Numerical Investigation of Subsonic Axial-Flow Tandem Airfoils for a Core Compressor Rotor

by

Jonathan McGlumphy

Dissertation submitted to the Faculty of the Virginia Polytechnic Institute and State University in partial fulfillment of the requirements for the degree of

Doctor of Philosophy

in

Mechanical Engineering

Approved:

Dr. Wing F. Ng (Chair)
Dr. Walter F. O’Brien
Dr. Clint L. Dancey
Dr. Pavlos P. Vlachos
Dr. Saad A. Ragab
Dr. Steven R. Wellborn

Defense: January 15, 2008
Blacksburg, Virginia

Key Words: gas turbine, compressor, tandem airfoil, cascade, high-loading, tandem blade, rotor, computational

Copyright by Jonathan McGlumphy 2008
Abstract

Numerical Investigation of Subsonic Axial-Flow Tandem Airfoils for a Core Compressor Rotor

Jonathan McGlumphy
Mechanical Engineering Department
Virginia Tech

The tandem airfoil has potential to do more work as a compressor blade than a single airfoil without incurring significantly higher losses. Although tandem blades are sometimes employed as stators, they have not been used in any known commercial rotors. The goal of this work is to evaluate the aerodynamic feasibility of using a tandem rotor in the rear stages of a core compressor. As such, the results are constrained to shock-free, fully turbulent flow. The work is divided into 2-D and 3-D simulations. The 3-D results are subject to an additional constraint: thick endwall boundary layers at the inlet.

Existing literature data on tandem airfoils in 2-D rectilinear cascades have been compiled and presented in a Lieblein loss versus loading correlation. Large scatter in the data gave motivation to conduct an extensive 2-D CFD study evaluating the overall performance as a function of the relative positions of the forward and aft airfoils. CFD results were consistent with trends in the open literature, both of which indicate that a properly designed tandem airfoil can outperform a comparable single airfoil on- and off-design. The general agreement of the CFD and literature data serves as a validation for the computational approach.

A high hub-to-tip ratio 3-D blade geometry was developed based upon the best-case tandem airfoil configuration from the 2-D study. The 3-D tandem rotor was simulated in isolation in order to scrutinize the fluid mechanisms of the rotor, which had not previously been well documented. A geometrically similar single blade rotor was also simulated under the same conditions for a baseline comparison. The tandem rotor was found to outperform its single blade counterpart by attaining a higher work coefficient, polytropic efficiency and numerical stall margin. An examination of the tandem rotor fluid mechanics revealed that the forward blade acts in a similar manner to a conventional rotor. The aft blade is strongly dependent upon the flow it receives from the forward blade, and tends to be more three-dimensional and non-uniform than the forward blade.
Acknowledgements

I would first like to express my utmost gratitude to my advisor, Dr. Wing-Fai Ng, for providing extraordinary levels of academic and professional guidance throughout my four and a half years at Virginia Tech. His mentoring has taught me the valuable lesson of always stepping back and looking at the “big picture” before hastily rushing into action. Also due the same level of gratitude are Dr. Steven Wellborn and Severin Kempf at Rolls-Royce Corporation. Both of them provided immense technical insight from an industry standpoint. I sincerely profess that without the help of the three aforementioned gentlemen this work would not have been accomplished in such a quality manner. Mike Howard at Rolls-Royce, Derby was also very helpful in collecting Smith chart data. I also thank Drs. Walter O’Brien, Clinton Dancey, Pavlos Vlachos, and Saad Ragab for serving on my advisory committee and for their valuable input into my thesis work.

The project was sponsored by Rolls-Royce Corporation, and I am immensely grateful for their financial support. I also would like to express thanks to the members of the Fan & Compressor Aerodynamics group in Indianapolis for making me feel very welcome during my twenty weeks of residence there.

Additional funding also came from the University Turbine Systems Research Industrial Fellowship, and the Virginia Space Grant Consortium. I am very grateful to both of those organizations for their support of my work.

Given the computational nature of the project, I must give due credit to Jamie Archual and Ben Poe in the Mechanical Engineering department for always being there to help whenever I had computer-related difficulties. A special thanks is also due to Luke Scharf, who specifically helped me to setup, operate, and maintain the cluster of computers that I used to perform many of my simulations.

I was always proud to be a member Dr. Ng’s research group, all of whom provided help and friendship throughout my time at Virginia Tech. Particular thanks goes to Austin Smith, who first showed me the ropes when I arrived, and Hugh Hill, whose immense patience and sense of humor helped me remain in high spirits.

I also wish to acknowledge Nick Cardwell and Sundar Narayan, who proved to be excellent study partners for the qualifying exam.
I would like to thank Rob English and the entire staff of the Rivermill Map Company. They provided me with a community place to celebrate my successes, drown my sorrows, and enjoy fellowship with my brethren in the suffering.

I am also very grateful for my father, David McGlumphy, who stepped in to help at moments of greatest need throughout my entire collegiate experience. Finally, I wish to thank my wife, Shannon, and daughter, Angel. Their love and patience saw me through the good times and the bad times of my Ph.D. program, and I am forever in their debt.

Jonathan McGlumphy

Virginia Polytechnic Institute and State University
January 2008
# Table of Contents

Abstract ...........................................................................................................................ii  
Acknowledgments .........................................................................................................iii  
Table of Contents ...........................................................................................................v  
List of Figures ..............................................................................................................viii  
List of Tables ..................................................................................................................x  
Nomenclature .................................................................................................................xi  
1. Introduction ................................................................................................................1  
  1.1 Background ............................................................................................................1  
  1.2 Specific Objectives of Current Work ......................................................................3  
    1.2.1 2-D Cascade Study .........................................................................................3  
    1.2.2 Selection of Geometry for 3-D Study ..............................................................3  
    1.2.3 3-D Rotor Study ............................................................................................3  
2. 2-D Tandem Airfoils ..................................................................................................4  
  2.1 2-D Cascade Literature Review ............................................................................4  
  2.2 2-D CFD Study .....................................................................................................8  
    2.2.1 Simple Design Rule .......................................................................................8  
    2.2.2 Tandem Airfoil CFD Setup ............................................................................10  
    2.2.3 Results: On-Design (Minimum Loss) ...........................................................12  
      2.2.3.1 Percent Pitch Variation .........................................................................12  
      2.2.3.2 Axial Overlap Variation .........................................................................19  
      2.2.3.3 Airfoil Loading Split ...............................................................................19  
    2.2.4 Results: Off-Design (Incidence Range) .........................................................21  
  2.3 General Summary of 2-D Tandem Study ...............................................................23  
3. 3-D Tandem Rotors ...................................................................................................24  
  3.1 3-D Rotor Literature Review .................................................................................24  
  3.2 3-D CFD Study .....................................................................................................27  
    3.2.1 Tandem Rotor Constraints and Design Goal .................................................28  
    3.2.2 Tandem Rotor Geometry ..............................................................................29  
    3.2.3 Single Blade Rotor Geometry ........................................................................30  
    3.2.4 CFD Setup and Procedure (3-D Specific) ......................................................30  
    3.2.5 Results: Overall Performance .......................................................................33  
    3.2.6 Results: Tandem Rotor Fluid Mechanics .......................................................36  
      3.2.6.1 Forward Blade .........................................................................................37  

v
3.2.6.2 Aft Blade ...................................................................................41

3.3 General Summary of 3-D Tandem Study ........................................49

4. Conclusions ........................................................................................50

4.1 Tandem Airfoil 2-D Cascade Flow .................................................50

4.2 Tandem Rotor 3-D Flow ................................................................51

4.3 Recommendations for Future Work ...............................................52

Appendix A In-Depth Review of Tandem Airfoils in 2-D Cascade ..........53

Appendix B Cascade Frame of Reference and AVDR ..............................57

Appendix C Airfoil Geometries for 2-D Study .........................................59

C.1 Development of the Simple Design Rule for Tandem Airfoils ..........59

C.2 Geometries of the Single Airfoils ....................................................61

C.3 2-D Tandem Airfoil Geometries for the Percent Pitch /
Axial Overlap Study ..............................................................................62

Appendix D Computational Methods (Additional Information) .................63

D.1 Overview of CFD code: ADPAC ..................................................63

D.1.1 Mesh Types .............................................................................63

D.1.1 Multi-Grid Scheme ..................................................................66

D.2 General CFD Procedure .............................................................67

D.2.1 Mesh Generation .....................................................................67

D.2.2 Grid Independence ...................................................................69

D.2.3 Convergence Criteria ...............................................................69

D.2.4 Solution Post-Processing .........................................................72

Appendix E 2-D Tandem Airfoil Loading Split ........................................73

Appendix F Review of Other Dual-Airfoil Type Rotors ............................75

F.1 Splitter Blades in Centrifugal Rotors ............................................75

F.2 Tandem Blades in Centrifugal Rotors ..........................................76

F.3 Splitter Blades in Axial-Flow Rotors ............................................77

F.4 Conical-Flow Tandem Rotor .......................................................77

Appendix G 2-D Mixed-Out Losses ....................................................80

G.1 Governing Equations (Derivation) .............................................81

G.2 Governing Equations (Q-Variable Form) .....................................84

G.3 Governing Equations (Discretized Form) ....................................86

G.4 Calculated Mixed-Out Losses (Single and Tandem) ....................89

Appendix H 3-D Rotor Flow Nonuniformities .......................................91
H.1 Definition of Mass-Averaged and Area-Averaged Loss Coefficients ...........91
H.2 Differences in Loss Coefficients in 3-D Flow ....................................................92
References ....................................................................................................................95
Vita ...............................................................................................................................103
List of Figures

Figure 1      2-D Profile view of highly loaded axial-flow single and tandem airfoil ........2
Figure 2      Tandem airfoil 2-D geometric parameters .....................................................4
Figure 3      Lieblein chart of pertinent literature data .......................................................7
Figure 4      Single airfoil CFD vs. tandem simple design rule (NACA-65) .....................10
Figure 5      Example multi-block H-mesh used for tandem airfoil simulations
               (2-D view) .....................................................................................................11
Figure 6      CFD results of selected percent pitch configurations at zero axial
               overlap ...........................................................................................................13
Figure 7      Surface isentropic Mach number at 0 AO, 90 PP ...........................................16
Figure 8      Mach number contours at 0 AO, 90 PP ............................................................16
Figure 9      Surface isentropic Mach number at 0 AO, 50 PP ...........................................17
Figure 10     Mach number contours at 0 AO, 50 PP ............................................................17
Figure 11     Surface isentropic Mach number at 0 AO, 5 PP ..............................................18
Figure 12     Mach number contours at 0 AO, 5 PP ............................................................18
Figure 13     CFD results of selected axial overlap configurations at 90 PP .......................20
Figure 14     Surface isentropic Mach number at 10 AO, 90 PP ...........................................20
Figure 15     Example loss bucket with incidence range .....................................................21
Figure 16     Incidence Range vs. Min. Loss D-Factor ............................................................22
Figure 17     Selected Single Blade Compressor and Tandem Rotor Design Points ............28
Figure 18     Tandem and Single Rotor 2-D Profiles .............................................................30
Figure 19     Forward and aft blade pressure surface meshes ..............................................31
Figure 20     Current tandem rotor inlet total pressure profile ...........................................32
Figure 21     Tandem and single rotor 100% speed performance characteristics .................35
Figure 22     Conventional rotor 3-D fluid mechanics .............................................................36
Figure 23     Forward blade incidence for tandem rotor and 2-D goal ..................................38
Figure 24     Forward blade deviation for tandem rotor and 2-D goal ..................................39
Figure 25     Forward blade D-Factor for tandem rotor and 2-D goal ..................................39
Figure 26     Forward blade losses for tandem rotor and 2-D goal ......................................40
Figure 27     Overall D-Factor for tandem rotor and 2-D goal ..............................................40
Figure 28     Aft blade incidence for tandem rotor and 2-D goal ..........................................42
Figure 29     Aft blade deviation for tandem rotor and 2-D goal ..........................................42
Figure 30     Aft blade D-Factor for tandem rotor and 2-D goal ..........................................43
List of Tables

Table 1  Current tandem rotor geometric parameters ................................................30
Table C-1  Simple Design Rule Inputs and Outputs .......................................................59
Table C-2  Single airfoil geometric parameters ..............................................................61
Table C-3  Tandem airfoil geometric parameters ...........................................................62
Table E-1  Tandem airfoil geometric parameters for loading split study .........................73
Nomenclature

AA  aft airfoil (used in 2-D cascade frame of reference)
AB  aft blade (used in 3-D rotating frame of reference)
AO  axial overlap of tandem airfoils (blades)
AVDR axial-velocity density ratio
C  chord
C_{\text{pedal}} Ideal pressure rise coefficient
D  Lieblein diffusion factor
FA  forward airfoil (used in 2-D cascade frame of reference)
FB  forward blade (used in 3-D rotating frame of reference)
h  enthalpy
LE  leading edge
M  Mach number
\dot{m}  corrected mass flow
P  pressure (static unless denoted otherwise)
PP  percent pitch of aft airfoil (or blade) leading edge relative to spacing
PR  pressure ratio, \( P_{0,2} / P_{0,1} \) (absolute frame of reference)
PS  pressure side
r  radial coordinate
s  pitchwise spacing between blade rows
SM  surge margin
SS  suction side
t  pitchwise spacing between forward airfoil (or blade) trailing edge and aft airfoil (or blade) leading edge
T  temperature
TE  trailing edge
U  blade speed
V  air velocity in absolute frame of reference (used in 3-D)
w  air velocity in cascade frame of reference (used in 2-D)
y^+  dimensionless distance from viscous surface
z  axial coordinate
Greek

$\alpha^*$ minimum-loss incidence angle

$\beta$ flow angle relative to axial coordinate

$\gamma$ ratio of specific heats

$\Delta x_1$ axial distance between forward airfoil (blade) trailing edge and aft airfoil (blade) leading edge

$\Delta x_2$ axial distance between aft airfoil (blade) leading edge and forward airfoil leading edge

$\zeta$ hub-to-tip radius ratio

$\theta$ pitchwise coordinate

$\theta^*$ boundary layer momentum thickness at trailing edge

$\kappa$ airfoil (blade) metal angle relative to axial coordinate

$\rho$ density

$\sigma$ solidity, $C / s$

$\Phi$ flow coefficient

$\varphi$ camber

$\psi$ work coefficient

$\omega_C$ stagnation pressure loss coefficient

$\omega_P$ momentum thickness loss parameter

Subscript

eff effective

ov overall

0 stagnation conditions

1 single airfoil (blade) inlet station

2 single airfoil (blade) exit station

11 forward airfoil (blade) inlet station

12 forward airfoil (blade) exit station

21 aft airfoil (blade) inlet station

22 aft airfoil (blade) exit station
1. Introduction

This dissertation is in a non-traditional format: it is based largely upon two high-level peer-reviewed papers, followed by several appendices that provide detailed background information.

Chapter 1 describes the background and overall goals of the project.

Chapter 2 is based on the first peer-reviewed paper [1]. It was presented at the 2007 International Mechanical Engineering Conference and Exposition in Seattle, WA, and has been accepted for publication in the *ASME Journal of Turbomachinery*. Chapter 2 reviews previous works on 2-D tandem airfoils in the cascade frame of reference, then covers the current 2-D numerical work.

Chapter 3 is based on the second paper [2], which has been submitted for publication at the 2008 ASME Turbo Expo in Berlin, Germany. That paper will likely also be submitted to the *Journal of Turbomachinery*. Chapter 3 reviews previous works on 3-D tandem blade rotors, then covers the current 3-D numerical work.

Chapter 4 contains the concluding remarks.

1.1 Background

Jet engine manufacturers are continuously seeking to produce engines that are lighter and smaller in order to reduce manufacturing and operating costs. Part of the approach is to design a compressor with fewer stages while maintaining the required overall pressure rise and efficiency. This necessitates more work, i.e. pressure rise, per stage.

A major limitation on the pressure rise in a subsonic axial-flow compressor stage is boundary layer separation on the blade suction surface and endwalls, i.e. hub and casing. One method of mitigating the suction surface separation is to employ tandem-airfoil blades. The basic concept is that a new boundary layer forms on the second (aft) airfoil, allowing for high overall loading without the large flow separations that would be seen with a single airfoil (Figure 1). The reader should distinguish between tandem blades (small overlap) and splitter blades (high overlap) such as are employed in centrifugal compressors.
Tandem airfoils are used as flaps and slats on aircraft to improve lift during takeoff and landing. In commercial turbomachinery, tandem blades have been employed as stators, examples of which include the GE J-79 compressor [3] and an advanced single-stage LP compressor built by Honeywell [4]. However, to the best of available knowledge they have not been used in commercial rotors. This lack of commercial use is the motivation for the current study.

Chapter 3 reviews experimental tandem rotors that have been constructed and tested. While the particular machines varied in size and scope, they all shared the shortcoming of narrow stability range from design conditions, which may explain why tandem blades have not yet been employed in commercial rotors.

The goal of this project is to evaluate the feasibility of a tandem blade rotor as applied to the rear stages of a core compressor. That section of the compressor is characterized by high hub-to-tip ratios and fully subsonic, shock-free flow at design conditions, a combination that has not previously been examined. If successful, the tandem rotor could be used when high loading is required while maintaining an acceptable overall efficiency and stability margin. It must be noted here that unlike previously tested tandem rotors, this study limits inlet Mach numbers to 0.6 to avoid the complexities of passage shocks.

Figure 1: 2-D Profile view of highly loaded axial-flow single (L) and tandem airfoil (R)
1.2 Specific Objectives of Current Work

There were three main tasks to evaluate the feasibility of using tandem blade rotors in the back of a core compressor, all of which were computational in nature: a 2-D study in the cascade frame of reference, selection of the “best-case” geometry from the 2-D study for further analysis in the 3-D rotating frame of reference, and the 3-D study.

1.2.1 2-D Cascade Study

Chapter 2 is dedicated to the 2-D analysis, the purpose of which was threefold. The first goal was to examine on-design performance trends over a wide range of overall loadings for a single airfoil family. The second goal was to examine off-design performance, particularly incidence range as a function of on-design loading. Finally, the CFD solutions were interrogated to evaluate the governing fluid mechanics.

1.2.2 Selection of Geometry for 3-D Study

Chapter 3.2.2 describes the “best-case” geometry from the 2-D study that was selected to be the basis for a fully 3-D tandem rotor. Desirable 2-D on-design performance was to achieve high loading without prohibitively high losses, leading to a tandem rotor that would produce a higher pressure rise than a conventional single-blade rotor without compromising efficiency. Desirable 2-D off-design performance was to have sufficient incidence range to produce a 3-D tandem rotor would have a surge margin comparable to a conventional rotor.

1.2.3 3-D Rotor Study

The remainder of Chapter 3 is dedicated to the tandem rotor simulated in 3-D viscous flow to examine its overall performance (pressure rise and efficiency) versus a comparable single-blade rotor. More importantly, the CFD solutions were thoroughly interrogated to provide some insight as to the complex 3-D fluid mechanisms of a tandem rotor. This information can be used to optimize a future rotor design for use in an engine.
Chapter 2  2-D Tandem Airfoils

This chapter is taken from Reference 1. There are some differences for clarity and flow. Section 2.1 gives the literature review on 2-D tandem airfoils in the cascade frame of reference. Section 2.2 describes the 2-D CFD study. Section 2.3 gives a summary of the 2-D work.

2.1 2-D Cascade Literature Review

The 2-D tandem airfoil geometric parameters are shown in Figure 2. Included are the common single-airfoil parameters of airfoil family, camber, chord, and solidity. Axial overlap and percent pitch are specific to tandem-airfoils. The effective chord, spacing, and solidity of the tandem airfoil is defined in terms of axial overlap assuming that the spacings are equal for both the forward and aft airfoils.

1.) Airfoil family, e.g. NACA, CDA
2.) Cambers:
   a. forward airfoil: \( \varphi_{FA} = \kappa_{11} - \kappa_{12} \)
   b. aft airfoil: \( \varphi_{AA} = \kappa_{21} - \kappa_{22} \)
   c. Overall: \( \varphi_{ov} = \kappa_{11} - \kappa_{22} \)
3.) Effective chord and solidity:
   a. chord: \( C_{eff} = (C_{FA} + C_{AA}) / (1+AO) \)
   b. spacing: \( s_{eff} = (1 - 0.5 \times AO) \times s \)
   c. solidity: \( \sigma_{eff} = C_{eff} / s_{eff} \)
4.) Axial Overlap: \( AO = \Delta x_1 / \Delta x_2 \)
5.) Percent Pitch: \( PP = t / s \)

Figure 2: Tandem airfoil 2-D geometric parameters
A survey of open literature on 2-D tandem airfoils reveals a wide range of methods and flow conditions that have been examined in rectilinear cascade configuration [5-29]. Experimental studies of various airfoil families and geometric configurations have been conducted for inlet Mach numbers ranging from 0.05 to 1.6. Some employed endwall and sidewall boundary layer removal, while others did not. Computational studies have ranged from potential flow with boundary layer codes to fully viscous Reynolds-Averaged Navier-Stokes (RANS) solvers. More detailed descriptions are given in Appendix A.

Because the current interest is in core compressor stages, the most pertinent works are those that meet the following criteria:

- Subcritical inlet flow (no shocks are present)
- Boundary layer removal employed for experimental data in order to achieve AVDR ~1.0 (AVDR and its importance in 2-D cascade flow are explained in Appendix B)

Of all the experimental works on 2-D tandem airfoils, only Railly & El-Sarha [9], Wu et al. [19], and Roy & Saha [22-24] meet the above criteria. The two computational works that are of interest are Sanger [17] and Canon-Falla [28]. Several key points about subsonic 2-D tandem airfoils emerge from these works:

- Axial overlap and percent pitch have a first-order effect on performance
- The best performance is achieved with high percent pitch and low axial overlap, similar to that shown in Figure 2
- At high loading an optimized tandem airfoil will produce less loss than a comparable single airfoil
- Airfoil camber ratio ($\varphi_{FA} / \varphi_{AA}$), which is proportional to the loading split, also affects tandem performance
- The interaction of the flow fields between the forward and aft airfoils is important

From a compressor designer’s standpoint, it would be useful to see the pertinent literature data in a graphical format such as a loss vs. loading correlation. To accomplish
this, the Lieblein diffusion factor (henceforth referred to as the D-Factor) is employed as a measure of airfoil loading, defined for a tandem airfoil by Equation 1.

\[ D \equiv \left(1 - \frac{w_{22}}{w_{11}}\right) + \left(\frac{W_{\theta,11} - W_{\theta,22}}{2\sigma_{eff}w_{11}}\right) \]  

**Equation (1)**

The first and second terms of Equation 1 respectively represent velocity diffusion and turning within the flow passage. The associated loss parameter is a representation of the boundary layer momentum thickness at the trailing edge normalized by the airfoil chord length, defined for a tandem airfoil by Equation 2.

\[ \omega_p \equiv \left(\frac{\theta^*}{C_{eff}}\right) \approx \omega_c \cos \beta_{22} \left(\frac{\cos \beta_{22}}{\cos \beta_{11}}\right)^2 \]  

**Equation (2)**

The symbol \( \omega_c \) in Equation 2 is the loss coefficient. It represents the loss in stagnation pressure across the tandem airfoil, defined by Equation 3. Unless otherwise stated, all loss coefficients reported in the current work are mass-averaged. Precise definitions of mass-averaged and area-averaged losses are in Appendix H.

\[ \omega_c \equiv \frac{P_{0,11} - P_{0,22}}{P_{0,11} - P_{11}} \]  

**Equation (3)**

These parameters allow for a wide range of cambers and solidities to collapse onto a single performance curve for a given airfoil family, as Lieblein [30] demonstrated with low-speed NACA single airfoil cascade data. The loss parameter vs. D-Factor correlation has historically proven to be an excellent indicator of 2-D performance even for flows that are radically different from Lieblein’s original assumptions [31]. Thus, it is quite reasonable to adapt the method for tandem airfoils.

Available data from References 9, 17, 19, 22, 23, 24, and 28 have been expressed in terms of D-Factor and loss parameter, as shown in Figure 3. The general trend is outlined by the dashed shape, and is similar to a typical single airfoil [30]. Scatter in the data can be partially attributed to the variety of airfoil families and airfoil shapes.
examined. However, it is noteworthy that scatter exists even among the data of researchers who used only one set of tandem airfoils [9, 28]. For example, Canon-Falla changed only axial overlap and percent pitch, yet there is noticeable variation in the loss levels of his tandem configurations. This serves as a visual reinforcement of the idea that percent pitch and axial overlap have first-order effects on overall performance.

Figure 3: Lieblein chart of pertinent literature data

Several of the points on Figure 3 lie above the general performance trend. The common characteristic among these outlying points is that each configuration had a camber ratio greater than unity, suggesting that the forward airfoil was more loaded than the aft airfoil. Ihlenfeld [7] recommended that the airfoils be equally loaded for the best loading-to-loss ratio.

Figure 3 serves to show the trend of tandem performance at minimum loss conditions (hereafter referred to as design conditions for convenience). Unfortunately, there is only one data point for each airfoil family / axial overlap / percent pitch combination. From the designer’s standpoint, it would be useful to have complete curves
of performance sensitivity to these parameters over a wide range of D-Factors, i.e. $0.4 < D < 0.7$. Given the size of the test matrix for an even modest sweep of axial overlap and percent pitch, gathering this data experimentally would be an expensive and time-consuming task. Therefore, a 2-D computational approach seems more appropriate, the results of which are presented in Section 2.2.

2.2 2-D CFD Study

The purpose of the CFD analysis was threefold. The first goal was to examine on-design performance sensitivity to the tandem airfoil parameters of axial overlap and percent pitch over a wide range of D-Factors for a single airfoil family.

The second goal was to examine off-design performance. By definition, the D-Factor is calculated at the minimum loss incidence angle, meaning that it is necessary to generate a loss coefficient vs. incidence plot (See Figure 15) for each tandem configuration. Since this data is requisite for a Lieblein chart, it was also possible to determine the incidence range for each tandem airfoil, giving an indication of off-design performance. Incidence range is defined here as the difference between the two flow angles that produce a loss coefficient twice that of the minimum value.

Finally, the CFD solutions were interrogated to evaluate the governing fluid mechanics behind both the on- and off-design performance trends.

2.2.1 Simple Design Rule

In order to perform the tandem airfoil CFD study in an efficient manner, a simple “back-of-envelope” design rule was developed. The purpose of the design rule was to estimate the required camber and stagger angles such that both the forward and aft airfoils were operating at minimum loss conditions.

The underlying assumption of the design rule is that the forward and aft airfoils act independently of one another. One can then specify the individual airfoil D-Factors, and subsequently calculate the overall D-Factor for the airfoils in tandem. If a loss vs. loading correlation is available for the chosen airfoil family, the individual airfoil losses can be determined based upon the specified D-Factors. Individual airfoil losses are then superposed to give the estimated overall loss for the tandem airfoil.
While certainly not the most potent of airfoils, there is a large amount of openly available data on the NACA-65 family [30], making it a convenient choice for use in this simple design rule. The NACA-65 data are presented in a way that allows one to select the required airfoil metal angles (κ₁₁, κ₁₂, κ₂₁, κ₂₂) for minimum loss operation at a desired D-Factor. The procedure is conceptually similar to the one used by Canon-Falla [28], who set his airfoil metal angles based upon NACA-65 Mellor diagrams. For a more detailed explanation of the simple design rule, see Appendix C.

As a final preparation, a series of CFD cases were run on various NACA-65 single airfoil geometries (listed in Appendix C) at an inlet Mach number of 0.6, consistent with the requirement that no shocks be present in the flow field. This gave a basis for direct comparison between the tandem airfoil and single airfoil. Also, the loss vs. loading curve for the single airfoil was used to estimate overall losses in the tandem design rule. Figure 4 shows a third order polynomial curve fit of the CFD-generated single airfoils and the design rule estimate for a tandem airfoil. The single airfoil curve follows the trend of NACA-65 experimental data taken by Lieblein [30]. Comparing the single airfoil to the tandem design rule, there is approximately a 20% drop in loss levels at an overall D-Factor of 0.62.

It should again be emphasized that the design rule assumes no interaction between the flow fields of the forward and aft airfoils. Other researchers—Bammert & Staude [43], for example—have concluded that the interaction effects of a properly designed tandem configuration can result in overall losses that are less than the sum of the individual airfoil losses. It can therefore be inferred that the simple design rule gives a conservative estimate of the loss reductions that can be achieved at high loading.

The design rule was also used in the tandem airfoil CFD study to set the airfoil metal angles for minimum loss operation. This is described the next section.
2.2.2 Tandem Airfoil CFD Setup

The aim was to generate data points ranging from an overall D-Factor of 0.4 to 0.7. Once an overall D-Factor was specified, the simple design rule was used to determine the individual airfoil D-Factors and required metal angles for minimum loss operation. Because the design rule relied upon NACA-65 data, those were the airfoil profiles used in the CFD study. As previously mentioned, the open literature (outlying data points on Figure 3 and Reference 7) suggest that losses tend to be greater when the airfoils are unequally loaded. Consequently, in order to minimize losses, the forward and aft airfoils were designed to have equal individual D-Factors. Details of the chosen geometries are given in Appendix C.

Once metal angles had been determined, the tandem airfoil flow fields were simulated with axial overlaps ranging from -10 to 30, and percent pitch values from 5 to 95. Negative axial overlap indicates a separation of the forward and aft airfoils. Loss
buckets for each axial overlap / percent pitch combination were all simulated at an inlet Mach number of 0.6.

Computational meshes were created to be radially thin slices of a blade profile at mid-span in order to simulate 2-D cascade flow (AVDR = 1.0). The forward and aft airfoil meshes were generated separately, and then combined into a single tandem airfoil mesh consisting of four blocks: Inlet, Lower Passage, Upper Passage, and Exit, and an example of which is shown in Figure 5. Tandem meshes were as large as 181,000 points.

Boundary conditions included inviscid endwalls in order to create a 2-D streamtube, and the viscous airfoil surfaces had no rotation since the goal was to model cascade flow. Both the inlet flow angles and exit back pressures were varied to ensure a constant inlet Mach number at each point on the loss buckets.

![Figure 5: Example multi-block H-mesh used for tandem airfoil simulations (2-D view)](image)

The CFD solver employed is called Advanced Ducted Propfan Analysis Code (ADPAC), a RANS code that was developed specifically to analyze ducted turbofan
engines. Time-marching was carried out by the explicit, 4-stage Runge-Kutta scheme. ADPAC has three available turbulence models: Baldwin-Lomax algebraic, Spalart-Allmaras one-equation, and Goldberg two-equation. A detailed description of how each are implemented in the code is given in Reference 32. In order to strike a reasonable balance between solution convergence time and accurate modeling of separated flows, all 2-D results were obtained using the Spalart-Allmaras model with fully turbulent freestream flow. The maximum value of $y^+$ was generally kept below 5.

Additional details of the computational methods, including a discussion of grid independency and convergence criteria, are given in Appendix D.

2.2.3 Results: On-Design (Minimum-Loss)

The caveat to the CFD data presented in this section is that it is based upon a decades-old airfoil family. This is reasonable, since the primary interest is sensitivity trends rather than absolute performance levels. It will be shown that the “sweet spot” of axial overlap and percent pitch is in agreement with previous works. Therefore, it is likely that the same general trends would continue to hold for a more modern airfoil family. The difference being that the absolute loss levels would likely be lower for a given overall D-Factor.

The general effect of varying axial overlap on tandem performance was similar for all values of percent pitch examined. Likewise, the effects of varying percent pitch were similar for all values of axial overlap. It is not fair to say that axial overlap and percent pitch are completely independent. However, examining each parameter separately provided insight into the basic flow mechanisms of the tandem airfoil. As such, the results are discussed in two sections.

2.2.3.1 Percent Pitch Variation

Figure 6 shows performance curves for several selected values of percent pitch at zero overlap, as well as the range of literature data, indicated by the dashed shape from Figure 3. As will be discussed in later in this subsection, zero axial overlap results in the best performance at a given percent pitch.
From Figure 6 it is evident that best performance is achieved at the high percent pitch (85, 90, and 95) configurations, which is consistent with findings in the literature. This gives some validation to the methodology, and boosts confidence in using the software tools for more advanced tandem blade analysis.

Also shown on Figure 6 is the CFD curve for a single NACA-65 airfoil that was used in the simple design rule (solid line in Figure 4). It can be seen that when airfoil interaction effects are included, there is as much as a 40% reduction in loss levels from a single airfoil to a comparable tandem airfoil at an overall D-Factor of 0.62. However, while a properly designed tandem is superior to a comparable single airfoil, the benefits can be quickly lost if a designer deviates too far from the optimum percent pitch, as evidenced by the curve of 50 PP. At first it would seem that the 0 AO, 50 PP configuration would match the simple tandem design rule. However, it will be seen from Figures 9 & 10 that at 50 PP the aft airfoil is operating several degrees away from minimum loss incidence, resulting in a higher overall loss parameter. Losses on low percent pitch configurations (5 & 15) grew so rapidly with overall D-Factor that converged solutions were unobtainable above $D_{ov} \sim 0.55$.

![Figure 6: CFD results of selected percent pitch configurations at zero axial overlap](image-url)
It is noteworthy that the trend of sensitivity to percent pitch is consistent throughout the range of D-Factors examined. While previous investigations certainly hinted at this, here it is presented for the first time in a comprehensive plot that could easily be incorporated into a design system.

As the simple design rule demonstrated in Figure 4, the tandem airfoil can be superior to a comparable single airfoil even without considering flow field interaction. When such interactions are taken into account, the tandem airfoil appears even more promising.

Figure 7 is a plot of airfoil surface isentropic Mach number vs. overall axial chord for the 0 AO, 90 PP configuration indicated by the solid arrow on Figure 6. (Surface Mach number is inversely proportional to surface static pressure.) Visualization of the flow field is shown in Figure 8. The solid curves of Figure 7 represent the forward and aft airfoils when simulated in tandem. The airfoils from this particular configuration were also simulated in isolation using the same inlet flow conditions as they experienced when in tandem (i.e. same inlet Mach numbers and incidence angles). This allows one to study the effects of flow field interaction. These isolated cases are shown by the dashed curves in Figure 7.

The most noticeable difference between the tandem and isolated cases in Figure 7 is the loading envelope of the forward airfoil, particularly on the pressure side. This is caused by the stagnation region of the aft airfoil near the forward airfoil trailing edge. In effect, the aft airfoil induces additional circulation on the forward airfoil, resulting in greater forward airfoil loading, i.e. more turning and a higher D-Factor. This can be considered a benefit so long as it does not push the forward airfoil to the point of flow separation.

On the aft airfoil there is a small region of acceleration on the suction side near the leading edge, resulting from the contracting gap between the two airfoils that acts as a nozzle (See Figure 7). This local acceleration leads to a favorable dumping velocity, a term used to describe the velocity of the fluid discharging from the forward airfoil trailing edge onto the aft airfoil leading edge. Without that region of accelerated flow, there could be a local adverse pressure gradient near the aft airfoil leading edge that could unfavorably affect the newly developing boundary layer.
Both the induced circulation and dumping velocity are phenomena commonly observed in external flows such as with a wing and flap. The reader is referred to Smith’s paper [33] on high-lift aerodynamics for a more detailed discussion.

Figure 9 is a surface Mach number plot for the 0 AO, 50 PP data point indicated by the dashed arrow on Figure 6. Figure 10 is a visualization of the flow. The configurations shown in Figures 7 / 8 and 9 / 10 are geometrically identical in every way except for the pitchwise location of the aft airfoil, yet their respective positions on the Lieblein chart differ dramatically. Upon moving the aft airfoil away from the forward airfoil in the pitchwise direction, the induced circulation diminishes resulting in less loading on the forward airfoil and a lower overall D-Factor. The aft airfoil is also subject to a noticeably higher incidence angle, as evidenced by the sharp velocity peak at the suction side leading edge (Figures 9 & 10). The subsequent strong diffusion on the aft airfoil suction side leads to significantly greater losses than are seen at the higher percent pitch configurations. The indication is that the forward airfoil acts to guide the flow onto the aft airfoil, and moving the aft airfoil too far away in the pitchwise direction can result in it operating at off-design. One could arguably place the aft airfoil in mid-passage and orient it such that it operates at minimum loss, but that would sacrifice the induced additional loading on the forward airfoil, reducing overall work capability. Hence the 50 PP curve on Figure 6 would shift down, but not to the right.

At low percent pitch the forward airfoil surface Mach number in tandem is nearly the same as in isolation (Figure 11). However, the aft airfoil leading edge immersed in the forward airfoil wake (Figure 12), resulting in early separation of the flow on the aft airfoil suction side. The aft airfoil was not simulated in isolation at the 5 PP conditions because the exact incidence angle was difficult to determine within the forward airfoil wake.
Figure 7: Surface isentropic Mach number at 0 AO, 90 PP

Figure 8: Mach number contours at 0 AO, 90 PP
Figure 9: Surface isentropic Mach number at 0 AO, 50 PP

Figure 10: Mach number contours at 0 AO, 50 PP
Figure 11: Surface isentropic Mach number at 0 AO, 5 PP

Figure 12: Mach number contours at 0 AO, 5 PP
2.2.3.2 Axial Overlap Variation

Figure 13 is a plot of mass-averaged loss parameter versus D-Factor for axial overlaps of -10, 0, and 10 at 90 percent pitch. The single airfoil curve from Figures 4 and 6 is also shown for reference. As evidenced by Figure 13, holding all other geometric parameters constant, varying the axial overlap from zero tends to diminish performance, particularly at high loading. This trend held true across the range of percent pitch values examined.

To understand the fluid mechanics behind the diminished performance at positive overlap, consider Figure 14, which is a plot of airfoil isentropic surface Mach number for the 10 AO, 90 PP configuration indicated by the dashed arrow on Figure 13. With positive overlap, the gap-nozzle acceleration on the forward airfoil pressure side moves upstream of the trailing edge. This tends to counteract the additional loading from the induced circulation, and can be seen by comparing the forward airfoil pressure side Mach numbers in tandem from Figures 7 & 14. At higher overlaps (i.e. AO > 15) the forward airfoil can experience a substantial reduction in loading, even to the point of contributing no work. This phenomenon holds true regardless of where the aft airfoil is positioned pitchwise, though it is most pronounced at high percent pitch.

With negative overlap, i.e. axial separation, the increasing distance between the forward airfoil trailing edge and the aft airfoil stagnation region reduces the induced circulation. Since both airfoils are designed to operate below the point of separation, reducing the forward airfoil loading will not substantially reduce the overall losses, the result being a mostly leftward shift on the Lieblein chart.

2.2.3.3 Airfoil Loading Split

A separate CFD study was conducted to examine how loading split affects the on-design performance of a 2-D tandem airfoil. It is described in detail in Appendix E. The conclusion was the same as had been suggested by the literature: shifting the loading distribution to either airfoil tends to increase losses for the tandem configuration.
Figure 14: Surface isentropic Mach number at 10 AO, 90 PP

Figure 13: CFD results of selected axial overlap configurations at 90 PP
2.2.4 Results: Off-Design (Incidence Range)

For each point on a Lieblein chart there is an associated loss bucket, an example of which is shown in Figure 15. The minimum loss incidence angle is denoted by $\alpha_*$. This is where the D-Factor is calculated for purposes of the Lieblein chart. Having a complete loss bucket also allows one to determine the incidence range of a given single or tandem airfoil. Here, the incidence range is defined as the difference in flow angles that produce a loss coefficient twice that of the minimum. The airfoils would be considered stalled at this point.

One way to evaluate airfoil off-design performance is to plot the incidence range of each loss bucket against its associated minimum loss D-Factor. Figure 16 is such a plot that includes tandem and single airfoil CFD data points from Figure 6, as well as the band of literature from Figure 6.

![Figure 15: Example loss bucket with incidence range](image-url)
It can easily be seen from Figure 16 that the incidence range of both tandems and single airfoils decreases with increasing minimum loss D-Factor. This is to be expected, since operating range tends to diminish with higher design loading. Secondly, the CFD indicates a clear trend that the tandem airfoil can offer better incidence range over a single airfoil. The reader should be reminded at this point that both the tandem and single airfoil CFD are for geometrically similar airfoils and the same inlet Mach number, allowing for direct comparison. However, the data from the literature were gathered using different airfoil families, and more importantly, at lower Mach numbers (0.3 or less vs. 0.6 for the CFD). Therefore the literature data should be taken only as an indication of trend from experiments, not as a direct comparison to the CFD.

The mechanism behind the superior incidence range of the tandem airfoil can be explained by considering the inlet flow to each airfoil separately. The aft airfoil sees a lower inlet Mach number than the forward airfoil (0.4 vs. 0.6 in the CFD study), so it tends to have a lower and wider loss bucket than the forward airfoil. Consequently, the forward airfoil can operate several degrees away from minimum loss incidence before the
aft airfoil experiences a substantial increase in loss. It must be emphasized that the CFD data in Figure 16 includes a wide range of percent pitch configurations (5 to 95) at 0 AO. It is rather interesting to note that the aft airfoil pitchwise position has little effect on incidence range, despite having a first-order effect on minimum loss performance. The dominating influence of the forward airfoil on overall tandem incidence range may explain this insensitivity.

2.3 General Summary of 2-D Tandem Study

Pertinent available 2-D data on tandem airfoils from the open literature have been compiled and presented in a format that is particularly useful for compressor designers. A “back-of-envelope” simple design rule has demonstrated that a tandem configuration can be highly loaded (i.e. $D_{ov} > 0.60$) and produce lower loss levels than a comparable single airfoil. A 2-D CFD study of the tandem airfoil was conducted to examine on- and off-design performance sensitivity to axial overlap and percent pitch. The major findings are summarized as follows:

- Tandem on-design performance is highly sensitive to axial overlap and percent pitch, and the trend is shown to be consistent throughout a wide range of overall D-Factors.
- The optimum configuration (high percent pitch, low overlap) is consistent with existing literature, giving some validation to the methods and tools used in this study.
- Tandem airfoil off-design performance is superior to that of a comparable single airfoil of the same family.

Given the results shown here, the tandem airfoil as applied to a core compressor was deemed worthy of further investigation. Chapter 3 discusses 3-D simulations of a tandem blade rotor using the same computational tools as in the 2-D study.
3. 3-D Tandem Rotors

This chapter is taken from Reference 2. There are some differences for clarity and flow. Section 3.1 gives the literature review on 3-D tandem blade rotors. Section 3.2 describes the 3-D CFD study. Section 3.3 gives a summary of the 3-D work.

It should be emphasized that this is a proof-of-concept study, constructed to simulate and understand the differences between 2-D and 3-D fluid mechanisms of tandem airfoils under the same modeling conditions. Therefore, the tandem rotor presented here represents a simple “first pass” design. Future work may involve optimizing the blade profiles and three-dimensional stack in addition to addressing mechanical constraints.

3.1 3-D Rotor Literature Review

Before reviewing the literature, the reader should be aware of several dimensionless parameters that are commonly employed to describe compressor performance, and they are used frequently throughout this work. The first is work coefficient (Equation 4), which is the stagnation enthalpy rise across a single stage normalized by the square of the blade tip speed. The second is flow coefficient (Equation 5), which is the mean inlet axial velocity in the absolute frame of reference normalized by blade tip speed. The third performance parameter is surge margin (Equation 6), which represents the range of stable compressor operation from the design point as the mass flow decreases at a constant rotational speed. Conversely, choke margin is the range of increased mass flow at a constant rotational speed before the flow passage becomes choked. A more detailed explanation of surge and choke in a compressor can be found in practically any textbook on compressor aerodynamics, Cumpsty’s [34] being a good example. Equation 6 is taken from Reference 34.

\[ \psi = \frac{\Delta h_{t}}{U_{tip}^2} \]  
Equation (4)

\[ \Phi = \frac{V_z}{U} \]  
Equation (5)
3-D experimental tandem-type rotors have been in existence since the Second World War. The earliest known example is a slotted-blade low-speed blower tested at the Stuttgart Institute [35]. The reader should distinguish between slotted blades and tandem blades. Slotted blades are created by cutting a gap in a long-chord single airfoil, whereas tandem blades are two distinct airfoils. The Stuttgart data indicated that the slotted-blade blower produced a higher pressure rise but lower efficiency than a comparable single blade blower. Sheets [36] later demonstrated that a well-designed slotted blade blower can achieve both high pressure rise and high efficiency.

Linemann [37] conducted a comprehensive series of low-speed tests using a blower with both a tandem rotor and tandem stator. He methodically varied the relative positions of the tandem blades in order to determine the optimum configuration. He concluded that zero axial overlap and 80 percent pitch produced the greatest pressure rise and efficiency for both the rotor and stator. He also observed that centrifugal effects of the boundary layer had no noticeable effect on the optimum overlap or percent pitch.

Railly & Mehra [38] tested a low-speed tandem blade blower with fixed blade orientations. Their traverse data indicated that the highest losses were near the hub, but they were unable to provide details of the 3-D fluid mechanisms.

Monsarrat et al. [39] describe a design for a highly loaded single-stage compressor with an inlet hub-to-tip ratio of 0.50 and a tip speed of 1,600 ft/s, resulting in an inlet tip relative Mach number of 1.6. Target rotor performance was a work coefficient of 0.30 at a flow coefficient of 0.34 with a polytropic efficiency of 89.7%. They proposed using either a slotted or tandem rotor to achieve this goal. However, there does not appear to be any evidence that this design was ever constructed or tested.

Stricker [40] developed a tandem blade rotor to increase thrust and reduce cavitation damage in bidirectional flow devices, i.e. a pump or fan with variable camber.
One application of a bidirectional pump is to provide a naval vessel with turning capability independent of the main steering systems.

Hashimoto [41] describes a tandem rotor for a fan on a high-bypass ratio engine. He started with the concept of an engine where two of the power turbine rotor blades extent outside the core engine and serve as contra-rotating propeller blades. Arguing that these propeller blades would suffer from excessive mechanical stress, he proposed replacing them with smaller tandem rotor blades that are also integral to the power turbine.

Of greatest relevance to the current study are the tandem rotor compressors of References 42 – 46 that underwent the full cycle of design, construction, and testing. These are described in detail below. The performance characteristics of pressure rise, efficiency, and stability margin are expressed using Equations 4, 5, and 6, respectively. The surge margin is computed using the left-most point on each of the respective references’ 100% design speedlines.

Brent & Clemmons [42] performed experiments on a single-blade rotor and two different tandem rotors. The mean hub-to-tip ratio of all three rotors was 0.80. Each rotor’s inlet tip speed was 757 ft/s, resulting in an inlet tip relative Mach number of 0.80. The two tandem rotors differed in loading split between the forward and aft blades: one had a 50 / 50 split and the other had a 20 / 80 split. The 50 / 50 split rotor in isolation achieved a work coefficient of 0.48 at a mean flow coefficient of 0.59. The work coefficient was only 4.5% below the design value, and the 50 / 50 rotor met the design polytropic efficiency of 91.2%. By comparison, both the 20 / 80 tandem and the baseline single-blade rotor fell considerably short of the design work coefficient and efficiency. Both of the tandem rotors had a surge margin of 17%, which was approximately the same as the baseline single rotor.

Bammert et al. [43 – 45] tested a multistage compressor that consisted of a single-blade rotor, three high-reaction tandem rotors, and another single-blade rotor (front to back). All stators were single-blade impulse type. The average hub-to-tip ratio throughout the compressor was 0.64. Average blade tip speed on-design was 620 ft/s, resulting in an inlet relative tip Mach number of 0.85. There was insufficient information available in the literature to calculate stage-by-stage performance values. However, the overall
pressure ratio and adiabatic efficiency can be used to estimate a whole-machine work coefficient of 0.77 and a polytropic efficiency of 85.6%. The flow coefficient based upon compressor inlet conditions and geometry was 0.51. The surge margin from nominal was less than 5%. It is also noteworthy that the compressor reached peak efficiency at 70% of design speed. This was attributed to “the decreasing influence of the Mach number.” Traverse data between the stages indicated that the highest losses were near the hub.

Hasegawa et al. [46] tested a single-stage transonic compressor with a tandem blade rotor and single blade stator followed by an outlet guide vane (OGV), making the stator effectively a tandem blade. Design inlet tip speed was 1,413 ft/s, resulting in an inlet relative tip Mach number of 1.40. The whole stage achieved a work coefficient of 0.51 with a polytropic efficiency of 79.4%. Insufficient information was provided to determine the flow coefficient. Surge margin was approximately 10% while there was very little choke margin. Traverse data indicated losses were higher at the tip than at the hub. It is worth noting that while the tandem rotor had been analyzed with a fully 3-D viscous flow solver, the effects of tip clearance leakage flows were not considered.

While the particular machines of References 42 – 46 varied in size and scope, they all shared the shortcoming of narrow stability range from design conditions, which may explain why tandem blades have not yet been employed in commercial rotors.

As a final note, there have been investigations of other dual-airfoil rotors reported in the literature. While they are not pertinent to the current study, they do bear mentioning as a matter of academic interest. They are reviewed separately in Appendix F.

### 3.2 3-D CFD Study

As already mentioned, the purpose of this chapter is to extend the earlier 2-D cascade tandem airfoil work into the 3-D rotating frame of reference. One of the objectives of the 3-D CFD study is to evaluate the overall performance of a first-pass tandem rotor design and compare it to a geometrically similar single-blade rotor.

The second objective is to evaluate the 3-D fluid mechanisms present in a tandem rotor. Pertinent experimental research on tandem rotors [42 - 46] has provided limited insight into the 3-D flows, mainly via downstream traverse data. CFD simulations allow for more detailed analysis of the flow field than has been previously published.
Stators are not considered in this study. However, given the high level of swirl that could be present at the exit of a tandem rotor, a well-designed stator will be essential to the performance of a whole stage. The knowledge gained from the tandem rotor flow field should prove useful in the future task of designing a robust stator for use in a tandem rotor / single stator stage.

3.2.1 Tandem Rotor Constraints and Design Goal

The mean-line design goal for the tandem rotor was chosen to match the profile losses of a single airfoil near $D = 0.50$. This corresponded to a D-Factor of ~0.62 of the best tandem airfoil in the 2-D study (described in Section 3.2.2). Based upon the flow angles from the 2-D study, the isolated tandem rotor geometry was targeted to produce a work coefficient, $\psi$ of 0.56 at a flow coefficient, $\Phi$ of 0.51. This goal represents the potential duty of a 2-D tandem blade geometry assuming no 3-D flow, shown on the Smith chart in Figure 17 along with values from several conventional rear stage configurations from open literature [47 – 56]. As can be seen, the tandem rotor target is above published levels by over 50%. However, the reader should bear in mind that only a rotor is being considered here, whereas the literature data are for complete stages.

![Figure 17: Selected Single Blade Compressor and Tandem Rotor Design Points](image)
To avoid the complexities of passage shocks the 2-D study was limited to subsonic flow, typical in the rear stages of a core compressor. This design constraint was maintained in the 3-D analysis. The design tip rotational speed was 698 ft/s, resulting in an inlet relative tip Mach number of 0.62 with a relative inlet flow angle, $\beta_{11}$, of 62.0. The 50 / 50 loading split was also maintained for the 3-D rotor design.

### 3.2.2 Tandem Rotor Geometry

Prior to setting the 3-D tandem rotor (hereafter referred to simply as “tandem rotor” for convenience), it was necessary to determine the “best-case” tandem airfoil configuration from the 2-D study. Considering Figure 6 again, it would seem that a 0 AO, 95 PP configuration would be the best choice. Most conventional (single-airfoil) rear stages are limited to $D < 0.5$ because of the rapid increase in loss at higher loading. The 0 AO, 95 PP tandem airfoil attains $D = 0.62$ before reaching the loss level of its single airfoil counterpart at $D = 0.50$. However, at 95 PP the forward and aft airfoils would be in very close proximity to one another, opening up the possibility of problems in manufacturing and / or aeromechanics. These are beyond the scope of this study.

A second consideration is the additional loss incurred by the mixing of the tandem airfoil wakes into the freestream flow. Since this study is concerned with aerodynamics, a FORTRAN program was developed to calculate these mixed-out losses from the 2-D CFD solutions. Details of the mixing loss analysis is given in Appendix G. Suffice to say here that based upon the mixed out losses and possible mechanical / manufacturing difficulties, it was decided that the tandem rotor blades should be at 85 percent pitch instead of 95.

The simple design rule was used to develop a 2-D tandem geometry meeting the design goals and constraints listed above. That geometry was simply extruded to a 3-D blade intended for use in the high reaction rear stage of a core compressor. The blade was not altered to accept incoming flow at the endwalls, having constant geometric parameters across the span. In addition, a high hub-to-tip ratio ($\zeta > 0.95$) and low aspect ratio was imposed. See Table 1 for the detailed geometric parameters.
3.2.3 Single Blade Rotor Geometry

A single blade rotor was generated as a baseline case to provide proof that the design point chosen for the tandem blade is well out of the normal design space for single airfoils. This single blade was the same airfoil family and had the same overall camber, effective chord and solidity, thickness (as a percentage of blade chord), and thickness distribution as the blades for the tandem rotor. Their respective 2-D profiles are shown in Figure 18. The single blade rotor was subject to the same boundary conditions and near wall meshing constraints as the tandem blade rotor.

Table 1: Current tandem rotor geometric parameters

<table>
<thead>
<tr>
<th>Airfoil Family</th>
<th>FB</th>
<th>AB</th>
<th>eff</th>
<th>FB</th>
<th>AB</th>
<th>Overall</th>
<th>$\sigma_{eff}$</th>
<th>AO</th>
<th>PP</th>
<th>% span</th>
<th>% $C_{eff}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>NACA-65</td>
<td>0.6675</td>
<td>0.6675</td>
<td>1.335</td>
<td>20.1</td>
<td>39.3</td>
<td>48.0</td>
<td>1.93</td>
<td>0.0</td>
<td>85.0</td>
<td>1.1</td>
<td>0.5</td>
</tr>
</tbody>
</table>

Figure 18: Tandem (L) and Single Rotor (R) 2-D Profiles

3.2.4 CFD Setup and Procedure (3-D Specific)

The general procedure for the 3-D CFD simulations is the same as was outlined in Section 2.2. There are two additional boundary conditions in 3-D that are not present in 2-D. The first is that of the endwalls, i.e. the hub and casing, which are represented by the minimum and maximum $j$-indices of the mesh. These endwalls are represented as solid, viscous surfaces with no-slip imposed.
The second 3-D-specific boundary condition is the leakage flow through the tip clearance region of the rotors, which was modeled using a periodic boundary condition between the respective blade pressure and suction sides of the gap. This is illustrated in Figure 19, which is a view from the flow passage of the forward and aft blade pressure surfaces. The black and blue mesh lines represent the solid, viscous surfaces of the forward and aft blades, respectively. The red lines at the top of the meshes represent the four cells used to model the tip leakage flow.

An alternative method of modeling the tip leakage flow would be to grid the clearance region itself rather than use periodic boundary conditions. The drawback to that approach is increased complexity and computational time (bearing in mind that the 3-D meshes already contain ~1.1 million grid points). Furthermore, Chima [57] performed a series of 3-D CFD cases demonstrating that a simple tip clearance model gave good agreement with the more complex model. Thus the simple periodic model is deemed sufficient for current purposes.

![Figure 19: Forward and aft blade pressure surface meshes](image-url)
Blade rotation was simulated by specifying a particular rotational speed for the viscous blade surface boundary conditions; in 2-D cascade they had been assigned a rotational speed of zero. Only 100% of the design rotational speed was considered for both the tandem and single blade rotor. Exit static pressure was varied at constant rotational speed in order to generate a speed-line. The rotors were considered stalled when one of two things occurred: mass flow decreased rapidly or large mass flow oscillations occurred. The Baldwin-Lomax turbulence model with wall functions was used in this study. Solutions typically converged within 500 iterations; 800 iterations was the maximum. Details of the convergence criteria are discussed in Appendix D.

A final consideration was that by the rear stages of a core compressor the hub and casing boundary layers have usually developed to a significant thickness. In order to model this as accurately as possible, a defect in total pressure was imposed at the inlet to the tandem and single blade rotors. This defect extended outwards by 30% blade span from both endwalls, and is shown in Figure 20 as a percentage of the pressure in the core flow region.

![Figure 20: Current tandem rotor inlet total pressure profile](image_url)
3.2.5 Results: Overall Performance

This section reviews overall performance of the tandem rotor compared to a single blade rotor. It also contains an estimate of the stage reduction potential of the tandem rotor in the rear of a core compressor. Section 3.2.6 provides insight into the flow field of a tandem rotor including differences between the forward and aft blades.

Figure 21 shows the speed-line characteristics of the tandem and single blade rotors at 100% design speed. Mass-averaged work coefficient and polytropic efficiency (rotor only) are plotted versus flow coefficient, which here is the inlet mean axial velocity across the entire blade span normalized by tip speed. Also included is the single point 3-D equivalent value taken from the 2-D cascade studies at minimum loss (no 3-D flow imposed). The 2-D target polytropic efficiency is based upon the loss levels of the chosen 2-D geometry at the target D-Factor of 0.62.

It can be seen from Figure 21 that the tandem rotor does not meet the target performance, designated by the equivalent 2-D goal. Instead, along an approximate constant throttle line, the 3-D rotor produces a work coefficient of 0.53 at a mean flow coefficient of 0.48. The miss in target performance (~5%) obviously comes from the 3-D flow field, as opposed to 2-D cascade flow, which will be discussed later. The polytropic efficiency at this operating point is 91%, while the peak efficiency is near 92% (corresponding to a work coefficient of 0.56). While not at industry leading efficiency levels, it should be noted the equivalent 2-D polytropic efficiency is 96%, suggesting a significant amount of 3-D losses that could be mitigated. By comparison, the single rotor only achieves a dismal peak efficiency of 82.4% at $\Phi = 0.45$, and at no point does it match the tandem rotor loading. This simply indicates that the single rotor would not be chosen to operate in this design space.

Using Equation 6, the current tandem rotor has a numerical stall margin of approximately 19% from the constant throttle line and 10% from peak efficiency. In reality the tandem rotor stall margin could be even higher than what is seen on Figure 21, since CFD tools commonly underpredict stall margin.

Conventional rear stage core compressor rotors are not designed with the level of loading achieved in the tandem rotor, owing to the obvious poor performance of the single rotor as seen in Figure 21. Neglecting the pressure losses across a stator for the
moment, the increased work coefficient of around 50% from conventional designs (Figure 17) suggests that a realistic estimate of the maximum tandem rotor potential in a core compressor is to replace three conventional stages with two tandem rotor stages. Of course, this is based solely on the aerodynamic loading capability of the tandem rotor. A tandem rotor would necessarily require a wider disc than a convention rotor. The author recognizes that a full system study, i.e. simulation of three conventional stages versus two tandem stages, will be necessary to conclusively determine if the tandem rotor is viable.

In order to properly design a whole stage using tandem rotors, three main challenges must be met. The first would be to boost the rotor efficiency levels slightly above the levels seen in Figure 21 by mitigating 3-D flows. The second challenge would be to design a stator than can turn the flow from the tandem rotor without introducing prohibitively high losses. The third, admittedly, is a host of mechanical issues that are beyond the scope of this work.

An understanding of the complex flow mechanisms in a tandem rotor passage will help address the first two challenges. As such, the next section focuses on the flow field within the tandem rotor.
Figure 21: Tandem and single rotor 100% speed performance characteristics
3.2.6 Results: Tandem Rotor Fluid Mechanics

The 2-D and 3-D fluid mechanics of a conventional rotor are well documented. Reference 34 is a good example. A brief description of some of the common 3-D fluid mechanisms of a conventional rotor are given in the following paragraphs, and illustrated in Figure 22.

Skewed inlet flow at the hub approaches the blade leading edge at a high incidence angle. This increases the adverse pressure gradient on the suction side near the endwall boundary layer. The result is separation in the endwall-suction side corner. Also, flow tends to be overturned near the hub.

The midspan tends to have the most 2-D flow of any region along the blade. However, the centrifugal force of the blade on its boundary layer tends to move fluid radially outward from the hub. This can cause 3-D flows in the midspan region.

At the blade tip there is a clearance region through which fluid flows due to the pressure gradient between the two sides of the blade. One consequence of this phenomenon is the formation of a vortex as the high-speed fluid coming through the gap becomes entrained in the freestream flow. The result is under-turning of the flow and higher losses than are seen at midspan. This is a very basic description of the fluid mechanics associated with tip clearance flows, a subject that has been researched.

Figure 22: Conventional rotor 3-D fluid mechanics
(reprinted from Cumpsty [34] with permission from Krieger Publishing Co.)
extensively over the course of many decades. The curious reader can examine References 57 – 65 for more information on tip clearance flow.

This section highlights the fluid mechanics of a 3-D tandem rotor, and in particular the differences between the behavior of the forward and aft blades. Focus is placed on two operating conditions: the “throttle-line” (Φ = 0.48) and near stall (Φ = 0.39) as shown in Figure 21. The near stall point represents the condition where the forward blade core region flow best matches the 2-D equivalent. The 2-D equivalent values from the cascade studies are also provided for comparison.

3.2.6.1 Forward Blade

Figures 23–26 represent the forward blade spanwise distributions of incidence, deviation, D-Factor, and total pressure loss coefficient. For losses (Equation 3), total pressures were mass-averaged across the pitch at each radial location and normalized by the forward blade inlet values. The values were collected 2% chord upstream and downstream of the forward blade. During discussions of the flow field, the hub region is defined as 0 to 30% span, the core region as 30 to 70% span, and the tip region as 70 to 100% span.

It can be seen in Figures 23–26 that the forward blade at the near stall condition (green circles) nearly matches the 2-D values (red squares) in the core region with only a small one-degree difference in core flow deviation. At throttle-line conditions, the forward blade incidence across the entire span is 3 to 4 degrees lower. This difference in incidence does not profoundly affect the forward blade deviation in the core region, but slightly increases losses due operating closer to the negative stall side of the 2-D loss bucket. Therefore, the core flow of the forward blade acts similar to the 2-D cascade equivalent.

The flow is skewed at both the hub and tip regions, which is expected given the thick boundary layer at both endwalls, creating more positive incidence values there for the forward blade (Figure 23). This higher endwall incidence pushes the forward blade well above a 0.5 D-Factor at the endwalls (Figure 25), resulting in higher deviation and losses than the 2-D cascade values (Figures 24 & 26). As will be discussed in Section 3.2.6.2, the higher endwall loadings and losses from the forward blade propagate
downstream and strongly affect the aft blade flow. This is previewed in Figure 27, which is the overall D-Factor for the tandem rotor.

Both the hub and tip develop classical secondary flow features due to the incoming vorticity profile. The hub region features under-turned flow near 10% span and over-turned flow very near the hub with increased hub region losses due to the secondary flow. The tip region features under-turned flow and increased losses due to clearance leakage flow. As expected, with increased endwall incidence and loading (going from throttle-line to near stall conditions), the deviation and loss values increase there. Therefore, although the endwall regions do not act like 2-D cascade flow, they are not dissimilar to conventional rotor blade fluid mechanics. This has design implications, since traditional techniques to reduce endwall loss should be applicable to the forward blade.

![Figure 23: Forward blade incidence for tandem rotor and 2-D goal](image-url)
Figure 24: Forward blade deviation for tandem rotor and 2-D goal

Figure 25: Forward blade D-Factor for tandem rotor and 2-D goal
Figure 26: Forward blade losses for tandem rotor and 2-D goal

Figure 27: Overall D-Factor for tandem rotor and 2-D goal
3.2.6.2 Aft Blade

Figures 28–31 represent the aft blade spanwise distributions of incidence, deviation, D-Factor, and total pressure loss coefficient. For aft blade losses (Equation 3), total pressures were mass-averaged across the pitch at each radial location and normalized by the aft blade inlet values. The values were collected 2% chord upstream and downstream of the aft blade.

Since the forward blade discharges the air at a slightly lower deviation value, the incoming incidence values of the aft blade are slightly more negative than the 2-D cascade in the core region. This tends to offload the airfoil in the core region, making it operate slightly off minimum loss incidence. In addition the aft blade endwall regions have increased incidence due to the under-turning of the fluid of the forward blade, except very near the hub.

Like the forward blade, the deviation of the aft blade is close to the 2-D values in the core region. The radial gradient in deviation through the core region is due to overall passage secondary flow as in a conventional rotor. Deviation increases going toward the hub, but then decreases rapidly around 10% span, showing additional overturning of the flow by the aft blade at the hub. The flow also has higher-than-2-D deviation near the tip, again due to the tip clearance.

The aft blade loss profiles, however, differ from the forward blade, indicating the 3-D behavior is more severe than a conventional rotor. The aft blade hub region losses are higher than in the forward blade hub region. This is reasonable since the low-momentum, high-entropy flow from the forward blade trailing edge at the endwall discharges directly on to the aft blade leading edge. Additionally, the overturned high-entropy flow from the forward blade hub is directed toward the aft blade suction side-endwall corner at the trailing edge, visualized in Figure 32. The result is that hub losses extend further into the core region. The outward flow is also evidenced in Figure 33, which are momentum vectors on the forward and aft blade suction surfaces (same vantage point as Figure 19, except the pressure surfaces have been rendered transparent). There is a larger region of reverse flow near the aft blade hub than the forward blade hub, consistent with the higher losses experienced by the aft blade in that region (Figure 31).
Figure 28: Aft blade incidence for tandem rotor and 2-D goal

Figure 29: Aft blade deviation for tandem rotor and 2-D goal
Figure 30: Aft blade D-Factor for tandem rotor and 2-D goal

Figure 31: Aft blade losses for tandem rotor and 2-D goal
Figure 32: Tandem rotor flow passage with streamlines seeded near the hub

Figure 33: Forward and aft blade suction surface vectors
Another noteworthy difference between the forward and aft blade losses can be seen at 90% span (Figure 26 vs. 31). There the aft blade losses decrease so much that they are below the 2-D value. Consider Figure 34, which is a streamline visualization of the tip leakage flow. Portions of the forward blade leakage flow become entrained in the freestream flow to form a vortex, as seen in a conventional rotor. However, starting around 60% forward blade chord the leakage flow migrates across the entire passage into the aft blade clearance region. It is possible that injecting additional fluid through the aft blade tip gap helps reduce losses there and further redistributes the flow radially. Given the exceedingly complex nature of clearance flows and the relatively simple method of modeling it here, the reader is cautioned that there may be other causes for the apparent differences between forward and aft blade losses near 90% span.
One more difference between the forward and aft blades is flow uniformity. The flow downstream of the forward blade is fairly uniform except close to the endwalls (Figures 23 – 26). However, the aft blade loss profiles in Figure 31 are nonuniform across the entire span. To illustrate this further, consider Figures 35 & 36, which show axial momentum contours in the radial-tangential plane at the forward blade / aft blade interface and the aft blade trailing edge, respectively. Both figures are looking upstream.

At both the throttle-line and near stall conditions (Figures 35a & b) the forward blade has a fairly smooth flow field both spanwise and pitchwise. A very small region of low-velocity fluid can be seen at the suction side-endwall corner, as would be expected of even a conventional rotor (Figure 22).

At the throttle-line point the aft blade (Figure 36a) has a noticeable velocity defect at the suction side-hub corner. Also, there is a second, less intense defect to the right of the suction side that stretches across most of the span. That is the wake from the forward blade mixing out into the freestream flow. This phenomenon results in higher overall losses than would be produced by the aft blade in isolation. Stewart [66] showed that the relationship between the mixed-out and mass-averaged losses due to 2-D mixing of compressible flow is a function of the exit freestream Mach number. The forward blade exit Mach number is around 0.4 for all flow conditions, which according to Stewart results in a mixed-out loss coefficient that is 1.2 times greater than the mass-averaged value. Therefore up to 20% of the aft blade loss can be contributed to the forward blade wake mixing.

At the near-stall condition (Figure 36b) there is a larger region of low-velocity fluid at the aft blade suction side-endwall corner and the flow field is more nonuniform. (As an aside, the nonuniformities can also be seen by comparing the aft blade losses when they are mass-averaged versus area-averaged. Appendix H discusses this in greater detail.) These nonuniformities will be a key consideration when designing a stator to handle the incoming flow from the tandem rotor.

In summary, the aft blade acts less like the 2-D cascade in a rotor environment than the forward blade, and in general the extent of 3-D flow is greater than observed in conventional, lower loaded, rotors. This should not be too unexpected given the increased loading levels of the complete tandem rotor relative to a conventional rotor, thus leading
to higher secondary flows. The aft blade loss distributions are complicated by the mixing of the forward blade wakes, and if the incoming flow is improved, the aft blade losses could be reduced.

Figure 35: Axial momentum contours at forward blade / aft blade interface looking upstream
Figure 36: Axial momentum contours at aft blade trailing edge looking upstream
3.3 General Summary of 3-D Tandem Study

A first-pass tandem rotor has been numerically simulated and compared to a geometrically equivalent single blade rotor. The tandem rotor outperformed the single blade rotor in terms of work coefficient, polytropic efficiency, and stability margin, indicating that the tandem rotor is operating in a design space beyond what is suitable for conventional blades.

The forward blade tends to behave like a conventional rotor across the operating range. Near stall, the forward blade endwall losses are higher than at midspan due to positive incidence in those regions and conventional secondary flow features. The hub region is the most affected, due to high local loading coupled with the interaction at the suction side-endwall corner.

The aft blade flow is strongly affected by the forward blade exit flow, and has highly 3-D flow that differs from what is seen in a conventional rotor. High losses from the forward blade hub region propagate downstream to the aft blade. Additionally, overturning at the hub region of the forward blade directs high-entropy fluid toward the aft blade suction side-endwall corner, resulting in high aft blade hub region loss and flow nonuniformities. Low-energy fluid from the aft blade hub region propagates radially outward toward the core. Some of the forward blade tip leakage flow appears to enter the aft blade tip gap, possibly influencing the aft blade clearance flow.

As a final comment, recall that Railly & Mehra [38] and Bammert & Beelte [44] reported high losses in the hub region of their experimental rotors, but did not offer a complete explanation for why that was the case. It is entirely possible that they did not foresee the magnitude of the forward blade / aft blade relationship described above, and did not take it full consideration during their design process.
4. Conclusions

The tandem airfoil concept has been revisited to determine its potential application as a rotor in a very specific application that has not been previously examined: the rear stages of a core axial-flow compressor. It must be emphasized that the conclusions drawn from this study should be considered valid within certain constraints, namely a high hub-to-tip ratio, subsonic, shock-free, fully turbulent flow, and thick endwall boundary layers at the inlet (3-D). Also, given that this is a numerical study, the results shown in Chapters 2 and 3 should be taken as an indication of trends (e.g. relative sensitivity to percent pitch and axial overlap), and not necessarily as absolute performance levels.

The current numerical work is a “ground-up” approach to examining tandem airfoils, starting with basic 2-D cascade performance and flow physics. NACA-65 airfoils were employed because of readily available experimental data that can be used in generating tandem airfoil geometries. The knowledge gained from that study was used to develop a first-pass tandem rotor geometry that was simulated in fully viscous, 3-D rotating flow. The tandem rotor was compared to a geometrically similar single blade rotor. The CFD simulations leant themselves to thorough examination of the tandem rotor 3-D fluid mechanics. The following conclusions are drawn from the current work.

4.1 Tandem Airfoil 2-D Cascade Flow

An extensive 2-D CFD study was conducted that examined the effects of percent pitch and axial overlap over a range of overall D-Factors that represent moderate to high loading. It was shown that the optimum tandem configuration—high loading without excessive loss—was zero axial overlap and high percent pitch. Taking flow field interaction into effect, the tandem airfoils produced as much as 40% less losses than a comparable single airfoil at $D = 0.62$. Significant deviation from the optimum percent pitch / axial overlap can result in a tandem airfoil that performs more poorly than a single airfoil. Also, the tandem airfoils should have a 50 / 50 loading split to achieve the minimum loss at a given overall loading. The CFD loss vs. loading data were compared to trends in the open literature and found to be similar, providing some validation of the method.
Some of the 2-D fluid mechanisms of a tandem airfoil are very similar in nature to the external flows seen on wing-and-flap configurations. One in particular is the induced circulation from the aft airfoil onto the forward airfoil, which can be used to increase the overall loading provided that it does not push the forward airfoil to the point of separation. A second fluid mechanism also seen with a wing and flap is locally accelerated flow through the gap region of the forward and airfoils at high percent pitch. This creates a favorable pressure gradient near the aft airfoil leading edge.

Lastly, the CFD data showed that a tandem airfoil will achieve a wider incidence range than its single-airfoil counterpart at a given overall D-Factor. This is due to the fact that the aft airfoil has a lower inlet Mach number than the forward airfoil, and consequently can operate farther away from minimum-loss incidence before producing significantly higher losses. Tandem airfoil incidence range also appeared to be insensitive to the pitchwise position of the aft airfoil, providing further evidence that the forward airfoil has a more dominant effect on the incidence range.

### 4.2 Tandem Rotor 3-D Flow

The 3-D rotor geometry was developed from the 2-D study. A zero axial overlap, 85 percent pitch configuration was chosen after considering mixed-out aerodynamic losses as well as the potential mechanical and/or manufacturing difficulties associated with very high percent pitch. Flow conditions were based upon a rear stage of a core compressor, namely fully subsonic flow and a thick inlet boundary layer at both endwalls. Only 100% rotational speed was considered. Stators were omitted in order to focus on the fluid mechanics of the tandem rotor. A geometrically similar single blade rotor was simulated under the same boundary conditions.

The tandem rotor produced a work coefficient of 0.53 at a mean flow coefficient of 0.48, which was approximately a 5% miss in target performance based upon the 2-D design. The polytropic efficiency at this operating point was 91%. By comparison, the single rotor only achieved a peak efficiency of 82.4% at $\Phi = 0.45$, and at no point does it match the tandem rotor loading. The 2-D target for the tandem rotor had a polytropic efficiency of 96%, suggesting that mitigating the losses due 3-D flows in the tandem rotor would be the first step in achieving industry standards for rotor efficiency.
Assuming that such an improvement could be made, the 50% increase in loading of the tandem rotor over published conventional designs suggests that two tandem rotor stages could replace three conventional stages in the back of a core compressor.

The 3-D fluid mechanics of the forward blade are similar to a conventional rotor, while the aft blade is strongly dependent upon the flow from the forward blade. In particular, high-entropy, low-momentum fluid accumulates in the aft blade suction side-endwall (hub) corner resulting in high losses in that region. This has a noticeable effect on the core region of the aft blade, resulting in higher losses and more nonuniform flow than the forward blade.

4.3 Recommendations for Future Work

The tandem rotor has been shown to have the potential to reduce the number of stages in the back of a core compressor. However, additional analytical work will be necessary in order to develop a working prototype. The next major task will be to optimize the tandem rotor with the goal of reducing the 3-D losses. Traditional techniques to reduce endwall loss should be applicable to the forward blade, since it acts like a conventional rotor. Optimization of the aft blade will require special attention to the improved flow field of the forward blade, particularly in the hub region. Also, a working prototype should be designed using modern airfoils in order to reduce the 2-D profile losses.

The second task will be to design a stator that can remove the potentially high swirl from the tandem rotor without introducing prohibitively high losses. The knowledge gained on the tandem rotor fluid mechanics should prove useful in this task. Once an acceptable stator has been designed, a full simulation comparing two tandem rotor stages to three conventional stages will be necessary to determine the tandem rotor’s viability.

Two final considerations will be aeromechanics and manufacturing. It ultimately may be necessary to deviate from the aerodynamically optimum tandem configuration. The results contained within this work provide some insight on how sensitive the tandem airfoil aerodynamics are to key geometric parameters. A future designer will be able to take these into account when considering the possible aerodynamic concessions that must be made in order to satisfy other design criteria.
Appendix A  In-Depth Review of Tandem Airfoils in 2-D Cascade

Spraglin [5] examined tandem airfoils from a purely mathematical standpoint, using transformation theory for both direct and inverse problems. The tandem cascade was mapped onto the region between two concentric circles containing suitably-located singularities.

Ohashi [6] performed theoretical analysis on tandem airfoils of NACA 0010, 4410, and 8410 profiles, all with equal turning between the forward and aft airfoils, i.e. $\Delta \omega_{\theta,FA} = \Delta \omega_{\theta,AA}$. (It should be noted that in a tandem configuration equal turning does not necessarily equate to equal airfoil loading). He also ran low-speed (i.e. incompressible) experiments on the NACA 8410 profiles of fixed shape with variable axial overlap and percent pitch. He concluded that minimum aerodynamic loss is achieved around 80 percent pitch.

Ihlenfeld [7] performed low-speed cascade experiments on airfoils of fixed shape and variable axial overlap and percent pitch. He concluded that the lowest loss was obtained when the aft airfoil suction side boundary layer grew within the forward airfoil wake, i.e. very high percent pitch. He also suggested that the airfoils be equally loaded.

Pal [8] performed both theoretical and low-speed experimental examinations on NACA 8410 profiles of fixed shape and orientation. He varied axial overlap and percent pitch, but only considered negative overlaps.

Railly & El-Sarha [9] performed low-speed wind tunnel tests on a tandem cascade with fixed airfoil shapes. They varied overall camber by changing the stagger angles of each airfoil. They also examined several axial overlap and percent pitch combinations.

Sieverding [10] performed higher-speed ($0.5 < M_{11} < 0.8$) wind tunnel tests on high-solidity airfoils of double-circular arc (DCA) profile, where the aft airfoil had more camber than the forward airfoil. Only one combination of axial overlap and percent pitch was considered: $AO = 0$, $PP = 78$. He concluded that the axial overlap and percent pitch were not yet optimized.

Hopwood [11] ran additional higher-speed tests on the airfoils in Reference 10 but varied axial overlap and percent pitch. He concluded that the best configuration is when the airfoils act independent of one another. He ran the same tests on an equivalent single
airfoil for comparison. Additionally, he ran tests on a slotted blade by machining a slot in the equivalent single airfoil. Both the slotted and tandem airfoils gave higher pressure rise and lower losses than the equivalent single airfoil, the slotted airfoil more so than the tandem.

Yip [12] also used transformation to model tandem airfoils. The forward airfoil was transformed to a circle while the aft airfoil was represented by two singularities. The method was satisfactory for the case of the aft airfoil being smaller than the forward airfoil, but not necessarily for the more realistic case of approximately equal sized airfoils.

Dettmering [13] designed a tandem stator intended for a supersonic compressor stage. It was meant to decelerate the flow from supersonic to subsonic, remove swirl, and further decelerate the flow from high subsonic to low subsonic. 2-D and 3-D stationary cascade tests showed that losses in his tandem stator were noticeably lower compared to a single cascade for the same flow conditions.

Katsanis & McNally [14] developed a finite-difference FORTRAN code that gave blade-to-blade solutions of the two-dimensional, subsonic, compressible (or incompressible), inviscid flow problem for a circular or straight infinite cascade of tandem or slotted turbomachine blades. The method is based on the stream function.

Railly & Deeb [15] performed additional low-speed wind tunnel tests of the tandem airfoils from Reference 9 at fixed blade metal angles. They considered one value of axial overlap and two values of PP. They compared their results from tandem to a ‘single’ airfoil made by joining the tandems and filling the gap with material. The tandem airfoil demonstrated higher lift-to-drag ratio, turning, and static pressure rise than the ‘single’. Their results also indicated that at very high percent pitch the tandem airfoil experienced a loss of turning.

Mikolajczak [16] developed a potential / boundary layer code to compute flow for slotted blades in incompressible flow and compared it to experiments. He concluded that slotted airfoils offer an increase in incidence range over single airfoils. However, the slotted airfoils did not necessarily reduce the minimum loss level compared to a single airfoil.

Sanger [17] used an incompressible potential flow model with a boundary layer code to analyze several configurations of DCA tandem airfoils. He examined the effects
of several variables, including chord ratio ($C_{FA} / C_{AA}$), camber ratio ($\phi_{FA} / \phi_{AA}$), axial overlap, as well as the convergence ratio of the gap formed between the two airfoils at the overlap region. His code indicated that the best performance resulted from a chord ratio of unity and a camber ratio of 0.5. It is noteworthy that said configuration resulted in an equal loading split between the two airfoils.

Haut [18] performed low-speed wind tunnel tests based upon Sanger’s [17] results. He examined a single airfoil, a negative overlap tandem airfoil, and three positive overlap tandem airfoils. Expressing his results in terms of lift and drag coefficients, he concluded that a tandem airfoil with slight positive overlap gave the best performance.

Wu et al. [19] examined DCA tandem airfoils in a series of moderate speed ($M_{11} \sim 0.30$) wind tunnel tests. By varying stagger angles, axial overlap, and percent pitch they had 24 combinations of the tandem airfoil design parameters. For best performance they recommended percent pitch values of between 75 and 80 and more camber on the aft airfoil than the forward airfoil. They also developed an empirical relationship for optimum percent pitch as a function of axial overlap.

Benetschik & Gallus [20] developed an inviscid / viscous CFD code to analyze the flow in transonic and supersonic cascades. They performed numerical analysis on the tandem airfoil geometry in Reference 13, and compared their results to the experimental data in order to validate their code.

Sachmann & Fottner [21] performed high subsonic ($M_{11} = 0.83$) wind tunnel experiments on tandem stators with small overlap. Using fixed airfoil shapes they varied stagger and percent pitch. Their data indicated that while a high percent pitch configuration produced the lowest loss, a 50 PP configuration had a wider incidence range. They also compared their experimental data to a 2-D Euler code and found good agreement.

Roy & Saha [22-24] conducted low-speed wind tunnel tests to compare a tandem airfoil to a single airfoil using controlled diffusion airfoils (CDA). They concluded that a CDA tandem can achieve higher on-design loading than an equivalent single CDA, but the tandem has higher off-design losses.

Weber & Steinert [25] performed a computational design and experimental evaluation of a high turning, transonic tandem stator. The forward airfoil was transonic,
while the aft airfoil was subsonic with twice the solidity as the forward airfoil. They were attempting to turn the flow by 60° with $M_{11} = 1.06$ without excessive losses. Results from both the computational and experimental efforts indicated success.

Vandeputte [26] performed moderate-speed wind tunnel experiments to examine the effects of flow control on a tandem inlet guide vane (IGV). The tandem airfoil had a zero percent pitch, slightly negative overlap orientation. The forward airfoil was held in a fixed position such that the inlet flow angle was 0°. The aft airfoil was pivoted such that the exit flow angle was 55°. Both boundary layer suction (from the airfoil surface) and trailing edge blowing were used as flow control techniques. He concluded that boundary layer suction could reduce the wake momentum thickness by over 20%.

Roy & Mallik [27] performed numerical simulations on a tandem airfoil using MISES, which is potential flow code coupled with a boundary layer solver. They concluded that MISES did not accurately capture the flow physics through the gap formed at the forward airfoil / aft airfoil interface.

Canon-Falla [28] used the commercially available code FLUENT to perform an incompressible-flow computational study on tandem airfoils of NACA-65 profiles. Using one set of forward and aft airfoils of fixed shape, he simulated them first in isolation, and then in tandem at several combinations of axial overlap and percent pitch. He arrived at a similar conclusion to previous researchers: low axial overlap and high percent pitch give the best on-design performance.

Nezym & Polupan [29] performed a statistical analysis on the data from Reference 19 using Group Method of Data Handling (GMDH). Using this method, they showed a strong correlation between aerodynamic losses and the geometric parameters of airfoil chord ratio, camber ratio, and effective solidity. This is very reasonable since those parameters directly affect overall loading, and losses are proportional to loading.
Appendix B  Cascade Frame of Reference and AVDR

An ideal rectilinear cascade is an infinite row of airfoils in which the flow field is purely two-dimensional in nature. In a numerical simulation this type of flow can be achieved by creating a 2-D computational mesh of the airfoil geometry and imposing periodic boundary conditions in the pitchwise direction. It can also be achieved experimentally by placing a sufficient number of airfoil models in the test section to produce periodic flow about the instrumented airfoil. An important consideration in 2-D experimental cascade data is a parameter known as axial velocity density ratio (AVDR), defined by Equation B-1.

$$AVDR = \frac{\int_{0}^{s} \rho_2 \cdot w_2 \cdot \cos \beta_2 \cdot d\theta_2}{\int_{0}^{s} \rho_1 \cdot w_1 \cdot \cos \beta_1 \cdot d\theta_1}$$

Several works published from the late 1960s to early 1980s [67 – 72] document the effects of AVDR in cascade experiments. Boundary layers develop on the endwalls of a solid walled cascade causing the mainstream flow to contract, resulting in an AVDR greater than unity, as shown in Figure B-1 [73]. This increases velocities and reduces adverse pressure gradients in the mid-span region, particularly on the airfoil suction side. As a result, measured losses tend to be lower when AVDR > 1.0 than when AVDR = 1.0. Since the whole purpose of cascade testing is to capture the 2-D fluid mechanics as accurately as possible, the researcher should take steps to ensure that there is a minimal amount of endwall boundary layer development in the cascade. One such method is endwall or sidewall bleed, illustrated in Figure B-1.

Cascade flows with an AVDR of unity can also be simulated numerically. This is achieved by imposing a solid, inviscid boundary condition on the endwalls of the computational mesh.
Solid wall
AVDR > 1.0

Porous or slotted wall
AVDR ≅ 1.0

Endwall boundary layer/secondary flow

Air bleed or suction

Figure B-1: Cascade flow contraction (L) and AVDR control (R)
(reprinted from Reference 73 with permission from Bo Song)
Appendix C  Airfoil Geometries for 2-D Study

Section C.1 describes the tandem airfoil simple rule in detail. Section C.2 lists the geometries of the single airfoils used in the 2-D study, and Section C.3 lists the geometries of the tandem airfoils used in the percent pitch / axial overlap study.

C.1 Development of the Simple Design Rule for Tandem Airfoils

A Lieblein performance curve is one of the desired outputs from the design rule. Other outputs are the appropriate airfoil metal angles to achieve minimum loss operation at a desired overall D-Factor. Table C-1 summarizes the inputs and outputs of the design rule.

<table>
<thead>
<tr>
<th>Inputs</th>
<th>Outputs</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flow</td>
<td>$\beta_{11}$</td>
</tr>
<tr>
<td>Loading</td>
<td>$D_{FA}, D_{AA}$</td>
</tr>
<tr>
<td>Geometric</td>
<td>$\sigma_{FA}, \sigma_{AA} (s_{FA} = s_{AA})$</td>
</tr>
<tr>
<td>Losses</td>
<td></td>
</tr>
</tbody>
</table>

Table C-1: Simple Design Rule Inputs and Outputs

Two initial assumptions are made that allow for easy development of the simple design rule. The first is that there is no interaction between the flow fields of the two airfoils. This eliminates the parameters of axial overlap and percent pitch from the design rule. The second assumption is that the flow field is incompressible. While not completely accurate for most turbomachinery applications—the typical inlet Mach number to a compressor stage is above 0.3—it does allow the D-Factor (Equation 1) to be expressed in terms of effective solidity and flow angles:

$$D = \left(1 - \frac{\cos \beta_{11}}{\cos \beta_{22}}\right) + \left(\frac{\cos \beta_{11} \tan \beta_{11} - \tan \beta_{22}}{2\sigma_{eff}}\right)

\text{Equation (C-1)}$$

Equation C-1 can be used for the individual airfoils by making the appropriate substitutions for flow angles and individual airfoil solidity, i.e. substituting $\beta_1$ for $\beta_{11}, \beta_2$.
for $\beta_{22}$, and $\sigma$ for $\sigma_{\text{eff}}$. Referring to Table C-1, the flow and loading parameters can then be found by the following procedure:

1. Specify the desired forward and aft airfoil D-Factors, $D_{\text{FA}}$ & $D_{\text{AA}}$
2. Specify the desired inlet flow angle, $\beta_{11}$
3. Calculate $\beta_{12}$ using $D_{\text{FA}}$ in Equation C-1 modified for the forward airfoil such that $\beta_{12}$ is equivalent to $\beta_2$
4. Set the aft airfoil inlet flow angle equal to the forward airfoil exit flow angle, i.e. $\beta_{21} = \beta_{12}$
5. Calculate $\beta_{22}$ using $D_{\text{AA}}$ in Equation C-1 modified for the aft airfoil
6. Calculate overall D-Factor, $D_{\text{ov}}$ using $\beta_{11}$ and $\beta_{22}$ in Equation C-1

The above procedure for finding overall loading and airfoil flow angles is valid for any family of airfoils. Information on losses and required metal angles are family-specific, and can be found either experimentally or computationally.

The decision was made to use the NACA-65 family of airfoils due to the large amount of experimental data on them that is available in the open literature [30]. Since the forward and aft airfoil D-Factors have already been specified, the loss parameters ($\omega_{P,\text{FA}}$ & $\omega_{P,\text{AA}}$) for the airfoils can be individually determined from the available correlation, or from CFD generated data that closely matches the experimental data, e.g. solid line on Figure 4. The respective stagnation pressure loss coefficients, $\omega_{C,\text{FA}}$ and $\omega_{C,\text{AA}}$, can then be found by substituting the appropriate flow angles for the forward and aft airfoils in Equation 2 and solving for $\omega_{C,\text{FA}}$ and $\omega_{C,\text{AA}}$. The loss in stagnation pressure due to the individual airfoils can be superposed to form the overall loss coefficient of the tandem configuration:

$$\omega_{C,\text{ov}} = \omega_{C,\text{FA}} + \left( \frac{\cos \beta_{11}}{\cos \beta_{21}} \right)^2 \omega_{C,\text{AA}}$$  \hspace{1cm} \text{Equation (C-2)}$$

The cosine-squared term in Equation C-2 accounts for the change in dynamic head between the forward airfoil inlet and the aft airfoil inlet. The overall loss parameter can then be calculated using Equation 2. The final output from the design rule is the
metal angles for each airfoil: $\kappa_{11}$, $\kappa_{12}$, $\kappa_{21}$, $\kappa_{22}$. The NACA-65 experimental data are presented in such a way that if the individual airfoil solidity and inlet flow angle are known, one can easily obtain the required metal angles from a chart [30].

C.2 Geometries of the Single Airfoils

All of the single airfoils were NACA-65 profiles with 1.0” chord. The maximum thickness for all airfoils was 6% of the chord, positioned at 40% chord. The leading and trailing edge radii for all single airfoils were 0.7% of the chord. Only camber and spacing were varied to produce a wide range of minimum-loss D-Factors. Table C-2 lists the metal angles, cambers, and solidities for each single airfoil. The CFD results (D-Factor and loss parameter) for all the single airfoils were fitted to a third-order polynomial for display on Figure 4.

<table>
<thead>
<tr>
<th>$\kappa_1$ (deg)</th>
<th>$\kappa_2$ (deg)</th>
<th>Camber (deg)</th>
<th>Solidity, C / s</th>
</tr>
</thead>
<tbody>
<tr>
<td>64.4</td>
<td>49.4</td>
<td>15.0</td>
<td>0.76</td>
</tr>
<tr>
<td>64.4</td>
<td>49.4</td>
<td>15.0</td>
<td>1.52</td>
</tr>
<tr>
<td>64.4</td>
<td>49.4</td>
<td>15.0</td>
<td>2.28</td>
</tr>
<tr>
<td>64.4</td>
<td>49.4</td>
<td>15.0</td>
<td>3.04</td>
</tr>
<tr>
<td>64.4</td>
<td>44.4</td>
<td>20.0</td>
<td>0.76</td>
</tr>
<tr>
<td>64.4</td>
<td>44.4</td>
<td>20.0</td>
<td>1.52</td>
</tr>
<tr>
<td>64.4</td>
<td>44.4</td>
<td>20.0</td>
<td>2.28</td>
</tr>
<tr>
<td>64.4</td>
<td>44.4</td>
<td>20.0</td>
<td>3.04</td>
</tr>
<tr>
<td>64.4</td>
<td>39.1</td>
<td>25.3</td>
<td>0.76</td>
</tr>
<tr>
<td>64.4</td>
<td>39.1</td>
<td>25.3</td>
<td>1.37</td>
</tr>
<tr>
<td>64.4</td>
<td>39.1</td>
<td>25.3</td>
<td>1.52</td>
</tr>
<tr>
<td>64.4</td>
<td>39.1</td>
<td>25.3</td>
<td>1.67</td>
</tr>
<tr>
<td>64.4</td>
<td>39.1</td>
<td>25.3</td>
<td>1.82</td>
</tr>
<tr>
<td>64.4</td>
<td>39.1</td>
<td>25.3</td>
<td>2.28</td>
</tr>
<tr>
<td>64.4</td>
<td>39.1</td>
<td>25.3</td>
<td>3.04</td>
</tr>
<tr>
<td>64.4</td>
<td>34.4</td>
<td>30.0</td>
<td>0.76</td>
</tr>
<tr>
<td>64.4</td>
<td>34.4</td>
<td>30.0</td>
<td>1.52</td>
</tr>
<tr>
<td>64.4</td>
<td>34.4</td>
<td>30.0</td>
<td>2.28</td>
</tr>
<tr>
<td>64.4</td>
<td>34.4</td>
<td>30.0</td>
<td>3.04</td>
</tr>
<tr>
<td>64.4</td>
<td>29.1</td>
<td>35.0</td>
<td>0.76</td>
</tr>
<tr>
<td>64.4</td>
<td>29.1</td>
<td>35.3</td>
<td>1.52</td>
</tr>
<tr>
<td>64.4</td>
<td>29.1</td>
<td>35.3</td>
<td>2.28</td>
</tr>
<tr>
<td>64.4</td>
<td>29.1</td>
<td>35.3</td>
<td>3.04</td>
</tr>
<tr>
<td>64.4</td>
<td>24.4</td>
<td>40.0</td>
<td>3.04</td>
</tr>
<tr>
<td>64.4</td>
<td>19.4</td>
<td>45.0</td>
<td>3.04</td>
</tr>
</tbody>
</table>

Table C-2: Single airfoil geometric parameters
C.3 2-D Tandem Airfoil Geometries for the Percent Pitch / Axial Overlap Study

The simple design rule was used to develop seven different tandem airfoil configurations that would produce a wide a range of minimum-loss D-Factors assuming no flow field interaction. Once created, these configurations were examined at various percent pitch and axial overlap combinations.

All airfoils (forward and aft) had chords of 0.6675”. The forward and aft airfoils had the same pitchwise spacing for each configuration. The maximum thickness for all airfoils was 6% of the chord, positioned at 40% chord. The leading and trailing edge radii for all single airfoils were 0.7% of the chord. Thus, the tandem airfoils were geometrically similar to their single airfoil counterparts. Only metal angles and spacing were varied. Table C-3 lists individual metal angles, solidities (which are the same for both airfoils because chords and spacings are equal), and the estimated individual and overall D-Factors from the simple design rule. Effective chord and solidity are dependent upon axial overlap (Figure 2), so they are not listed.

<table>
<thead>
<tr>
<th>$\sigma_{FA} = \sigma_{AA}$</th>
<th>$\kappa_{11}$ (deg)</th>
<th>$\kappa_{12}$ (deg)</th>
<th>$\phi_{FA}$ (deg)</th>
<th>$D_{FA}$</th>
<th>$\kappa_{21}$ (deg)</th>
<th>$\kappa_{22}$ (deg)</th>
<th>$\phi_{AA}$ (deg)</th>
<th>$D_{AA}$</th>
<th>$D_{ov}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.015</td>
<td>56.9</td>
<td>46.2</td>
<td>10.7</td>
<td>0.287</td>
<td>49.2</td>
<td>35.3</td>
<td>13.9</td>
<td>0.287</td>
<td>0.420</td>
</tr>
<tr>
<td>1.015</td>
<td>57.5</td>
<td>44.0</td>
<td>13.5</td>
<td>0.326</td>
<td>48.5</td>
<td>30.6</td>
<td>18.0</td>
<td>0.326</td>
<td>0.469</td>
</tr>
<tr>
<td>1.015</td>
<td>58.2</td>
<td>41.5</td>
<td>16.7</td>
<td>0.368</td>
<td>47.6</td>
<td>24.0</td>
<td>23.7</td>
<td>0.368</td>
<td>0.520</td>
</tr>
<tr>
<td>1.215</td>
<td>63.3</td>
<td>48.9</td>
<td>14.4</td>
<td>0.395</td>
<td>53.1</td>
<td>32.9</td>
<td>20.3</td>
<td>0.396</td>
<td>0.569</td>
</tr>
<tr>
<td>1.015</td>
<td>63.7</td>
<td>42.4</td>
<td>21.3</td>
<td>0.450</td>
<td>54.0</td>
<td>18.5</td>
<td>35.5</td>
<td>0.450</td>
<td>0.619</td>
</tr>
<tr>
<td>0.815</td>
<td>63.6</td>
<td>31.4</td>
<td>32.2</td>
<td>0.519</td>
<td>47.2</td>
<td>3.2</td>
<td>44.0</td>
<td>0.519</td>
<td>0.670</td>
</tr>
<tr>
<td>1.215</td>
<td>64.1</td>
<td>34.5</td>
<td>29.7</td>
<td>0.545</td>
<td>46.5</td>
<td>-10.8</td>
<td>57.3</td>
<td>0.545</td>
<td>0.712</td>
</tr>
</tbody>
</table>

Table C-3: Tandem airfoil geometric parameters
Appendix D  Computational Methods (Additional Information)

Section D.1 of this chapter provides additional information on the computational methods used throughout the study. Section D.2 describes the general procedure used in this work for performing CFD analysis.

D.1 Overview of CFD code: ADPAC

The CFD solver employed is called Advanced Ducted Propfan Analysis Code (ADPAC). It was developed specifically to analyze ducted turbofan engines. A detailed description of the code can be found in the ADPAC manual [32]. Several particular features are described below, including mesh types and solving techniques.

D.1.1 Mesh Types

A CFD analysis consists of breaking up a physical domain into a grid, or mesh, of discrete points. Discretized versions of the governing equations are then numerically solved for each point in the mesh. This process is repeated until certain convergence criteria (discussed further in Section D.2.3) are met.

All computational domains in the current study are structured grids that can be either single-block, or multi-block, illustrated in Figure D-1 for a 2-D mesh (from Hall et al. [32]). Breaking large domains into multiple smaller blocks allows for parallel processing of solutions whereby multiple CPUs perform computations on the individual blocks, then communicate with each other to arrive at a final solution. In theory, a domain can be dissected into any number of blocks, provided enough CPUs are available. The drawback to increasing the number of blocks is greater inter-block communication time. In practice, a relatively simple geometry such as the nozzle shown in Figure D-1 would be modeled as a single-block mesh. More complex geometries such as the tandem airfoil would be modeled as a multi-block mesh.
Mesh blocks can also follow different structure patterns. One example is the H-O-H multi-block mesh illustrated in Figure D-2 from Hall et al. [32]. Block #1 is an H-mesh that serves to model the inlet to a compressor blade row. Block #2 is a body-centered O-mesh of the compressor blade itself. Block #3 is an H-mesh that serves to model the exit of the blade row. As can be seen, the H-mesh has orthogonal grid lines with the \( i \) index corresponding to the axial coordinate, the \( j \) index corresponding to the radial coordinate, and the \( k \) index corresponding to the pitchwise coordinate. By contrast, the body-centered O-mesh has an \( i \) index that follows the curvature of the airfoil surface, and a \( k \) index that runs normal to the surface.
Figure D-2: H-O-H multi-block grid system at constant $j$ index and computational domain communication
D.1.2 Multi-grid Scheme

ADPAC incorporates multi-grid (not to be confused with multi-block), which is a numerical technique intended to accelerate solution convergence. During execution every other grid line is removed and a calculation is performed on the coarser mesh, as illustrated by Figure D-3 from Hall et al. [32]. The calculated values are then imposed on the original, finer mesh and interpolated onto the points that had been eliminated from the coarser mesh. This technique is particularly useful for starting up solutions for which the initial boundary conditions may be significantly different from the final values. In order to be compatible with the multi-grid scheme, mesh features such as internal boundaries and overall size must conform to Equation D-1, where \( n \) is the index of the feature and \( m \) is the desired level of multi-grid. In principle there is no limit as to the number of coarse mesh levels that can be used, but standard practice is to use two or three.

\[
\frac{n - 1}{2^{m-1}} = \text{integer} \\
\text{Equation (D-1)}
\]

![Figure D-3: Mesh coarsening for accelerated solution convergence](image)
D.2 General CFD Procedure

The general procedure used throughout this study is outlined below.

1. Define the problem (2-D or 3-D; cascade or rotating)
2. Define geometry (single or tandem)
3. Generate computational mesh
4. Specify input and boundary conditions for solver
5. Execute solver
6. Check convergence
7. Post-process solutions

D.2.1 Mesh Generation

All computational domains were set up as passage-centered H-meshes. The inlet and exit planes of each mesh were situated one axial chord upstream and downstream of the particular airfoil geometry being modeled. Single airfoil meshes were single-block, one of which is shown in Figure D-4. The tandem meshes were created by first generating the forward and aft airfoil meshes separately. The FORTRAN program that created the tandem meshes had a subroutine that would as nearly as possible distribute \( k \)-points in the Upper and Lower Passages proportional to the pitchwise position of the aft airfoil. For example, at high percent pitch the Upper Passage would have fewer \( k \)-points than the Lower Passage.

All 2-D meshes were created to be radially thin slices of a blade profile at mid-span in order to simulate cascade flow (AVDR = 1.0). The 3-D meshes were generated using the same software as the 2-D, but with grid points in the \( j \)-direction as well as the \( i \)- and \( k \)-directions.
Figure D-4: Example single-block H-mesh used for single airfoil simulations (2-D view)
D.2.2 Grid Independence

Grid independency was ensured by taking a baseline tandem mesh and incrementally increasing the number of points on the airfoil surfaces in the axial, or \(i\)-direction to see if pertinent aerodynamic quantities (e.g. incidence, turning) changed for a given set of boundary conditions. The number of airfoil surface points was increased until there was no longer a significant variation—2 percent or less—in aerodynamic quantities. The same procedure was repeated for the number of pitchwise, or \(k\)-points in the flow passages, and later in 3-D for the number of radial, or \(j\)-points.

The number of points for a tandem mesh was controlled by the single airfoil mesh generator, since the tandem meshes were created by combining the individual forward and aft airfoil meshes. All single airfoil meshes had a minimum of 53 \(i\)-points on the airfoil surface and 69 \(k\)-points in the flow passage. A minimum of 53 \(j\)-points were used for 3-D. 2-D tandem meshes contained up to 181,000 grid points, while the 3-D meshes contained approximately 1.1 million grid points.

D.2.3 Convergence Criteria

Each simulation undergoes a number of iterations that is specified by the user prior to execution. Upon completion, ADPAC outputs a convergence history file containing several values: Inlet and exit mass flow, mass-averaged pressure ratio and adiabatic efficiency, maximum and root-mean-square (RMS) residuals, the number of grid points that have reverse flow, and the number of grid points that have supersonic flow. All of these parameters are shown for a typical 3-D tandem rotor restart simulation in Figures D-5 through D-8.

Throughout both the 2-D and 3-D study the mass flows and residuals are given primary consideration for determining solution convergence. First and foremost, inlet mass flow must be equal to exit mass flow (or be extremely close). The residuals are expressed as the log of the error between that particular iteration and the previous iteration divided by the error between the first iterations and the initial conditions. For example, a residual of -6.0 indicates that the error between the current iteration and the previous iteration is \(10^6\) times less than the error between the initial conditions and the first iterations. The maximum residual across the entire domain as well as the RMS value
of all grid points were both considered. Typically the RMS residual was less than -6.0 for a converged solution. Solutions that did not meet these criteria within the number of user-input iterations were restarted and run until they converged.

Figure D-5: Residual traces for a typical 3-D tandem rotor restart run

Figure D-6: Mass flow traces for a typical 3-D tandem rotor restart run
Figure D-7: Pressure ratio and efficiency traces for a typical 3-D tandem rotor restart run

Figure D-8: Supersonic and separated points traces for a typical 3-D tandem rotor restart run
D.2.4 Solution Post-Processing

All flow visualization was done using the commercially available software FIELDVIEW [74 & 75]. Additionally, a FORTRAN program was used to read in the raw solution files and rapidly calculate pertinent aerodynamic quantities such as flow angles and loss coefficients based upon conditions 2% chord upstream and downstream of the respective airfoil(s). The program was able to process both 2-D and 3-D solutions.
Appendix E  2-D Tandem Airfoil Loading Split

The tandem airfoils in the percent pitch / axial overlap study had been designed for a 50 / 50 loading split. Recall that the simple design rule did not take the interaction of the forward and aft airfoils into account. However, the results presented in Sections 2.2.3.1 and 2.2.3.2 indicate that flow field interaction does affect the loading split, particularly on the forward airfoil. A separate series of tandem airfoils were simulated with variable loading splits at approximately the same overall D-Factor. Axial overlap and percent pitch were held constant at 0 and 85, respectively. The loading splits were varied by adjusting only the respective cambers of each airfoil, e.g. the forward airfoil camber was increased while the aft airfoil camber was held constant to put more loading on the forward airfoil.

Table E-1 lists individual metal angles, solidities (which are the same for both airfoils because chords and spacings are equal), and the loading split and overall D-Factors from the CFD results. Effective chord and solidity are dependent upon axial overlap, so they are not listed.

<table>
<thead>
<tr>
<th>$\sigma_{FA} = \sigma_{AA}$</th>
<th>$\kappa_{11}$ (deg)</th>
<th>$\kappa_{12}$ (deg)</th>
<th>$\phi_{FA}$ (deg)</th>
<th>FA load split (%)</th>
<th>$\kappa_{21}$ (deg)</th>
<th>$\kappa_{22}$ (deg)</th>
<th>$\phi_{AA}$ (deg)</th>
<th>AA load split (%)</th>
<th>$D_{ov}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.965</td>
<td>59.7</td>
<td>48.4</td>
<td>11.3</td>
<td>44.5</td>
<td>58.4</td>
<td>14.4</td>
<td>44.0</td>
<td>55.5</td>
<td>0.623</td>
</tr>
<tr>
<td>0.965</td>
<td>62.0</td>
<td>41.9</td>
<td>20.1</td>
<td>50.1</td>
<td>53.3</td>
<td>14.0</td>
<td>39.9</td>
<td>49.9</td>
<td>0.631</td>
</tr>
<tr>
<td>1.015</td>
<td>63.7</td>
<td>42.4</td>
<td>21.3</td>
<td>51.4</td>
<td>54.0</td>
<td>18.5</td>
<td>35.5</td>
<td>48.6</td>
<td>0.623</td>
</tr>
<tr>
<td>0.965</td>
<td>66.1</td>
<td>28.5</td>
<td>37.6</td>
<td>59.0</td>
<td>41.8</td>
<td>11.8</td>
<td>30.0</td>
<td>41.0</td>
<td>0.631</td>
</tr>
</tbody>
</table>

Table E-1: Tandem airfoil geometric parameters for loading split study

Figure E-1 shows the loading split of the four different tandem airfoil configurations (rounded to the nearest one percent). There is almost no difference between the 50 / 50 and the 51 / 49 split. Loading split does not appear to have a noticeable effect on losses until one airfoil has at least 55 % of the total loading. This is reasonable since as the overall loading shifts to one airfoil that airfoil will have a greater propensity for the flow to separate.
Figure E-1: Loading split of 0 AO, 85PP tandem airfoils producing $D_{ov} \sim 0.625$
Appendix F  Review of Other Dual-Airfoil Type Rotors

The current work focused on axial-flow tandem airfoils. However, there are other dual airfoils that have been used in either experimental or production rotors. The most common production dual airfoil rotor is the splitter arrangement, which is where the forward and aft blades have a significant amount of overlap. This is in contrast to the low overlap of the tandem blade. Splitters have been commonly used in centrifugal compressors for decades (Figure F-1). Section F.1 reviews some of the work on centrifugal splitter blades since the 1970s. There is also literature on tandem blade centrifugal rotors, recent examples of which are reviewed in Section F.2.

Splitter blades in axial-flow rotors were extensively investigated in the 1970s. Those works, along with some that are more recent, are reviewed in Section F.3. Lastly, there is at least one known example of a conical-flow tandem rotor compressor, which is described in Section F.4.

F.1 Splitter Blades in Centrifugal Rotors

Ogawa & Gopalakrishnan [76 & 77], Bhargava & Gopalakrishnan [78], and Fabri [79] performed computations on splitted centrifugal rotors based upon potential flow models. Millour [80] examined the same configuration using a 3-D Euler analysis with simplified viscous forces. All noted that the primary effect of the splitters is to decrease the loading on the main blades, as well as to reduce the jet/wake effect at the rotor exit.

Figure F-1: Centrifugal rotor with splitter blades
Fradin [81] performed an extensive set of experiments on the flow fields of two centrifugal rotors: one with splitters, and one without. In both cases the flow field was transonic. The geometry of the splitters was the same as the main blades. They were circumferentially positioned half way between the main blades. The result most pertinent to the current study is that the flow field at the rotor (impeller) exit was more homogenous when the splitters were used.

Ogawa & Gopalakrishnan [76] indicated in their potential calculations that the position and size of the splitter could have a profound effect on its overall performance. Gui et al. [82] performed a series of incompressible flow regime experiments on two centrifugal fans: one with no splitter and one with variable geometry splitters. They examined the effects of splitter length, circumferential position, and stagger angle. Results indicated that while splitters do reduce the load and velocity gradients on the main blades, they also introduce additional losses that are greatly dependent upon their geometry. It was shown that the pressure coefficient increases when the splitter is placed closer to the suction side of the main blade. Increasing the length of the splitter can raise the pressure coefficient with little or no effect on efficiency. However, they indicated no rule of thumb as to the limit on splitter length, which would certainly have to be taken into account in a transonic flow field, where shocks are present.

More recent work on transonic impellers by Oana et al. [83] focused on the fraction of mass flow in the two splitter channels. Splitters are typically located at mid-pitch between the main blades. Maintaining this circumferential position, the splitter incidence angle was adjusted such that there was even mass flow rate between the two channels. This proved to increase the overall efficiency of the impeller at a given pressure ratio.

F.2 Tandem Blades in Centrifugal Rotors

A tandem blade centrifugal rotor is an axial-flow inducer at the inlet, followed by a separate radial-flow impeller. The axial- and radial-flow blades usually have zero or slightly negative overlap, and are mounted on the same rotating shaft. Two prominent examples of work on this type of rotor are reviewed here.
Josuhn-Kadner & Hoffman [84] experimentally investigated the effects of the clocking, or percent pitch, of the inducer relative to the impeller. The whole rotor had a design pressure ratio of 3.9 with an adiabatic efficiency of 93.5%. The maximum rotational speed tested was 88% due to facility limitations. They concluded that zero percent pitch gave the highest efficiency, but that the maximum efficiency penalty was only one percentage point at the worst percent pitch.

Roberts & Kacker [85] conducted a similar numerical study on a different geometry. They also concluded that the best configuration was zero percent pitch. However, their results indicated that the effect of percent pitch on overall performance was more profound than what had been reported previously: efficiency dropped as much as 3.8 percentage points at the worst percent pitch.

**F.3 Splitter Blades in Axial-Flow Rotors**

In the early 1970s the United States Air Force undertook research to build a single transonic axial flow stage with a stagnation pressure ratio of 3.0 and an adiabatic efficiency of 0.82. Inlet Mach number ranged from 1.28 at the hub to 1.52 at the tip, with a tip velocity of 1,600 ft/s [86]. Upon testing, the constructed stage fell dramatically short of design. This was blamed largely upon poor flow control within the rotor passage. Increasing the solidity seemed an obvious fix, except that it would have led to higher shock losses due to increased incidence resulting from higher blade blockage. However, since the local blade angle was lower near the aft portion of the passage, the decision was made to use a splitter, in the hopes that better flow control could be achieved without incurring additional losses [87]. Due to time constraints, Air Force researchers had to make many decisions regarding the splitter based upon engineering judgment. These are described below.

The trailing edge of the splitter was located in the same plane as the trailing edge of the main blades. The leading edge of the splitter was positioned in a region of subsonic flow. It was recognized that there would be a shock near the entrance region (Figure F-2). Therefore, the splitter leading edge was placed far enough downstream so as not to have a significant influence on the shock shape. The splitter camber line was identical to the main blade at the same location, and was circumferentially positioned at mid-pitch of the
main blades, which was recognized as not necessarily being optimum. Span was the same as the main blades [88].

The result of adding the splitter was that the rotor performance improved, but the overall stage performance was still short of goals: pressure ratio was 2.76, and efficiency was 0.68. However, the stage was much less sensitive to incidence variations at off-design conditions, indicating that the splitter improved flow control within the rotor [89].

A 2-D, inviscid computation was performed afterwards. Although limited in usefulness, the results did indicate that the lower passage of the splitter was choked at design conditions, resulting in higher losses than anticipated [90].

Fifteen years later Tzuoo et al. [91] performed both inviscid and viscid 3-D computations on Wennerstrom’s rotor. They determined that the system of passage shocks was more complex than had originally been envisioned by Wennerstrom (Figure F-2). It was also noted that by moving the splitter closer to the main blade suction side, the likelihood of choking one of the splitter passages was reduced.

Lastly, two recent papers [92 & 93] report that the axial-flow splitter rotor is being reinvestigated in China. They indicate additional improvements over Wennerstrom’s original ideas.

F.4 Conical-Flow Tandem Rotor

Wood et al. [94] experimentally investigated a conical-flow (mix between axial- and radial-flow) compressor comprised of a tandem rotor and tandem stator. Tip speed was 355.7 m/s at the rotor forward blade and 473.6 m/s for the rotor aft blade (transonic). Particular emphasis was put on tip clearance variation, which was accomplished using shims on the casing. They concluded that peak efficiency decreased by 0.024 for every 0.01 increase in clearance as a fraction of an average radial blade height. They also concluded that in general, the rotor blade element efficiency was high near the hub and dropped rapidly near the tip.
Figure F-2: Profile view of Wennerstrom’s splitter rotor with inlet passage shock (L), and shock pattern calculated by Tzuoo et al. (R)
Appendix G  2-D Mixed-Out Losses

In a typical experimental investigation of 2-D cascade flow the losses are measured $\frac{1}{2}$ to $1\frac{1}{2}$ chords downstream of the airfoil trailing edge. While this practice does allow the researcher to capture the wake profile, it does not fully account for the loss due to mixing of the wake into a uniform flow. CFD analyses of 2-D cascade flows suffer from the same shortcoming, usually because the exit plane of the computational mesh is not placed far enough downstream for the flow to become uniform. This is because the distance required for complete wake mixing is not the same for all cascade flows, thus making it difficult to establish a single length parameter for the computational exit plane.

A method has been developed to calculate the mixing loss from a 2-D CFD solution. Assuming incompressible flow, the principles of conservation of mass and momentum are applied to a control volume that is located immediately downstream of the computational mesh exit plane. The nonuniform flow field of a converged CFD solution at the exit plane is used as the inlet conditions to the control volume, and a uniform flow is imposed at the exit of the control volume. Final uniform values of velocity and pressure can be calculated and compared to the uniform values upstream of the cascade inlet. These velocities and pressures are then used to calculate a mixed-out loss coefficient and Lieblein diffusion factor.

To implement the method described above, a FORTRAN 90 code has been developed that reads CFD solution files, interpolates the flow field values on evenly spaced points at the inlet and exit, and finally performs the necessary numerical integration to solve for the mixed-out velocity and pressure. Currently, the code is setup to work with a purely 2-D solution (i.e. only one grid point in the radial direction). Improvements to the code would be to extend it to a fully 3-D mesh as well as to account for compressibility effects at the cascade exit.

The governing equations are described in Section G.1. They are converted to PLOT3d output in Section G.2, and discretized in Section G.3. Section G.4 contains the results of the code as applied to the 2-D CFD data from the percent pitch study (Section 2.2.3.1).
G.1 Governing Equations (Derivation)

a. Problem Statement: The nonuniform flow field at the exit of a 2-D stationary cascade is known from CFD data. Find the additional losses due to the mixing of the wake, i.e. the stagnation pressure level when the flow has become uniform.

b. Deliverables: Mixed-out loss coefficient, loss parameter, and D-Factor based upon uniform flow fields at the cascade Inlet and Final stations, 1 & 3 respectively.

c. Diagram:

![Diagram](image)

**Figure G-1: Model for 2-D Mixed-Out Losses**

d. Assumptions:
- Inlet flow field is completely uniform
- Exit flow angles are uniform and constant, i.e. \( \beta_2 = \beta_3 = K_1 \)
- Exit pressure is uniform, i.e. \( P_2 = K_2 \) (valid because CFD solutions impose a uniform Exit static pressure)
- Exit flow regime is incompressible
e. Analysis: Begin with Reynolds Transport Theorem for the control volume shown

\[
\frac{DN_{sys}}{Dt} = \frac{\partial}{\partial t} \iiint_{C.V.} \rho \eta \ dV + \iint_{C.S.} \rho \eta (\hat{n} \cdot \vec{V}) dA \quad \text{Equation (G-1)}
\]

where \( N \) is a system quantity (e.g. momentum, energy, etc.)

\( \eta \) is \( N \) on a per unit mass basis

\( \vec{V} \) is the local velocity vector

\( \hat{n} \) is the unit vector normal to the control surface

Treat the upper and lower bounds of the control volume as streamlines, i.e. no flow across them. Then apply conservation of mass in steady flow to Equation G-1 to get

\[
0 = \iint_{C.S.} \rho (\hat{n} \cdot \vec{V}) dA = \int_{0}^{\beta_3} \rho_2 (\theta) w_2 (\theta) \cos \beta_2 d\theta - \int_{0}^{\beta_3} \rho_3 (\theta) w_3 (\theta) \cos \beta_3 d\theta
\]

Since \( \beta_3 = \beta_2 \), the cosine terms cancel. Then solve for Final velocity, \( w_3 \)

\[
w_3 = \left( \frac{1}{\rho_3 s} \right) \int_{0}^{\beta_3} \rho_2 (\theta) w_2 (\theta) d\theta \quad \text{Equation (G-2)}
\]

Next apply conservation of momentum in steady flow to Equation G-1 to get

\[
\sum F_z = \iint_{C.S.} \rho V (\hat{n} \cdot \vec{V}) dA
\]

\[
s (P_2 - P_3) = \int_{0}^{\beta_2} \rho_2 (\theta) w_2^2 \cos \beta_2 d\theta - \int_{0}^{\beta_3} \rho_3 w_3^2 \cos \beta_3 d\theta
\]

Solve for final pressure, \( P_3 \)

\[
P_3 = P_2 + \frac{\cos \beta_2}{s} \left[ \rho_3 w_3^2 s - \int_{0}^{\beta_3} \rho_2 (\theta) w_2^2 d\theta \right] \quad \text{Equation (G-3)}
\]

Because the exit state is assumed to be incompressible, a further simplification can be made by taking the area-average of exit density and equate it to the final density

\[
\rho_3 = \left( \frac{1}{s} \right) \int_{0}^{\beta_3} \rho_2 (\theta) d\theta \quad \text{Equation (G-4)}
\]
The ideal gas law can be used to determine the final temperature

\[ T_3 = \frac{P_3}{\rho_3 R} \]  

Equation (G-5)

The local Mach No. is

\[ M_3 = \frac{w_3}{\sqrt{\gamma RT_3}} \]  

Equation (G-6)

Lastly, the final stagnation pressure is found from the isentropic relation

\[ P_{03} = P_3 \left[ 1 + \frac{(\gamma - 1)}{2} M_3^2 \right]^{\gamma/(\gamma-1)} \]  

Equation (G-7)

The D-Factor can be found using \( w_1, \ w_3, \ \beta_1, \) and \( \beta_3 \) in Equation 1. The loss coefficient, \( \omega_C \), can be found using \( P_{03} \) and the inlet conditions in Equation 3. The loss parameter, \( \omega_P \), can be found using Equation 2.
G.2 Governing Equations (Q-variable Form)

The FORTRAN program used to carry out the mixed-out calculations reads PLOT3d solution files that are output by ADPAC [32]. As such, it is necessary to express the pertinent equations in terms of PLOT3d Q-variables, which are in Cartesian coordinates. All Q-variables in the following equations are taken from the exit plane of the CFD solutions.

\[
\begin{bmatrix}
Q_1 \\
Q_2 \\
Q_3 \\
Q_4 \\
Q_5 \\
\end{bmatrix} = \begin{bmatrix}
\rho \\
\rho w_x \\
\rho w_y \\
\rho w_z \\
\frac{P}{(1 - \gamma)} + \frac{1}{2} \rho \left( w_x^2 + w_y^2 + w_z^2 \right)
\end{bmatrix} \quad \text{ALL DIMENSIONLESS}
\]

First calculate the Final density

\[
\rho_3 = \left( \frac{1}{s} \right) \int_0^\gamma Q_1(\theta) d\theta 
\]

Equation (G-4a)

Next calculate Exit static pressure and flow angles in terms of Q-variables

\[
P_2 = \frac{(\gamma - 1)}{s} \int_0^\gamma Q_5(\theta) - \frac{1}{2} \left( \frac{Q_5^2(\theta) + Q_5^2(\theta) + Q_4^2(\theta)}{Q_1(\theta)} \right) d\theta 
\]

Equation (G-8)

\[
\beta_3 = \beta_2 = \left( \frac{1}{s} \right) \int_0^\gamma \arctan \left( \frac{Q_3(\theta)}{Q_2(\theta)} \right) d\theta
\]

Equation (G-9)

Then calculate Final velocity and static pressure

\[
w_3 = \left( \frac{1}{\rho_3 s} \right) \int_0^\gamma \sqrt{Q_5^2(\theta) + Q_5^2(\theta) + Q_4^2(\theta)} d\theta
\]

Equation (G-2a)
\[
\begin{align*}
P_3 &= P_2 + \frac{\cos \beta_2}{s} \left[ \rho_3 w_3^2 s - \int_0^s \frac{Q_2^2(\theta) + Q_3^2(\theta) + Q_4^2(\theta)}{Q_1(\theta)} d\theta \right] \quad \text{Equation (G-3a)}
\end{align*}
\]

Lastly, because the \( Q \)-variables are dimensionless, express Equations G-5 & G-6 appropriately

\[
T_3 = \frac{P_3}{\rho_3} \quad \text{Equation (G-5)}
\]

\[
M_3 = \frac{w_3}{\sqrt{\gamma T_3}} \quad \text{Equation (G-6)}
\]

From here Equation G-7 is used to calculate the Final stagnation pressure, and the procedure for finding D-Factor and loss values is the same is in Section G.1.
G.3 Governing Equations (Discretized Form)

Because the CFD data are discrete points on the computational mesh the continuous Equations G-4a, G-8, G-9, G-2a, and G-3a must be represented as discrete functions. The chosen method for numerically integrating these discrete functions between two points $a$ and $b$ is to divide the domain into $n-1$ ($n$ is an odd number) rectangles of equal width $\Delta x = \frac{b-a}{n}$, each centered at point $i$ as shown in Figure G-2. This is called the mid-point method. When discretized as such, the integral is expressed as

\[
\int_{a}^{b} f(x) \, dx = \left[ \frac{(y_1 + y_n) \cdot \Delta x}{2} \right] + \sum_{i=2}^{n-1} [y_i \cdot \Delta x]
\]

Note that the points $i = 1$ and $i = n$ are both given half-weighting.

Figure G-2: Integration of Equally Spaced Discrete Data
An alternative to the mid-point method would be to place the left-hand sides of the rectangles at point $i$. The disadvantage to doing this is that value of $y_n$ will not be accounted for in the summation. Likewise, the same would hold true for the value of $y_1$ if the right-hand side of the rectangles were placed at point $i$.

Typically, the grid points along the Inlet and Exit planes of a computational mesh are unevenly spaced, as illustrated in Figure G-3. In order to use the above integration technique it is necessary to interpolate the Inlet and Exit $Q$-variables onto equally spaced points along their respective computational planes. An algorithm to perform such an interpolation has been successfully incorporated into the FORTRAN code.

![Figure G-3: Mesh with unequal grid spacing in pitchwise direction](image)

The discretized equations are designated A, B, C, D, and E correspond to Equations G-4a, G-8, G-9, G-2a, and G-3. These are programmed into the FORTRAN code. Here they are shown for use at the Exit plane. Note that $n$ must be an odd number, and $\Delta x = \frac{s}{n-1}$. 

87
A: \[ p_2 = \left( \frac{y-1}{s} \right) \sum_{i=2}^{n} \left[ \left( \frac{Q_{5,i} - Q_{3,i} + Q_{4,i}}{2Q_{1,i}} \right) \Delta x \right] + \left[ \frac{(Q_{5,1} + Q_{5,n}) \Delta x}{2} \right] \]

\[ -\left[ \left( \frac{Q_{2,1} + Q_{3,1} + Q_{4,1}}{2Q_{1,1}} \right) \Delta x \right] - \left[ \left( \frac{Q_{2,n} + Q_{3,n} + Q_{4,n}}{2Q_{1,n}} \right) \Delta x \right] \]

B: \[ \beta_3 = \left( \frac{1}{s} \right) \sum_{i=2}^{n-1} \left[ \arctan \left( \frac{Q_{3,i}}{Q_{2,i}} \right) \Delta x \right] + \left[ \arctan \left( \frac{Q_{3,1}}{Q_{2,1}} \right) + \arctan \left( \frac{Q_{3,n}}{Q_{2,n}} \right) \right] \frac{\Delta x}{2} \]

C: \[ \rho_3 = \left( \frac{1}{s} \right) \sum_{i=2}^{n} \left[ Q_{i,i} \Delta x \right] + \left[ (Q_{1,i} + Q_{1,n}) \frac{\Delta x}{2} \right] \]

D: \[ w_3 = \left( \frac{1}{\rho_1 s} \right) \sum_{i=2}^{n} \left[ \sqrt{Q_{2,i}^2 + Q_{3,i}^2 + Q_{4,i}^2} \Delta x \right] + \left[ \sqrt{Q_{2,1}^2 + Q_{3,1}^2 + Q_{4,1}^2} \frac{\Delta x}{2} \right] \]

\[ + \left[ \sqrt{Q_{2,n}^2 + Q_{3,n}^2 + Q_{4,n}^2} \frac{\Delta x}{2} \right] \]

E: \[ p_3 = p_2 + \left( \frac{\cos \beta_3}{s} \right) \left\{ \rho_3 w_3^2 - \sum_{i=2}^{n-1} \left[ \left( \frac{Q_{2,i}^2 + Q_{3,i}^2 + Q_{4,i}^2}{Q_{1,i}} \right) \Delta x \right] \right\} \]

\[ - \left[ \left( \frac{Q_{2,1}^2 + Q_{3,1}^2 + Q_{4,1}^2}{Q_{1,1}} \right) \frac{\Delta x}{2} \right] - \left[ \left( \frac{Q_{2,n}^2 + Q_{3,n}^2 + Q_{4,n}^2}{Q_{1,n}} \right) \frac{\Delta x}{2} \right] \]

For the inlet plane the equations are the same, substituting the subscript 1 for 2 and 2 for 3. Also, E is not needed at the inlet since its purpose is to calculate mixed-out pressure.
G.4 Calculated Mixed-Out Losses (Single and Tandem)

Figure G-4 is a Lieblein chart of the CFD data shown earlier on Figure 6 in Section 2.2.3.1, except that the \( y \)-axis is now the mixed-out loss parameter. Consequently, the loss values shown here are higher than in Figure 6. The general trends remain unchanged. High percent pitch, low overlap tandem airfoils still produce less loss than their single-airfoil counterparts. When the additional losses due to wake mixing are considered, 85 PP appears to be the most desirable configuration. Of course, as Cumpsty & Horlock [95] have pointed out, the mixed-out loss coefficient is of limited use, since very rarely do the wakes in actual turbomachines become uniform before entering the next blade row. All the same, it is worthwhile to understand why the 85 PP configuration appears more desirable when wake mixing is considered.

Figure G-5 shows the wake profiles of the 85 and 95 PP configurations indicated by the solid and dashed arrows on Figure G-4, respectively. In the figure, the \( x \)-axis is distance across the pitch, where 0 is the middle of the flow passage. The profiles are expressed as the local loss coefficient (no averaging) measured 2\% of the overall chord downstream of the aft airfoil trailing edge. The two configurations are identical in every way except for the position of the aft airfoil.

The 95 PP tandem airfoil produces a large wake that is the combination of both the individual forward and aft airfoil wakes. This wide, strong defect in the flow would naturally result in high shear with the freestream flow, generating significant additional entropy. At 85 PP the airfoils are far enough apart that the individual wakes have not merged, and individually they are weaker and narrower than the combined wake at 95 PP. The result is less shear against the freestream flow, a presumably desirable characteristic. Bearing in mind this as well as the potential manufacturing / mechanics difficulties, it was decided that the tandem rotor in the 3-D study should be 85 PP instead of 95 PP.
Figure G-4: Mixed-Out losses of selected percent pitch configurations at zero axial overlap

Figure G-5: Wake profiles for selected percent pitch configurations at zero axial overlap
Appendix H 3-D Rotor Flow Nonuniformities

Section H.1 defines mass-averaged and area-averaged losses. Section H.2 showcases flow nonuniformities in the tandem rotor by comparing the different spanwise trends between the mass- and area-averaged losses.

H.1 Definition of Mass-Averaged and Area-Averaged Loss Coefficients

The expression for stagnation pressure loss coefficient is shown in Equation 6. There are traditionally three different ways to calculate it in 2-D flow: mass-averaging, area-averaging, and mixed-out. There are others, as described by Cumpsty & Horlock [95] for example.

The 2-D area-averaged loss coefficient is simply an average of the stagnation pressure defect across the pitch, shown in Equation H-1. Mass-averaging takes into account pitchwise variations in axial momentum, shown in Equation H-2. Inlet stagnation and static pressures are assumed to be uniform in Equations H-1 and H-2.

\[
\omega_c (a - a) = \frac{\int_{0}^{s} (P_{0,1} - P_{0,2} (\theta))d\theta}{\int_{0}^{s} (P_{0,1} - P_{1})d\theta} \tag{Equation (H-1)}
\]

\[
\omega_c (m - a) = \frac{\int_{0}^{s} \rho_{w}\theta (\theta) * (P_{0,1} - P_{0,2} (\theta))d\theta}{\int_{0}^{s} \rho_{w}\theta (\theta) * (P_{0,1} - P_{1})d\theta} \tag{Equation (H-2)}
\]

The area-averaged loss coefficient is the easiest to determine in an experimental situation, since only one measurement (stagnation pressure) need be made downstream of the test section. One convenience CFD is that all the information required for both averages is contained within the solution. This makes it easy to compare the differences between the two averages when looking for flow nonuniformities. Uniform flows tend to produce different values of mass- and area-averaged loss coefficients, but similar trends across a 3-D blade span.
H.2 Differences in Loss Coefficients in 3-D Flow

In Chapter 3, the pitchwise mass-averaged loss coefficients for the tandem rotor were plotted at various radial locations across the span. One would expect that the mass-averaged and area-averaged losses will yield different values (mass-averaged is nearly always smaller). The spanwise trends may also differ depending upon the level of nonuniformities in the flow.

Figure H-1 showcases the mass- and area-averaged loss coefficients for the forward blade at the throttle-line point ($\Phi = 0.48$), the near stall point ($\Phi = 0.39$), and the 2-D Goal for the tandem rotor (the top chart in Figure H-1 is Figure 22 repeated for easy comparison). Except for a shift in level, there is no significant difference between the mass- and area-averaged loss profiles at the throttle-line point. Even at the near stall condition the only noticeable difference is around 10% span. The agreement in profile shapes suggests that the forward blade flow is quite uniform on- and off-design, which is consistent with the flow field visualization in Figure 34.

Figure H-2 is the same comparison as Figure H-1, except for the aft blade. Note that the scales are different for the mass- and area-averaged charts. Both trends show spanwise nonuniformities, but there are differences between the two. The area-averaged profile shows a greater loss reduction at 90% span. The area-averaged losses also follow a different trend near the hub by increasing continuously from 50 to 10% span before decreasing, whereas the mass-averaged profile has a dip in losses around 15% span. These disparities in aft blade loss profiles serve to emphasize that the flow field at the aft blade trailing edge is highly nonuniform, a serious consideration when designing a stator downstream of the rotor.
Figure H-1: Forward blade mass-averaged vs. area-averaged losses
Figure H-2: Aft blade mass-averaged vs. area-averaged losses
References


86. Wennerstrom, A. & Hearsey, R., 1971, “The Design of an Axial Compressor Stage for a Total Pressure Ratio of 3 to 1,” Aerospace Research Laboratories report AR 71-0061, Wright-Patterson AFB, Dayton, Ohio