled}}(5) \cdot \cos((2 \cdot \pi \cdot f(5) \cdot (y-y_0)) + \text{phase}(5)) \]
```
\[ +2 \cdot m_{\text{scaled}}(7) \cdot \cos((2\pi \cdot f(7) \cdot (y-y_0)) + \text{phase}(7)) \]
\[ +2 \cdot m_{\text{scaled}}(4) \cdot \cos((2\pi \cdot f(4) \cdot (y-y_0)) + \text{phase}(4)) \]
\[ +2 \cdot m_{\text{scaled}}(9) \cdot \cos((2\pi \cdot f(9) \cdot (y-y_0)) + \text{phase}(9)) \]
\[ +2 \cdot m_{\text{scaled}}(26) \cdot \cos((2\pi \cdot f(26) \cdot (y-y_0)) + \text{phase}(26)) \]
\[ +2 \cdot m_{\text{scaled}}(10) \cdot \cos((2\pi \cdot f(10) \cdot (y-y_0)) + \text{phase}(10)) \]
\[ +2 \cdot m_{\text{scaled}}(6) \cdot \cos((2\pi \cdot f(6) \cdot (y-y_0)) + \text{phase}(6)) \]
\[ +2 \cdot m_{\text{scaled}}(8) \cdot \cos((2\pi \cdot f(8) \cdot (y-y_0)) + \text{phase}(8)) \]
\[ +2 \cdot m_{\text{scaled}}(12) \cdot \cos((2\pi \cdot f(12) \cdot (y-y_0)) + \text{phase}(12)) \]
\[ +2 \cdot m_{\text{scaled}}(11) \cdot \cos((2\pi \cdot f(11) \cdot (y-y_0)) + \text{phase}(11)) \]
\[ +2 \cdot m_{\text{scaled}}(27) \cdot \cos((2\pi \cdot f(27) \cdot (y-y_0)) + \text{phase}(27)) \]
\[ +2 \cdot m_{\text{scaled}}(28) \cdot \cos((2\pi \cdot f(28) \cdot (y-y_0)) + \text{phase}(28)) \]
\[ +2 \cdot m_{\text{scaled}}(29) \cdot \cos((2\pi \cdot f(29) \cdot (y-y_0)) + \text{phase}(29)) \]
\[ +2 \cdot m_{\text{scaled}}(25) \cdot \cos((2\pi \cdot f(25) \cdot (y-y_0)) + \text{phase}(25)) \]
\[ +2 \cdot m_{\text{scaled}}(14) \cdot \cos((2\pi \cdot f(14) \cdot (y-y_0)) + \text{phase}(14)) \]
\[ +2 \cdot m_{\text{scaled}}(30) \cdot \cos((2\pi \cdot f(30) \cdot (y-y_0)) + \text{phase}(30)) \]
\[ +2 \cdot m_{\text{scaled}}(16) \cdot \cos((2\pi \cdot f(16) \cdot (y-y_0)) + \text{phase}(16)) \]
\[ +2 \cdot m_{\text{scaled}}(24) \cdot \cos((2\pi \cdot f(24) \cdot (y-y_0)) + \text{phase}(24)) \];

figure(4)
plot(profile, y, 'r')
hold on
plot(v, y, '.')

**A.2 Fluent UDF Code**

**Code for udfs, 10k, no TEB**

```c
#include "udf.h" /* header file - necessary */
#include "mem.h"

DEFINE_PROFILE(no_teb_10k_pressure_inlet, thread, nv)
    /* note the name of the function called NO_TEB_10k is defined here */
    /* all UDF's begin with a define Macro */
    /* NO_TEB_10k will be identified through the Fluent BC panel */
    /* thread, and nv are dynamic references and is used for internal book keeping */
{
    float x[3]; /* variable to hold position values*/
        /* in C index starts at 0 - hence the three variables are x[0], x[1], x[2] */
    float y; /* definition of a single precision real variable y */
```
float pi;
face_t f;       /* a structure defined in "udf.h" by fluent */

/* coefficients for cosine inlet function */
float a0,a1,a2,a3,a4,a5,a6,a7,a8,a9,a10,a11,a12,a13,a14,a15,a16,a17,a18,a19,a20;
float f1,f2,f3,f4,f5,f6,f7,f8,f9,f10,f11,f12,f13,f14,f15,f16,f17,f18,f19,f20;
float p1,p2,p3,p4,p5,p6,p7,p8,p9,p10,p11,p12,p13,p14,p15,p16,p17,p18,p19,p20;

begin_f_loop(f,thread)    /* a looping MACRO used to access all cells or cell faces */
{
    F_CENTROID(x,f,thread);  /* a MACRO that assigns Cell positions to x */
    y = x[1];
    pi=3.14159;

    a0=100.962;  f1=22.8550;  p1=-1.0867;
    a1=66.1366;  f2=91.4201;  p2=-2.5083;
    a2=31.7971;  f3=68.5651;  p3=-.73120;
    a3=29.9774;  f4=45.7101;  p4=-2.6752;
    a4=28.3677;  f5=369.9409; p5=-.8261;
    a5=20.1083;  f6=159.9852; p6=-1.8428;
    a6=17.2224;  f7=205.6953; p7=2.1259;
    a7=13.1943;  f8=251.4054; p8=2.4438;
    a8=9.6001;   f9=228.5503; p9=-.6077;
    a9=7.9397;   f10=114.2752; p10=.5867;
    a10=7.7289;  f11=182.8403; p11=2.7308;
    a11=7.3173;  f12=137.1302; p12=-2.3181;
    a12=6.9766;  f13=319.9705; p13=1.7704;
    a13=6.6289;  f14=617.0859; p14=2.0378;
    a14=4.6108;  f15=365.6805; p15=1.3162;
    a15=3.8219;  f16=594.2308; p16=2.1592;
    a16=3.6834;  f17=388.5356; p17=1.9588;
    a17=3.5669;  f18=297.1154; p18=-.3164;
    a18=3.3131;  f19=411.3906; p19=1.3767;
    a19=3.0927;  f20=571.3758; p20=2.1030;
F_PROFILE(f, thread, nv) = a0 + 2*a1*cos(2*pi*f1*y+p1) 
+ 2*a2*cos(2*pi*f2*y+p2) + 2*a3*cos(2*pi*f3*y+p3) 
+ 2*a4*cos(2*pi*f4*y+p4) + 2*a5*cos(2*pi*f5*y+p5) 
+ 2*a6*cos(2*pi*f6*y+p6) + 2*a7*cos(2*pi*f7*y+p7) 
+ 2*a8*cos(2*pi*f8*y+p8) + 2*a9*cos(2*pi*f9*y+p9) 
+ 2*a10*cos(2*pi*f10*y+p10) + 2*a11*cos(2*pi*f11*y+p11) 
+ 2*a12*cos(2*pi*f12*y+p12) + 2*a13*cos(2*pi*f13*y+p13) 
+ 2*a14*cos(2*pi*f14*y+p14) + 2*a15*cos(2*pi*f15*y+p15) 
+ 2*a16*cos(2*pi*f16*y+p16) + 2*a17*cos(2*pi*f17*y+p17) 
+ 2*a18*cos(2*pi*f18*y+p18) + 2*a19*cos(2*pi*f19*y+p19) 
+ 2*a20*cos(2*pi*f20*y+p20);

/* the above MACRO assigns the profile to the face f */

end_f_loop(f, thread)

DEFINE_PROFILE(pressure_outlet, t, i) {
    Thread *t0;
    cell_t c0;
    face_t f;

    /* neighboring cells of face f, and their corresponding threads */

    begin_f_loop(f, t) {
        t0 = THREAD_T0(t);
        c0 = F_C0(f, t);
        F_PROFILE(f, t, i) = C_P(c0, t0);
    }
    end_f_loop(f, t)
}

DEFINE_PROFILE(temperature_outlet, t, i) {
    Thread *t0;
    cell_t c0;
    face_t f;
/* neighboring cells of face f, and their corresponding threads */

begin_f_loop(f,t)
{
    t0 = THREAD_T0(t);
    c0 = F_C0(f,t);
    F_PROFILE(f,t,i)=C_T(c0,t0);
}
end_f_loop(f,t)

Code for udfs, 11k, no TEB

#include "udf.h" /* header file - necessary */
#include "mem.h"

DEFINE_PROFILE(no_teb_llk_pressure_inlet, thread , nv)
    /* note the name of the function called NO_TEB_10k is defined here */
    /* all UDF's begin with a define Macro */
    /* NO_TEB_11k will be identified through the Fluent BC panel */
    /* thread, and nv are dynamic references and is used for internal book keeping */
{
    float x[3];    /* variable to hold position values*/
        /* in C index starts at 0 - hence the three variables are x[0], x[1], x[2] */

    float y;      /* definition of a single precision real variable y */
    float pi;
    face_t f;     /* a structure defined in "udf.h" by fluent */

    /* coefficients for cosine inlet function */
    float a0,a1,a2,a3,a4,a5,a6,a7,a8,a9,a10,a11,a12,a13,a14,a15,a16,a17,a18,a19,a20;
    float f1,f2,f3,f4,f5,f6,f7,f8,f9,f10,f11,f12,f13,f14,f15,f16,f17,f18,f19,f20;
float
p1,p2,p3,p4,p5,p6,p7,p8,p9,p10,p11,p12,p13,p14,p15,p16,p17,
p18,p19,p20;

begin_f_loop(f,thread) /* a looping MACRO used to access all cells or cell faces */
{
    F_CENTROID(x,f,thread);  /* a MACRO that assigns Cell positions to x */
y = x[1];
    pi = 3.14159;

    a0 = 100666.42;
a1 = 144.6575; f1 = 22.8550; pl = 1.1582;
a2 = 47.6355; f2 = 45.7101; p2 = -2.5673;
a3 = 42.1205; f3 = 91.4201; p3 = -2.9241;
a4 = 27.5719; f4 = 137.1302; p4 = -0.4226;
a5 = 24.2675; f5 = 68.5651; p5 = -2.6638;
a6 = 23.4073; f6 = 182.8403; p6 = 2.6908;
a7 = 20.0214; f7 = 571.3758; p7 = -2.3234;
a8 = 18.0148; f8 = 205.6953; p8 = 1.4259;
a9 = 15.8974; f9 = 114.2752; p9 = 1.0304;
a10 = 14.6417; f10 = 251.4054; pl0 = -0.5384;
a11 = 14.4277; f11 = 159.9852; p11 = -2.6698;
a12 = 11.9274; f12 = 228.5503; p12 = -1.8941;
a13 = 9.7796; f13 = 594.2308; p13 = -1.0456;
a14 = 8.1967; f14 = 617.0859; p14 = 0.9183;
a15 = 3.7844; f15 = 639.9409; p15 = 0.9374;
a16 = 3.6278; f16 = 548.5208; p16 = -2.8562;
a17 = 3.3479; f17 = 297.1154; p17 = 3.0662;
a18 = 2.5218; f18 = 662.7959; p18 = 1.0896;
a19 = 2.4527; f19 = 342.8255; p19 = 0.9087;
a20 = 2.1274; f20 = 274.2604; p20 = 1.2410;

F_PROFILE(f, thread, nv) = a0 + 2*a1*cos(2*pi*f1*y+p1) + 2*a2*cos(2*pi*f2*y+p2) + 2*a3*cos(2*pi*f3*y+p3) + 2*a4*cos(2*pi*f4*y+p4) + 2*a5*cos(2*pi*f5*y+p5) + 2*a6*cos(2*pi*f6*y+p6) + 2*a7*cos(2*pi*f7*y+p7) + 2*a8*cos(2*pi*f8*y+p8) + 2*a9*cos(2*pi*f9*y+p9) + 2*a10*cos(2*pi*f10*y+p10) + 2*a11*cos(2*pi*f11*y+p11) + 2*a12*cos(2*pi*f12*y+p12) + 2*a13*cos(2*pi*f13*y+p13) + 2*a14*cos(2*pi*f14*y+p14) + 2*a15*cos(2*pi*f15*y+p15) + 2*a16*cos(2*pi*f16*y+p16) + 2*a17*cos(2*pi*f17*y+p17) + 2*a18*cos(2*pi*f18*y+p18) + 2*a19*cos(2*pi*f19*y+p19)
$$+2*a_{20}\cos(2*pi*f_{20}*y+p_{20});$$

/* the above MACRO assigns the profile to the face f */

end_f_loop(f,thread)
}

DEFINE_PROFILE(pressure_outlet, t, i)
{
  Thread *t0;
  cell_t c0;
  face_t f;

  /* neighboring cells of face f, and their corresponding threads */
  begin_f_loop(f,t)
  {
    t0 = THREAD_T0(t);
    c0 = F_C0(f,t);
    F_PROFILE(f,t,i)=C_P(c0,t0);
  }
  end_f_loop(f,t)
}

DEFINE_PROFILE(temperature_outlet, t, i)
{
  Thread *t0;
  cell_t c0;
  face_t f;

  /* neighboring cells of face f, and their corresponding threads */
  begin_f_loop(f,t)
  {
    t0 = THREAD_T0(t);
    c0 = F_C0(f,t);
    F_PROFILE(f,t,i)=C_T(c0,t0);
  }
  end_f_loop(f,t)
}
Appendix B

\[ \alpha = -4 \]
\[ M = 0.1 \]

- inside symbols indicates lower surface pressures

\[ \alpha = -4 \]
\[ M = 0.2 \]

- inside symbols indicates lower surface pressures

\[ \alpha = -2 \]
\[ M = 0.1 \]

- inside symbols indicates lower surface pressures
alpha = -2
M = 0.2

- inside symbols indicates lower surface pressures

alpha = 0
M = 0.1

- inside symbols indicates lower surface pressures

alpha = 0
M = 0.2

- inside symbols indicates lower surface pressures

alpha = 4
M = 0.1

- inside symbols indicates lower surface pressures
alpha = 4
M = 0.2

- inside symbols indicates lower surface pressures

alpha = 8
M = 0.1

- inside symbols indicates lower surface pressures

alpha = 8
M = 0.2

- inside symbols indicates lower surface pressures
alpha = 12
M = 0.1

- inside symbols indicates lower surface pressures

\[ C_p \]

alpha = 12
M = 0.2

- inside symbols indicates lower surface pressures

\[ C_p \]

alpha = 16
M = 0.1

- inside symbols indicates lower surface pressures

\[ C_p \]
alpha = 16
\[ M = 0.2 \]

- inside symbols indicates lower surface pressures

alpha = 20
\[ M = 0.2 \]

- inside symbols indicates lower surface pressures