Determination of a Simplified High-Order Vortex Equation for Radial Equilibrium with CFD Verification

Travis K. Matsumoto

Embry-Riddle Aeronautical University - Daytona Beach

Follow this and additional works at: https://commons.erau.edu/edt

Part of the Aerodynamics and Fluid Mechanics Commons

Scholarly Commons Citation

https://commons.erau.edu/edt/169
DETERMINATION OF A SIMPLIFIED HIGH-ORDER VORTEX EQUATION
FOR RADIAL EQUILIBRIUM WITH CFD VERIFICATION

By
Travis K. Matsumoto

A Thesis Submitted to the Graduate Studies Office in Partial Fulfillment of the Requirements for
the Degree of Master of Science in Aerospace Engineering

Embry-Riddle Aeronautical University
Daytona Beach, FL
Spring, 2015

Determination of a Simplified High-Order Vortex Equation for Radial Equilibrium with CFD Verification

By Travis K. Matsumoto

This thesis was prepared under the direction of the candidate's thesis committee chair, Dr. Magdy S Attia, Department of Aerospace Engineering, and has been approved by the members of his thesis committee. It was submitted to the Aerospace Engineering Department and was accepted in partial fulfillment of the requirements for the degree of Master of Science in Aerospace Engineering.

THESIS COMMITTEE

Dr. Magdy Attia
Chairman

Dr. Mark Ricklick
Member

Dr. William Engblom
Member

Department Chair, Dr. Anastasios Lyrantzis
or Graduate Program Coordinator, Dr. Yi Zhao

Dean of College of Engineering, Dr. Maj Mirmirani

Associate VP for Academics, Dr. Robert Oxley

4/3/2015
Date

4/3/15
Date

4-10-2015
Date
ACKNOWLEDGEMENTS

I would like to thank Dr. Magdy Attia for all the support, guidance, and education provided through our interactions. My loving wife, Rochelle, who has endured my frustrations and the many late nights encountered while working on this paper. Additionally, I would like to extend my gratitude to Christopher Fulton and Margot Viard for their assistance in the derivations used within this paper; Darrell Stevens and all my coworkers in the Embry-Riddle Gas Turbine Lab over the years for their assistance in this topic. None of my work would have been possible without the tireless efforts of those mentioned above.
The goal of this thesis is to determine a flow model that provides a better blade design over current design techniques utilizing a hybrid vortex model. This hybrid vortex model combines the well-established simple radial equilibrium vortex models into a higher order equation that will establish the basis for a more flow-accurate model. In this paper, we will discuss the basis and derivation of these vortex models, the shortcomings of current techniques, and verification of the new vortex model with empirical data via Computational Fluid Dynamic (CFD). The simple radial equilibrium equation set has been known to the scientific community since the first gas turbine engines designs. Active research into the vortex models associated with radial equilibrium, has declined with the advent of robust CFD solvers capable of representing fluid through turbomachinery. Since there are no closed form of the Navier-Stokes Equations in existence, CFD is bound by errors in modelling turbulence, mixing planes, boundary layer transitions, as well as other loss models that are incorporated into these programs. The Simplified High-Order Vortex Equation was utilized to increase the surge margin of up to 3.32% compared to a rotor designed using the free vortex method.
# TABLE OF CONTENTS

Acknowledgements........................................................................................................... iii

Abstract ................................................................................................................................... iv

Table of Contents ................................................................................................................... v

Table of Figures ...................................................................................................................... viii

Table of Tables ...................................................................................................................... xii

Nomenclature ......................................................................................................................... xiii

1. Problem Statement .......................................................................................................... 14

2. Background and Theory ................................................................................................... 15

2.1 Fluid Mechanics ............................................................................................................ 15

2.1.1 Continuity Equation ................................................................................................ 15

2.1.2 Conservation of Momentum ................................................................................... 16

2.2 Thermodynamics .......................................................................................................... 17

2.3 Simple Radial Equilibrium (SRE) ............................................................................... 19

2.3.1 Alternative SRE Derivation .................................................................................. 22

2.4 Former Investigations into Vortex Solutions ................................................................. 23

2.5 Actuator Disc Theory .................................................................................................... 24

2.6 Streamline Curvature Equation .................................................................................... 25

2.7 Extended SRE ................................................................................................................ 27

2.8 Simplified High-Order Vortex Equation ....................................................................... 27
3. Model and Methodology

3.1 Baseline Model Selection

3.2 Redesign Methodology

3.3 Rotor 67 Data Match

3.4 Rotor 67 Reverse Engineering

3.5 High-Order Vortex Redesign

3.5.1 High-Order Vortex Derivation

3.5.2 High-Order Vortex Procedure

3.6 2D Computational Fluid Dynamics

3.7 3D Computational Fluid Dynamics

4. Results

4.1 2D Mach Contour

4.2 Final 3D Blade Geometry

4.3 3D CFD Results

4.3.1 Baseline Model

4.3.2 Modified Vortex Models

4.4 Surge Margin Differential

5. Conclusions and Recommendations

5.1 General Conclusions

5.2 Recommendations
References ............................................................................................................................. 72

Appendix A – Meanline Results ............................................................................................. 74

Appendix B – GRAPE Mesh Output ....................................................................................... 79

Appendix C – RVCQ3D Output ............................................................................................... 100
# TABLE OF FIGURES

Figure 1: Flow field comparison between actual flow and SRE ................................................................. 19
Figure 2: Infinitesimal Fluid Element Free Body Diagram ................................................................................. 22
Figure 3: Methods to distribute discs within an axial turbo-machine ............................................................... 24
Figure 4: Streamline Comparison between SRE and Actuator Disc Theory ..................................................... 25
Figure 5: Intersection of S1 and S2 Surfaces within Blade Passage (Wu, 1952) .................................................. 26
Figure 6: Mesh of the Complete NASA Rotor 67 Annulus ............................................................................... 30
Figure 7: Chord Length versus Blade Span ......................................................................................................... 31
Figure 8: Thickness to Chord versus Blade Span ............................................................................................... 32
Figure 9: Edge Radius versus Blade Span ........................................................................................................... 33
Figure 10: Blade Redesign Flow Chart ............................................................................................................... 34
Figure 11: Surface Mesh of Blade Passage at Hub .............................................................................................. 36
Figure 12: Extracted Baseline Relative Tangential Velocity Profile .................................................................. 37
Figure 13: Baseline Rotor Characteristic Line .................................................................................................. 38
Figure 14: Vortex variation along blade span ..................................................................................................... 40
Figure 15: Rotor 67 2D Mesh at rotor hub ........................................................................................................... 41
Figure 16: Baseline Model Mach Contour at 50% Span .................................................................................... 44
Figure 17: Baseline Model Mach Contour at 0% Span ...................................................................................... 45
Figure 18: Baseline Model Mach Contour at 25% Span .................................................................................... 45
Figure 19: Baseline Model Mach Contour at 75% Span .................................................................................... 46
Figure 20: Baseline Model Mach Contour at 100% Span ................................................................................... 46
Figure 21: Case 1 Mach Contour at 0% Span ..................................................................................................... 47
Figure 22: Case 1 Mach Contour at 25% Span .................................................................................................... 47
Figure 23: Case 1 Mach Contour at 75% Span .................................................................................................... 48
<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>24</td>
<td>Case 1 Mach Contour at 100% Span</td>
<td>48</td>
</tr>
<tr>
<td>25</td>
<td>Case 2 Mach Contour at 0% Span</td>
<td>49</td>
</tr>
<tr>
<td>26</td>
<td>Case 2 Mach Contour at 25% Span</td>
<td>49</td>
</tr>
<tr>
<td>27</td>
<td>Case 2 Mach Contour at 75% Span</td>
<td>50</td>
</tr>
<tr>
<td>28</td>
<td>Case 2 Mach Contour at 100% Span</td>
<td>50</td>
</tr>
<tr>
<td>29</td>
<td>Case 3 Mach Contour at 0% Span</td>
<td>51</td>
</tr>
<tr>
<td>30</td>
<td>Case 3 Mach Contour at 25% Span</td>
<td>51</td>
</tr>
<tr>
<td>31</td>
<td>Case 3 Mach Contour at 75% Span</td>
<td>52</td>
</tr>
<tr>
<td>32</td>
<td>Case 3 Mach Contour at 100% Span</td>
<td>52</td>
</tr>
<tr>
<td>33</td>
<td>Case 4 Mach Contour at 0% Span</td>
<td>53</td>
</tr>
<tr>
<td>34</td>
<td>Case 4 Mach Contour at 25% Span</td>
<td>53</td>
</tr>
<tr>
<td>35</td>
<td>Case 4 Mach Contour at 75% Span</td>
<td>54</td>
</tr>
<tr>
<td>36</td>
<td>Case 4 Mach Contour at 100% Span</td>
<td>54</td>
</tr>
<tr>
<td>37</td>
<td>Meridional View of Case 1</td>
<td>55</td>
</tr>
<tr>
<td>38</td>
<td>Rotor 67 Stacked Blade Sections</td>
<td>56</td>
</tr>
<tr>
<td>39</td>
<td>Case 1 Stacked Blade Sections</td>
<td>56</td>
</tr>
<tr>
<td>40</td>
<td>Case 2 Stacked Blade Sections</td>
<td>57</td>
</tr>
<tr>
<td>41</td>
<td>Case 3 Stacked Blade Sections</td>
<td>57</td>
</tr>
<tr>
<td>42</td>
<td>Case 4 Stacked Blade Sections</td>
<td>58</td>
</tr>
<tr>
<td>43</td>
<td>2D Section Comparison at 0% Span</td>
<td>59</td>
</tr>
<tr>
<td>44</td>
<td>2D Section Comparison at 25% Span</td>
<td>59</td>
</tr>
<tr>
<td>45</td>
<td>2D Section Comparison at 50% Span</td>
<td>60</td>
</tr>
<tr>
<td>46</td>
<td>2D Section Comparison at 75% Span</td>
<td>60</td>
</tr>
<tr>
<td>47</td>
<td>2D Section Comparison at 100% Span</td>
<td>61</td>
</tr>
</tbody>
</table>
Figure 48: ANSYS Workbench Schematic

Figure 49: Baseline Rotor Characteristic Line

Figure 50: Case 1 to Baseline Rotor Characteristic Line Comparison

Figure 51: Case 2 to Baseline Rotor Characteristic Line Comparison

Figure 52: Case 3 to Baseline Rotor Characteristic Line Comparison

Figure 53: Case 4 to Baseline Rotor Characteristic Line Comparison

Figure 54: Case 1 Mesh at 0% Span

Figure 55: Case 1 Mesh at 25% Span

Figure 56: Case 1 Mesh at 75% Span

Figure 57: Case 1 Mesh at 100% Span

Figure 58: Case 2 Mesh at 0% Span

Figure 59: Case 2 Mesh at 25% Span

Figure 60: Case 2 Mesh at 75% Span

Figure 61: Case 2 Mesh at 100% Span

Figure 62: Case 3 Mesh at 0% Span

Figure 63: Case 3 Mesh at 25% Span

Figure 64: Case 3 Mesh at 75% Span

Figure 65: Case 3 Mesh at 100% Span

Figure 66: Case 4 Mesh at 0% Span

Figure 67: Case 4 Mesh at 25% Span

Figure 68: Case 4 Mesh at 75% Span

Figure 69: Case 4 Mesh at 100% Span

Figure 70: Rotor 67 Mesh at 0% Span

Figure 71: Rotor 67 Mesh at 25% Span
Figure 72: Rotor 67 Mesh at 50% Span ................................................................. 97
Figure 73: Rotor 67 Mesh at 75% Span ................................................................. 98
Figure 74: Rotor 67 Mesh at 100% Span ................................................................. 99
Figure 75: Case 1 Density Contour at 0% Span ...................................................... 100
Figure 76: Case 1 Density Contour at 25% Span .................................................... 101
Figure 77: Case 1 Density Contour at 75% Span .................................................... 102
Figure 78: Case 1 Density Contour at 100% Span .................................................. 103
Figure 79: Case 2 Density Contour at 0% Span ...................................................... 104
Figure 80: Case 2 Density Contour at 25% Span .................................................... 105
Figure 81: Case 2 Density Contour at 75% Span .................................................... 106
Figure 82: Case 2 Density Contour at 100% Span .................................................. 107
Figure 83: Case 3 Density Contour at 0% Span ...................................................... 108
Figure 84: Case 3 Density Contour at 25% Span .................................................... 109
Figure 85: Case 3 Density Contour at 75% Span .................................................... 110
Figure 86: Case 3 Density Contour at 100% Span .................................................. 111
Figure 87: Case 4 Density Contour at 0% Span ...................................................... 112
Figure 88: Case 4 Density Contour at 25% Span .................................................... 113
Figure 89: Case 4 Density Contour at 75% Span .................................................... 114
Figure 90: Case 4 Density Contour at 100% Span .................................................. 115
Figure 91: Rotor 67 Density Contour at 0% Span .................................................... 116
Figure 92: Rotor 67 Density Contour at 25% Span .................................................. 117
Figure 93: Rotor 67 Density Contour at 50% Span .................................................. 118
Figure 94: Rotor 67 Density Contour at 75% Span .................................................. 119
Figure 95: Rotor 67 Density Contour at 100% Span ................................................ 120
TABLE OF TABLES

Table 1: NASA Rotor 67 Design Conditions (Strazisar, Wood, Hathaway, & Suder, 1989) ..........29
Table 2: Case Modification Descriptions .................................................................................39
Table 3: Surge Margin Percent Difference .................................................................................68
Table 4: Meanline Code Results at 0% Span .............................................................................74
Table 5: Meanline Code Results at 25% Span .............................................................................75
Table 6: Meanline Code Results at 50% Span .............................................................................76
Table 7: Meanline Code Results at 75% Span .............................................................................77
Table 8: Meanline Code Results at 100% Span .........................................................................78
## NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>(a)</td>
<td>Vortex Constant Coefficient</td>
</tr>
<tr>
<td>(b)</td>
<td>Vortex Constant Coefficient</td>
</tr>
<tr>
<td>(F_b)</td>
<td>Body Force</td>
</tr>
<tr>
<td>(F_s)</td>
<td>Surface Force</td>
</tr>
<tr>
<td>(h_o)</td>
<td>Total Enthalpy per unit mass</td>
</tr>
<tr>
<td>(h)</td>
<td>Enthalpy per unit mass</td>
</tr>
<tr>
<td>(K)</td>
<td>Vortex Constant</td>
</tr>
<tr>
<td>(m)</td>
<td>Mass</td>
</tr>
<tr>
<td>(\dot{m})</td>
<td>Mass Flow per unit Time</td>
</tr>
<tr>
<td>(P_o)</td>
<td>Total Pressure</td>
</tr>
<tr>
<td>(P)</td>
<td>Pressure</td>
</tr>
<tr>
<td>(Q)</td>
<td>Heat Energy</td>
</tr>
<tr>
<td>(S)</td>
<td>Entropy</td>
</tr>
<tr>
<td>(T_o)</td>
<td>Total Temperature</td>
</tr>
<tr>
<td>(T)</td>
<td>Temperature</td>
</tr>
<tr>
<td>(U)</td>
<td>Internal Energy</td>
</tr>
<tr>
<td>(\bar{U})</td>
<td>Blade Rotational Velocity</td>
</tr>
<tr>
<td>(V_r)</td>
<td>Radial Velocity</td>
</tr>
<tr>
<td>(V_z)</td>
<td>Axial Velocity</td>
</tr>
<tr>
<td>(V_\theta)</td>
<td>Tangential Velocity</td>
</tr>
<tr>
<td>(V)</td>
<td>Velocity Magnitude</td>
</tr>
<tr>
<td>(\dot{W})</td>
<td>Work</td>
</tr>
<tr>
<td>(\alpha)</td>
<td>Absolute Flow Angle</td>
</tr>
<tr>
<td>(\beta)</td>
<td>Relative Flow Angle</td>
</tr>
<tr>
<td>(\rho)</td>
<td>Density</td>
</tr>
<tr>
<td>(\forall)</td>
<td>Volume</td>
</tr>
<tr>
<td>(\omega)</td>
<td>Blade Wheel Speed</td>
</tr>
</tbody>
</table>
1. PROBLEM STATEMENT

Gas turbine engines are continually pushed toward improved performance; higher thrust; and most importantly, higher efficiency. With the continual advancement of computational power, computational fluid dynamics (CFD) is quickly becoming the de facto method for blade optimization and improvement. However, the blades are initially designed with the simple radial equilibrium vortex solution which was developed prior to World War II. In this era the advancement of jet powered aircraft was top secret for national security. These primitive vortex solutions were developed in an era before computers, but in the current information age, a computer resides in the pockets of many. This increase in availability of computational power allows a more complex vortex solution to be used in the design process. With less simplifications a more physically accurate solution can be developed.

The performance metric of a gas turbine compressor used for this investigation is the surge margin. The surge margin, as the name suggests, is the difference between the operating point and the compressor surge line as found on a compressor map; increasing the surge margin allows the compressor a wider operational range. This wider compressor operational range allows for a more powerful and efficient gas turbine engine.

The vortex model is overlooked in the modern age and the CFD is only an optimization; therefore, CFD can only improve the performance by a limited amount. Additionally, CFD optimizations are applicable to all vortex models, a better baseline design will lead to a better optimized solution. It is the intent of this research to aid in the development of a more physically accurate vortex solution to extend the surge margin, and thus performance, of a gas turbine compressor.
2. BACKGROUND AND THEORY

2.1 Fluid Mechanics

The derivation of the simple radial equilibrium (SRE) equation as well as the simplified high-order vortex equation are derived from the continuity equation and conservation of momentum equation. The SRE equation can be derived directly as discussed further later in this paper.

2.1.1 Continuity Equation

The generalized form of the continuity equation in integral form is shown in Equation 1.

\[ \frac{\delta}{\delta t} \iiint_{CV} \rho \, d\forall = - \iiint_{CS} \rho \mathbf{V} \cdot d\mathbf{S} \]

For cylindrical coordinates, the momentum equations with no viscous or body forces are given below in the Equations 2 - 4.

\[ V_r \frac{\partial V_r}{\partial r} + V_\theta \frac{\partial V_r}{\partial \theta} + V_z \frac{\partial V_r}{\partial z} - \frac{V_\theta^2}{r} = - \frac{1}{\rho} \frac{\partial p}{\partial r} \]

\[ V_r \frac{\partial V_\theta}{\partial r} + V_\theta \frac{\partial V_\theta}{\partial \theta} + V_z \frac{\partial V_\theta}{\partial z} + \frac{V_r V_\theta}{r} = - \frac{1}{\rho r} \frac{\partial p}{\partial \theta} \]

\[ V_r \frac{\partial V_z}{\partial r} + V_\theta \frac{\partial V_z}{\partial \theta} + V_z \frac{\partial V_z}{\partial z} = - \frac{1}{\rho} \frac{\partial p}{\partial z} \]

Applying the divergence theorem to Equation 1 and transforming into cylindrical coordinates the continuity equation is transformed into equation 5.

\[ \frac{\delta}{\delta t} \iiint_{CV} \rho \, d\forall = - \iiint_{CS} \rho \left[ \frac{\delta}{\delta r} (rV_r) + \frac{\delta}{\delta \theta} (V_\theta) + \frac{\delta}{\delta z} (V_z) \right] d\forall \]

The fluid along an axial flow compressor is assumed to be steady, inviscid, and incompressible. An additional assumption of continuum, which assumes the fluid properties are maintained regardless of
size, as well as the continuity and momentum conservation allows the above equation to simplify to equation 6 found below.

\[ \frac{1}{r} \frac{\delta}{\delta r} (r \cdot V_r) + \frac{1}{r} \frac{\delta}{\delta \theta} (V_\theta) + \frac{\delta}{\delta z} (V_z) = 0 \]

2.1.2 Conservation of Momentum

Considering an infinitesimally small control volume, the total force applied must be equal to the change in momentum with respect to time as shown in Equation 7.

\[ F = \frac{d}{dt}(mV) \]

From the above equation, the right hand side can be manipulated into the integral form as shown with the following derivation. In a control volume, the momentum change can be imparted on the surface or varied internally by flow conditions (Schobeiri, 2005). The momentum change on surface is determined by the momentum flux through the surface.

\[ \frac{d}{dt}(mV)_{\text{surface}} = \iint_{CS} (\rho V \cdot dS) \cdot V \]

The time rate of change of the internal momentum is described by the following equation. The product of mass, velocity, and the differential volume taken over the control volume:

\[ (mV)_{\text{internal}} = \frac{d}{dt} \iiint_{CV} \rho \cdot V \cdot d\nu \]

Combining Equations 8 and 9 the right hand side shown below in Equation 10.

\[ \frac{d}{dt}(mV) = \iint_{CS} (\rho V \cdot dS) \cdot V + \frac{d}{dt} \iiint_{CV} \rho \cdot V \cdot d\nu \]

Similarly, the forces acting on the infinitesimally small control volume can be split into body and surface forces; however, there is an additional pressure force acting on the control volume. Leaving the surface and body forces a generic \( F \), Equation 7 is derived as the following.
\[ F - \iiint_{CS} p \cdot dS = \iiint_{CS} (\rho V \cdot dS) \cdot V + \frac{d}{dt} \iiint_{CV} \rho \cdot V \cdot dV \]

Converting Equation 11 to volume integrals, the following equation is derived.

\[ F - \iiint_{CV} (\nabla p) \cdot d\mathcal{V} = \iiint_{CV} (\rho V \cdot \nabla) V \cdot d\mathcal{V} + \iiint_{CV} \rho V \cdot (\nabla \cdot V) \cdot d\mathcal{V} + \frac{d}{dt} \iiint_{CV} \rho \cdot V \cdot d\mathcal{V} \]

Separating the above equation into the cylindrical coordinate components and simplifying, the radial component of the above equation can be rewritten in a per volume form as shown below.

\[ f_r - \frac{1}{\rho} \frac{\delta p}{\delta r} = V_r \frac{\partial V_r}{\partial r} + \left( - \frac{V_\theta^2}{r} - V_\theta \frac{\partial V_r}{\partial \theta} + V_\theta \frac{\partial V_r}{\partial r} \right) + \left( V_z \frac{\partial V_z}{\partial z} - V_z \frac{\partial V_r}{\partial z} \right) + \frac{dV_r}{dt} \]  

With the assumptions of axial symmetry, \( \frac{\delta}{\delta \theta} = 0 \), and the radial velocity set to zero, \( V_r = 0 \), the pressure form of the simple radial equilibrium equation is derived.

\[ \frac{1}{\rho} \frac{dP}{dr} = \frac{V_\theta^2}{r} \]  

2.2 Thermodynamics

The differential form of the First Law of Thermodynamics is shown below in Equation 15. The flow is assumed to undergo a reversible process and heat transfer is represented by the Second Law of Thermodynamics. These assumptions are used on Equation 15, producing Equation 16.

\[ \delta Q = \delta U + \delta W \]

\[ TdS = \delta U + p\delta V \]

Substituting the definition of enthalpy into the above, Equation 16, and differentiating with respect to the radius as well as moving to a per-mass system; the following equation is derived.

\[ \frac{\delta h}{\delta r} = T \frac{\delta s}{\delta r} + \frac{\delta T}{\delta r} \delta s + \frac{1}{\rho} \frac{\delta p}{\delta r} \]
From Equation 17, the total specific enthalpy can be derived.

$$\delta h_o = \delta h + \frac{v^2}{2}$$ \hfill 18

Separating the velocity into cylindrical components, differentiating with respect to the radius, and substituting equation 17 into equation 18, the resulting equation is determined (Schobeiri, 2005).

$$\frac{\delta h_o}{\delta r} = T \frac{\delta s}{\delta r} + \frac{V_r v^2}{r} + V_r \frac{\delta V_r}{\delta r} + V_\theta \frac{\delta V_\theta}{\delta r} + V_z \frac{\delta V_z}{\delta r}$$ \hfill 19

Additionally, equation 15 can be rewritten to solve for the work done by the rotor as shown below. If the heat transfer is assumed to be zero, the following equation is easily derived.

$$\dot{W} = (\Delta h_o) \cdot \dot{m}$$ \hfill 20

For a rotating fluid, work done can also be defined by the product of torque and angular velocity. The torque is defined as the change in angular momentum, and is shown below.

$$-\frac{\dot{W}}{\dot{m}} = \Delta (\vec{U} \cdot V_\theta)$$ \hfill 21

By substituting equation into the above equation, a direct relationship, also known as the Euler turbo-machinery equation, is derived.

$$\Delta h_o = -\Delta (\vec{U} \cdot V_\theta)$$ \hfill 22

These above relationships demonstrate the direct relationship between the work done and change in tangential velocity. This relationship leads to design challenges for turbo-machinery, a reasonable tangential velocity while still obtaining the desired work. If the required tangential velocity is too large the flow will separate from the airfoil resulting in compressor surge.
2.3 Simple Radial Equilibrium (SRE)

For the Simple Radial Equilibrium, three assumptions are made. The first assumption is the assumption of constant work per unit length of the rotor blade, $\frac{\delta h_o}{\delta r} = 0$. The second assumption is the assumption of constant radial velocity, $\frac{\delta v_r}{\delta r} = 0$. The third assumption is the constant entropy per unit length of the rotor blade, $\frac{\delta s}{\delta r} = 0$. The flow profile of the SRE is show below in Figure 1 (Cumpsty & Greitzer, 2004).

![Flow field comparison between actual flow and SRE](image)

The above assumptions are substituted into Equation 19, thus Equation 23, the velocity form of the simple radial equation.
\[
\frac{V_\theta^2}{r} + V_\theta \frac{\delta V_\theta}{\delta r} + V_z \frac{\delta V_z}{\delta r} = 0
\]

From this equation, it can be seen that the radial variations of the axial and tangential velocity components are coupled. This velocity form of the simple radial equilibrium is the most practical form for use in gas turbine engine design. This is due to the fact that the flow velocities are an integral part in the design process of gas turbine rotors. From these derivations it is clear the SRE is only valid for the following conditions:

- Axisymmetric flow
- Steady flow
- Inviscid fluid
- Incompressible fluid
- \( \frac{\delta h_\theta}{\delta r} = 0 \)
- \( \frac{\delta s}{\delta r} = 0 \)
- \( \frac{\delta V_r}{\delta r} = 0 \), at the leading and trailing edge of blade rows

Since the axial and tangential velocity are coupled, once one velocity is determined the other is easily equated by direct integration of Equation 23. If a blade is designed with a blade twist such that it produces a tangential velocity profile that follows the following relationship.

\[
V_\theta (r) = a \cdot r^n \pm \frac{b}{r}
\]

The tangential velocity is plainly governed by the exponential n. In practice the distributions with \( n = 0, \pm 1 \) are most commonly used. An exponential distribution is when \( n=0 \), free vortex when \( n=-1 \), and forced vortex when \( n=1 \) (Horlock, Axial Flow Compressor: Fluid Dynamics and
Thermodynamics, 1958). The forced vortex is sometimes called a constant reaction blade because of the following relationship.

\[
Reaction = 1 - \frac{V_z}{2U} (\tan \alpha_1 + \tan \alpha_2)
\]

By substituting Equation 24 into the above relation, it can be shown that for a forced vortex solution, the degree of reaction is constant. The degree of reaction is defined as the isentropic ratio of enthalpy change across the rotor versus the enthalpy change across the stage. From velocity triangles and the Brayton Cycle on an H-S diagram, it can be shown that the degree of reaction is given by Equation 26.

\[
\frac{V_{\theta 1} + V_{\theta 2}}{2U} = 1 - Reaction
\]

As shown in Equation 29, if \( n = 1 \), as is the case for a forced vortex solution, the blade reaction is constant along the blade span; however, this is only accurate if the relationship in Equation 26 is used to define the blade reaction.

If the tangential velocity profile is accurately described by the above model, the work done can easily be shown to be independent of the radius. This is a requirement due to the assumption of constant
work per unit length of the rotor blade \( \frac{\delta h_\theta}{\delta r} = 0 \), made in the derivation of the SRE.

2.3.1 Alternative SRE Derivation

As mentioned earlier in the paper, a more direct method exists to derive the SRE by simply looking at the forces acting on an infinitesimal volume in equilibrium, as shown in Figure 2.

From the above figure, the mass per unit depth is calculated as \( \rho \cdot r \cdot d\theta \cdot dr \) and taking the sum of forces equal to zero due to the fluid element being in equilibrium, the following derivation can be performed.
\[ r \cdot d\theta \left( P + \frac{dP}{dr} dr \right) - P \cdot r \cdot d\theta \frac{V_{\theta}^2}{r} \frac{\rho \cdot r \cdot d\theta \cdot dr}{r} = 0 \]

Equation 32, is the pressure form of the SRE as also derived from the conservation of momentum. If the flow is assumed incompressible the velocity form can be easily derived with the assumption the total pressure is independent of radial location.

### 2.4 Former Investigations into Vortex Solutions

Investigations into an improved vortex solution have occurred since the application of the SRE in gas turbine design. The current design methodology is a design-by-analysis, sensitivity, or optimization but the foundation has not changed since the 1930’s (Molinari & Dawes, 2006). Methods include the addition of terms in the tangential velocity components, the addition of radial velocity components, as well as the move toward the streamline curvature equations. The additional tangential velocity terms are an expansion of the derived SRE from above. Standard convention is to set \( n = 0, \pm 1 \); however, a solution exists for all values of \( n \in \mathbb{R} \). The more complex the tangential velocity profile, the more difficult the direct integration used for determining the corresponding axial velocity component becomes. The addition of radial velocity components allows the flow field to more accurately mimic actual flow conditions. This vortex model is called the Actuator Disc Theory and the theory poses two distinct challenges that will be discussed further in the following section. The streamline curvature equations are also known as through-flow has become the standard design methodology for turbo-machinery.
2.5 Actuator Disc Theory

The actuator disc theory is a mathematical theory and a continuation of the radial equilibrium condition of no radial velocity components behind the rotor and stators (Cohen & White, 1943). This theory replaces the blade row with an infinitesimally thin disc that causes a discontinuous tangential velocity. The first issue with the actuator disc theory arises with this transformation, where to place the disc in the flow. As shown below in Figure 3, there are multiple locations to place the disc each with a different interference pattern (Horlock & Denton, A Review of Some Early Design Practice Using Computational Fluid Dynamics and a Current Perspective, 2005).

![Figure 3: Methods to distribute discs within an axial turbo-machine (Horlock, Some Actuator-Disc Theories for the Flow of Air Through an Axial Turbo-Machine, 1958)](image)

With each method, the interference pattern must be calculated for each disc. This introduces the second difficulty of this theory. The axial and tangential flow velocities caused by the infinitesimally thin disc are mathematically described with infinite series (Horlock, Some Actuator-Disc Theories for the Flow of Air Through an Axial Turbo-Machine, 1958). This complex velocity profile removes the
requirement of radial equilibrium that is derived previously in this paper. Instead the actuator disc allows small variation within the radial velocity to allow an analytical solution to the equations of motion (Larson, 1975).

As shown in Figure 4 above, the flow variation between the two vortex models minimizes as the axial distance from the blade edge increases. Due to the development of this theory in the 1960’s, the difficulties to obtain an accurate solution limited the usage; however, with the increase in computational power since then, the mathematical difficulties are largely overcome. This increase in computational power has allowed designers to utilize another mathematical tool, the streamline curvature equations.

2.6 Streamline Curvature Equation

The basis for the streamline curvature equations are first discussed by Wu in a paper published in 1952. Again, this model is developed before the advent of the computers, and equally ahead of its time. For this reason, the equations were rarely fully utilized. The most common usage of this method is by taking the stream surface that runs through the turbo-machinery from hub to tip. This surface, the S2m surface as shown below in Figure 5, allows designers to model the rotor as a 2D system.
With computers capable of solving the stream surfaces, the relative simplicity has allowed the stream curvature method to dominate in through-flow codes. The issue with simplifying the rotor into an axisymmetric 2D stream surface is the need for non-uniform loss models. These loss models have limited accuracy if the design varies significantly from which they were developed; thus, the loss model used in the calculation along the stream surface has a larger impact on the accuracy than the mathematical approximations used to solve the stream curvature equations.

From a preliminary design aspect, the streamline curvature equations are of great importance. These equations are used to define the flow within a passage and provide no details about the physical design of each blade. Through-flow codes are capable of providing a 2D analysis of the flow through all blade passages simultaneously, thus allowing for a solid starting point when designing a compressor.
2.7 Extended SRE

For a preliminary design methodology, the SRE is a quick, easy vortex approximation allowing the designer to obtain the coupled tangential and axial velocities along the blade span. With a direct correlation to rotor work requirements, redesign via optimization can be easily performed without any computationally complex mathematical approximations. Due to the oversimplified nature of the SRE, mainly due to overcome the computational limitations at the time, an extension with higher order terms was investigated by Larson in 1975. The paper proposed including a second order term in the assumed tangential velocity profile as shown in Equation 33.

\[ V_{\theta_i} = a_2 \cdot r^2 + a_1 \cdot r + a_0 + \frac{b_i}{r} \]

From this equation the special cases of free vortex, forced vortex, and exponential are easily derived by setting the appropriate coefficients to zero (Larson, 1975). It is also possible to show that the work done by this profile is independent of span as well. The paper also details an optimization scheme to include correction factors to account for annulus flare as well as the results from the actuator disc theory; however, this paper is only investigating the performance improvement offered by the development of a simplified high-order vortex equation.

2.8 Simplified High-Order Vortex Equation

The simplified high-order vortex equation is a modification of the SRE as well. Similar to the extended SRE, additional terms are added to the tangential velocity profile. The terms added are linear and parabolic variations to the SRE velocity profile already established earlier in this paper. The purpose of these variations is to produce a more realistic tangential velocity profile without requiring complex models or model dependent loss models. With a more accurate flow profile, blades are better tailored to the actual flow conditions improving both performance and efficiency. As mentioned earlier in the
paper, the optimization scheme used in current SRE designs will still be applicable for this modified velocity profile. The goal is to produce a better baseline design from which to work from. The new profile will still follow the assumptions detailed above, most importantly, constant work per unit span and constant radial velocity along the leading and trailing edge of the blade.

The focus of this paper is the determination of an improved vortex model, therefore a pre-existing rotor is selected as a baseline comparison; as well as, the initial design point. NASA Rotor 67 is chosen as the baseline model for this paper, reasons for this selection as well as more details about the SRE modifications will be discussed in chapter 3.
3. MODEL AND METHODOLOGY

3.1 Baseline Model Selection

As mentioned previously, the goal of this paper is to improve the surge margin by varying the tangential velocity profile. For this reason, the modifications must maintain a similar blade loading as the original blade. As mentioned earlier in this paper, the NASA rotor 67 is chosen as the baseline rotor design. The decision to use NASA rotor 67 is brought about by the desire to verify the CFD analysis performed on the blades. An investigation of the flow field around NASA rotor 67 using a laser anemometer was performed at the NASA Lewis single stage compressor test facility (Strazisar, Wood, Hathaway, & Suder, 1989).

From the testing and the associated data found in NASA-TP-2879, a characteristic line of the compressor map as well as experimentally determined flow characteristics, are published and the rotor design conditions are found in Table 1. This published data acts as a verification tool for all CFD

### Table 1: NASA Rotor 67 Design Conditions (Strazisar, Wood, Hathaway, & Suder, 1989)

<table>
<thead>
<tr>
<th>Description</th>
<th>Value</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>Total Pressure Ratio</td>
<td>1.63</td>
<td>-</td>
</tr>
<tr>
<td>Mass Flow</td>
<td>33.25</td>
<td>kg/s</td>
</tr>
<tr>
<td>Circumferential velocity of the blade tip</td>
<td>429</td>
<td>m/s</td>
</tr>
<tr>
<td>Relative Mach Number of the blade tip</td>
<td>1.38</td>
<td>-</td>
</tr>
<tr>
<td>Number of Blades</td>
<td>22</td>
<td>-</td>
</tr>
<tr>
<td>Blade Aspect Ratio, based on average span and the root axial chord</td>
<td>1.56</td>
<td>-</td>
</tr>
<tr>
<td>Relative Profile Thickness on the casing/blade tip</td>
<td>3.114/1.29</td>
<td>-</td>
</tr>
<tr>
<td>Casing diameter, inlet</td>
<td>0.514</td>
<td>m</td>
</tr>
<tr>
<td>Casing diameter, outlet</td>
<td>0.485</td>
<td>m</td>
</tr>
<tr>
<td>Hub diameter, inlet</td>
<td>0.193</td>
<td>m</td>
</tr>
<tr>
<td>Hub diameter, outlet</td>
<td>0.232</td>
<td>m</td>
</tr>
<tr>
<td>Tip Gap</td>
<td>0.001</td>
<td>m</td>
</tr>
<tr>
<td>Reference Temperature</td>
<td>288.15</td>
<td>K</td>
</tr>
<tr>
<td>Reference Pressure</td>
<td>1.01325</td>
<td>bar</td>
</tr>
</tbody>
</table>
performed on the NASA rotor 67 model as shown below in Figure 6. Included in the document are 14 blade profiles along the span. These airfoil points are verified by NASA to accurately match the experimentally obtained flow conditions. The authors note a difference between the calculated hot blade and the laser anemometer data. Only hot blades are investigated in this paper to match the points provided. From the provided airfoil data several design trends are easily extrapolated. These trends are useful in reverse engineering the rotor.
different blade designs, the redesigned blades must match several physical criteria as well as the aerodynamic and thermodynamic similarities discussed earlier. Matching the meridional chord length along the span is done to physically match all modified blades to the baseline rotor (National Aeronautics and Space Administration, 1965).

**Chord Length versus Blade Span**

![Chord Length versus Blade Span](image)

There are two chord lengths in Figure 7, the chord length is the actual chord with respect to the 3D airfoil data. The planar chord length is a 2D projection onto the meridional plane, the axisymmetric plane of the component. For modified blades, only the meridional chord length is matched. This method allows the modified rotor to vary in the tangential direction, thus allowing a change in blade stagger to match the required Mach profiles along the blade surface.
The second trend is the thickness to chord versus blade span as shown in Figure 8. The thickness will have a profound impact on the Mach number contours of the suction and pressure surfaces of the blade. These Mach number contours are directly related to the blade loading, thus a similar thickness to chord ratio must be matched along the blade span.

The third trend is the edge radius versus blade span as shown in Figure 9. The edge radius is important in determining the deviation between the designed metal angle and the actual flow angle. If this difference is increased the flow will not follow the designed aerodynamics of the rotor; if the difference is continually increased the flow will have a tendency to separate causing a rotating stall or surge.
The three parameters modeled above are not fixed quantities of the blade redesign. All future blade designs initially fulfill all these parameters but in the process described in section 3.2, the values are allowed to vary in order to maintain a constant blade loading for all blade sections.

### 3.2 Redesign Methodology

The redesign methodology was performed simultaneously for all vortex cases. The procedure of this redesign is outlined in the block diagram, shown below in Figure 10, and is described briefly in this section. The following sections provide greater detail on the procedure and desired outcome of each step.
The first step is to recreate the experimental data documented in NASA-TP-2879. This document provided 14 section cuts throughout the span as well as the upstream conditions. With this blade data as a starting point, the aerodynamic and thermodynamic performance can be modeled and compared to empirical data given in NASA-TP-2879. Once the CFD software settings are determined by which the most accurate results are obtained, the aerodynamic and thermodynamic properties can be reverse engineered.

![Blade Redesign Flow Chart](image)

*Figure 10: Blade Redesign Flow Chart*
The second step is to determine the velocity triangles at the leading and trailing edges. With these values and thermodynamic relationships, the aerodynamic and thermodynamic properties of the rotor can be determined.

From this point, the hybrid vortex models are implemented producing a unique blade twist for each model. For simplicity, the vortex model blades are only modeled using 5 section cuts versus the 14 given in NASA-TP-2879. Using the data extracted from the baseline model, NASA Rotor 67, new airfoils are generated. These airfoils are modified until they maintain a similar Mach profile as the original model to ensure similar blade loading. With the 2D sections similar, the blade can be restacked and 3D CFD can be performed.

### 3.3 Rotor 67 Data Match

As stated above, NASA-TP-2879 provides the blade geometry as well as the measured empirical data. From this measured data a 3D model is created and imported into ANSYS BladeGen. From this point, the ANSYS Workbench work flow is followed. After importation into ANSYS BladeGen, the flow direction and rotation direction are verified before importing the model into ANSYS TurboGrid.

Within ANSYS TurboGrid, one passage is modeled as shown in Figure 11. An inlet and outlet domain are structured H-grids to minimize computational complexity in areas that are not of interest in this study. The blade domain is a standard C-grid orientation. The grid is split to allow a smoother contour mapping of the blade twist as per program defaults. The tip clearance is modeled in ANSYS TurboGrid and is done so with a sliding mesh interface. With mostly default parameters the rotor is meshed producing approximately 200k node mesh.
This mesh was continuously refined in problems areas such as the leading edge, trailing edge, and the blade tip. The final mesh results in approximately a 1 million node mesh. This mesh was then transferred into the Computational Fluid Dynamics solver, ANSYS CFX. Again default inputs applicable to a turbomachinery problem are applied, along with all relevant data from NASA-TP-2879. For all the experimental data, total pressure and total temperature at the inlet are user input. Various outlet conditions are tested in an effort to match the provided characteristic line provided in NASA-TP-2879. A prescribed exit static pressure allows the rotor to choke while a prescribed mass flow allows the airfoil to stall with relative ease.
3.4 Rotor 67 Reverse Engineering

With the performance characteristics matching between the empirical data as well as the numerical model, NASA Rotor 67 can be reversed engineered. The accuracy of this step will determine the outcome of the work done in this paper. The first step is to solve the meanline conditions of the rotor. ANSYS CFX Post allows various flow conditions to be extracted from the data-matched numerical model. From this data the blade is verified to be designed via radial equilibrium. From this data, thermodynamic relationships are applied to the flow. From the solved mid-span values, other sections are solved using the radial equilibrium relationships verified from the exported data.

\[ W_\theta (\text{m/s}) \text{ vs. Radius} \]

*Figure 12: Extracted Baseline Relative Tangential Velocity Profile*
The velocity triangles for the five sections being investigated are created using the following extracted flow data and associated thermodynamic relationships: relative tangential velocity profile, Figure 12; pressure ratio, Figure 13; and the axial velocity profile. From this data, a linear curve fit was applied. Ignoring the data at the tip due to tip losses, an experimentally determined vortex constant is determined. From this data the five sections are calculated at the hub, 25\% span, mid-span, 75\% span, and the tip. The vortex constant used to design NASA Rotor 67 is representative of the free vortex method. The variation of this vortex constant is the heart of this paper and is discussed in the next section. The characteristic line shown in Figure 13, is the baseline that all future redesigns will be based against. An increased pressure ratio for a given mass flow indicates an increased surge margin.

Figure 13: Baseline Rotor Characteristic Line
3.5 High-Order Vortex Redesign

3.5.1 High-Order Vortex Derivation

From Equation 24 as derived above, it can be shown that the free vortex model can be reduced to a linear model. This linear model shown below in Equation 34, is the vortex model used to design NASA Rotor 67 and is the basis of the modifications that were performed for this paper.

\[ V_\theta(r) = \frac{k}{r} \]

Table 2 shown below lists the various modifications done to Equation 24.

<table>
<thead>
<tr>
<th>Case</th>
<th>Modification</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Linear variation with the Tip having the highest value</td>
</tr>
<tr>
<td>2</td>
<td>Linear variation with the Tip having the lowest value</td>
</tr>
<tr>
<td>3</td>
<td>Parabolic variation with the Tip/Hub having the highest value</td>
</tr>
<tr>
<td>4</td>
<td>Parabolic variation with the Tip/Hub having the lowest value</td>
</tr>
</tbody>
</table>

A linear radial variation of the vortex constant, as found in case 1 and case 2, is shown below in Equation 37 and is shown to be a hybrid vortex model encompassing both the free vortex and exponential distribution.

\[ V_\theta(r) * r = k + \alpha * r \]

\[ V_\theta(r) = \frac{k}{r} + \alpha * r^0 \]

\[ V_\theta(r) = \alpha * r^0 + \frac{a \pm b}{r} \]

A parabolic variation of the vortex constant, as found in case 3 and case 4, is shown below in Equation 40 and is shown to be a hybrid vortex model encompassing both the free vortex and force vortex models.
This derivation, as shown above, assumes the calculated vortex constant \( k \) for NASA Rotor 67, is a linear combination of the coefficients \( a \) and \( b \) in Equation 24.

3.5.2 **High-Order Vortex Procedure**

To implement these changes on NASA Rotor 67, the vortex model is modified by an addition with a maximum value of approximately \( \pm 20\% \) with respect to the baseline model for the linear variations, Case 1 and Case 2. For the parabolic variation the vortex constant is modified by a maximum of approximately \( \pm 25\% \) with respect to the baseline model. The variation, as described above in Table 2, is shown below in Figure 14.

![Vortex variation along blade span](image)
The values chosen for these variations are a preliminary modification and future research, as discussed in section 5.2, should be further investigated for this topic. All cases share a mid-span blade section with no variation to the vortex constant, thus producing the same theoretical velocity triangle in that location. This is done to allow as much of a direct correlation between cases by providing the same meanline conditions for all cases.

3.6 2D Computational Fluid Dynamics

With a meanline design and velocity triangles determined for various spanwise locations, the Mach profile of each section must be matched for all new cases. The meanline code does not specify the airfoil thickness or stagger angles, thus allowing a design to float to match the Mach contour requirements. To calculate the section Mach number, each section is meshed using the 2D NASA code, GRAPE. The NASA GRAPE code is a 2D meshing code that produces a C-grid around an airfoil, as shown in Figure 15.

Figure 15: Rotor 67 2D Mesh at rotor hub
The grid is calculated from user input boundary points with all other points being calculated as a solution to Poisson's equation. The flow is then analyzed using the quasi-3D NASA code, RVCQ3D. The NASA RVCQ3D code accounts for 3D effects such as rotation, radius change, and stream surface variation (Lakshminarayana, 2004). The code utilizes a second-order central difference multistage Runge-Kutta scheme, third-order accurate artificial viscosity, as well as the Wilcox k-ω Turbulence Model to determine the flow through the passage. Each spanwise location of the blade is compared for all cases as well as the baseline model, if the Mach contour does not match the established baseline the thickness and stagger angles are modified to adjust the calculated Mach contour. This iterative process is repeated on each section until all sections match Mach contours with the corresponding baseline contour. The stacking axis is assumed the same for all cases and is assumed to be equal to the baseline model. With the 2D sections stacked a 3D analysis can be performed.

### 3.7 3D Computational Fluid Dynamics

To perform the modifications required for the 2D sections, ANSYS BladeGen was the sole program used. This allowed a quick method to perform 3D analysis. With each blade being sectioned for the 2D analysis, the 3D blade was already designed. Using the same mesh parameters as the baseline model, the 3D blade of each case was easily meshed with similar attributes. The inlet and exit domains remained the same for all cases; as well as, the physical geometry of the casing and hub. With a 3D mesh, each case was modeled at the choke condition found by the baseline model as well as the design condition given in NASA-TP-2879. To investigate the improvement of the compressor surge margin, all data calculated from the modified vortex cases are from the region closer to the stall line. It is assumed that the cases will perform similar from choke to design and the most gain will occur close to the stall region.
The solver used to calculate the rotor performance was ANSYS CFX. The upstream conditions never changed for all cases and runs. The exit mass flow or exit static pressure was user specified and the inlet mass flow was monitored to verify continuity. Only near the choke region, due to the difficulties of capturing compressor choke, was the outlet static pressure used. To reach convergence was a two-step process, the first step was the MAX residual was calculated as less than 1E-03 or oscillation of the flow residuals, mass and momentum equations. Once that point was reached, the flow continuity was matched within 0.1% difference between inlet and outlet. The closer to stall the more the residuals would oscillate. It should be noted the RMS values for the flow residuals would reach a point of less than 1E-04 for all cases. The baseline mode has mass flow increments of 0.25 kg/s for the baseline model, the vortex cases incremented the mass flow by 0.50 kg/s. This was done in an effort to minimize computational time required for each case due to short comings discussed further in Section 5.2.
4. RESULTS

4.1 2D Mach Contour

With the baseline model 3D results established, the 2D Mach contours are mapped for each section cut: 0%, 25%, 50%, 75%, and 100% of blade span. The 50% span section remains constant for all cases; therefore, the mid-span Mach contour is shown below in Figure 16. For the 2D sections the leading edge of the 0% span section and is the origin of all axial distances. Due to blade sweep only the 0% span will start at an axial distance of 0 cm.

![Rotor 67](image)

*Figure 16: Baseline Model Mach Contour at 50% Span*

The remaining section Mach contours for the baseline model are shown in Figure 17 to Figure 20. As seen in Figure 17, the flow exhibits separation along the surface. This separation causes a ripple of the Mach contour and was not matched due to the transient nature of separation.
Figure 17: Baseline Model Mach Contour at 0% Span

Figure 18: Baseline Model Mach Contour at 25% Span
Figure 19: Baseline Model Mach Contour at 75% Span

Figure 20: Baseline Model Mach Contour at 100% Span
Figure 21: Case 1 Mach Contour at 0% Span

Figure 22: Case 1 Mach Contour at 25% Span
Figure 23: Case 1 Mach Contour at 75% Span

Figure 24: Case 1 Mach Contour at 100% Span
Figure 25: Case 2 Mach Contour at 0% Span

Figure 26: Case 2 Mach Contour at 25% Span
Figure 27: Case 2 Mach Contour at 75% Span

Figure 28: Case 2 Mach Contour at 100% Span
Figure 29: Case 3 Mach Contour at 0% Span

Figure 30: Case 3 Mach Contour at 25% Span
Figure 31: Case 3 Mach Contour at 75% Span

Figure 32: Case 3 Mach Contour at 100% Span
Figure 33: Case 4 Mach Contour at 0% Span

Figure 34: Case 4 Mach Contour at 25% Span
Figure 35: Case 4 Mach Contour at 75% Span

Figure 36: Case 4 Mach Contour at 100% Span
As shown above in Figure 21 (case 1) and Figure 33 (case 4), the vortex model has drastically eliminated the separation of the flow at the 0% span. The cases associated with these sections are more likely to exhibit desirable characteristics in the 3D CFD and are discussed more in Section 4.3. These sections are ignored for this 2D redesign; however, the improvement seen is cause for interest in future research as discussed in Section 5.2.

4.2 Final 3D Blade Geometry

With the 2D sections of interest exhibiting similar blade loading to the original, the 3D blade geometry is created. It is assumed that the modifications to the vortex theory discussed in this paper have a minimal effect on blade staking and therefore all redesigned blades have the same stacking axis as the original baseline model. Figure 38 to Figure 42 shows the stacking of each 2D section, with each section color coded. The rotational direction of this compressor is clock-wise, therefore the direction of rotation of these blades is toward the bottom of the page. The meridional view from ANSYS BladeGen is similar for all cases and Case 1 is shown below in Figure 37. The dashed blue line represents the leading edge of the 0% span section and is the origin of all axial distances used in this paper.
Figure 38: Rotor 67 Stacked Blade Sections

Figure 39: Case 1 Stacked Blade Sections
Figure 40: Case 2 Stacked Blade Sections

Figure 41: Case 3 Stacked Blade Sections
To help visualize the physical change to each blade, Figure 43 to Figure 47 is an overlay of each blade at a given section. From these figures, the effects on blade geometry of the varied vortex model is shown. The 0% span demonstrates the largest change due to the relationship of the exit $\beta$ with the radial dependence of tangential velocity. This same relationship causes the change of a few degrees at the tip. The ability to vary the stagger angle is apparent as well in the 0% span section. This variance allows the trailing edge to shift tangentially, vertically in the graph, producing variously cambered blades for all cases of the modified vortex constant.
Figure 43: 2D Section Comparison at 0% Span

Figure 44: 2D Section Comparison at 25% Span
Figure 45: 2D Section Comparison at 50% Span

Figure 46: 2D Section Comparison at 75% Span
4.3  3D CFD Results

The baseline model has been discussed in Section 3.3, but the following section will go into more details about the 3D CFD procedure and results. The 3D CFD results of each case are also discussed in the subsequent sections. As mentioned in Section 3.7, the region close to stall is the area of interest and therefore all data was gathered within that region.

4.3.1  Baseline Model

As discussed in Section 3.4, the baseline model is compared to the results shown in NASA-TP-2879. The results shown previously in Figure 13, and shown below for convenience, shows the characteristic line overlaid to the measured data.
The baseline data is comprised of two data sets. For areas close to the choke line, the outlet static pressure is specified and the mass flow is monitored to guarantee continuity is upheld. The region close to the stall line is determined by specifying a mass flow through the exit and monitoring the mass flow difference to guarantee continuity. The region close to the designed operating point is overlapped by both techniques and both methods provide similar results in this region. The data shows an accurate 3D CFD model until close to the stall region. As the airfoil approaches stall, the CFD under predicts the pressure ratio. The errors shown below are caused by a numerous factors, a few
causes could be: the lack of Fluid-Structural Interactions (FSI), empirical measurement uncertainty, CFD turbulence model inaccuracies, and other flow imperfections not mentioned or modeled. The goal of this paper is to measure the increase in surge margin, these inaccuracies will exist in all cases and allow a consistent error that can be accounted for.

4.3.2 Modified Vortex Models

The modified vortex models are all derived from copies of the baseline model ANSYS Workbench. The 3D model from BladeGen was modified for each vortex model and the model is directly linked to the TurboGrid module. Within the module, the user options do not change between copies. This method allowed the mesh options to remain constant for all cases as well as the baseline model. The initial model and mesh workbench layout is found below in Figure 49.

Figure 49: ANSYS Workbench Schematic

The initial static pressure exit case as well as the designed mass flow case are also attached in the above system. The initial cases are duplicated and linked to guarantee the same model is used for each point along the characteristic line. The solution is linked to the previous solved point to aid in convergence within the stall region. The determination that the modified vortex case has reached stall is determined by the pressure ratio. A sudden drop in pressure ratio corresponds to the airfoils reaching stall
conditions. The only exception is case 3, parabolic variation with the Tip/Hub having the highest value. This case was deemed finished after the mass flow had decreased by 13% but the Isentropic Efficiency has risen to over 99%. This increased efficiency is apparent in most cases when approaching stall and is most likely a numerical issue with the CFD. The implications of this error are discussed in Section 5.2.

Case 1, Figure 50, demonstrates a marginal increase in surge margin with the stall located just beyond the lower limit of the flow shown in the figure. The characteristic line is also at a higher pressure ratio along the region investigated.

**Characteristic Line of Case 1**

*Figure 50: Case 1 to Baseline Rotor Characteristic Line Comparison*
Case 2, Figure 51, is the only case to show a dramatic decline in performance. The effect of the vortex modification has caused a downward shift of the rotor stall line. This design fails to match the performance of the original design.
Case 3, Figure 52, is another case with an increase in performance. The stall line is moved to a lower mass flow and the characteristic line is at higher pressure ratios.

Case 4, Figure 53, shows a marginal performance increase but the stall line is shifted to a higher mass flow. This vortex modification is a tradeoff between an increased surge margin and a narrower characteristic line. For all cases the Isentropic Efficiency is approximately 91.5 ± 0.5%, at the design point of 33.25 kg/s. As the flow decreases the efficiency decreases until approximately the same location as the stall region shown in NASA-TP-2879. As the mass flow continues to decrease, the efficiency suddenly jumps to approximately 95% and continues to increase as the mass flow decreases.
until stall is reached, only case 2 does not exhibit this peculiarity due to the stall happening before the baseline. For this reason the results of this paper should be used in a preliminary fashion; however, these increases in Isentropic Efficiencies might arise from numerical errors in the calculations and will be ignored for the remainder of this paper.

**Characteristic Line of Case 4**

![Figure 53: Case 4 to Baseline Rotor Characteristic Line Comparison](image)

4.4 Surge Margin Differential

Due to the baseline rotor reaching stall at 31.75 kg/s and some cases extending beyond that point, the percent difference will only be calculated from the design point to the baseline stall. Any point beyond that is considered a 100% improvement and is omitted from Table 3 shown below. The data is excluded due to the uncertainty of the data accuracy caused by the high calculated efficiency.
Table 3: Surge Margin Percent Difference

<table>
<thead>
<tr>
<th></th>
<th>31.75 (kg/s)</th>
<th>32.25 (kg/s)</th>
<th>32.75 (kg/s)</th>
<th>33.25 (kg/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case 1</td>
<td>3.32%</td>
<td>2.50%</td>
<td>0.33%</td>
<td>0.22%</td>
</tr>
<tr>
<td>Case 2</td>
<td>-17.94%</td>
<td>-6.56%</td>
<td>-0.45%</td>
<td>0.24%</td>
</tr>
<tr>
<td>Case 3</td>
<td>1.54%</td>
<td>0.67%</td>
<td>-2.08%</td>
<td>-1.21%</td>
</tr>
<tr>
<td>Case 4</td>
<td>-15.54%</td>
<td>3.28%</td>
<td>0.40%</td>
<td>1.08%</td>
</tr>
</tbody>
</table>

An interesting correlation can be interpreted from the data. If the tip has a lower vortex constant, which corresponds to a higher tangential velocity and thus higher exit $\beta$, the rotor will stall earlier. This flow characteristic is consistent with the expected results. The increased tangential velocity causes the flow to separate from the airfoil due to the higher change in $\beta$ across the rotor. I suspect that case 4 outperforms case 2 due to the increase in work done by the hub sections of the blade, an increased tangential velocity is proportional to an increase in the work done by the rotor as shown in Equation 22. The lower performance of case 3 close to the design point can also be explained by this relationship. Ignoring the extension of the characteristic line, case 1 and 3 accomplished the goal of this paper. Both cases are not without the caveat of the unrealistic efficiency beyond the stall region; however, within the confines of a direct comparison to the baseline NASA rotor 67 and the redesigned blades using the modified vortex, there is improvement that needs to be studied further.
5. CONCLUSIONS AND RECOMMENDATIONS

A numerical study involving the modification of the vortex theory used with radial equilibrium was performed. Results have been shown in Figure 50 to Figure 53 as well as Table 3. This study aims to produce a more flow-accurate vortex model for use in designing compressor rotors.

5.1 General Conclusions

Results are consistent with the physical flow dynamics. Due to the negative value of the vortex constant, the cases with the highest value causes a decrease in the magnitude. The increased tangential velocity, related the modified vortex theory, causes the flow to separate prematurely. This variance in tangential velocity also has an effect on the pressure ratio attained.

By increasing the pressure ratio of the rotor, the surge margin is subsequently increased. The larger surge margin allows a larger operating region. The linear and parabolic variation with the positive vortex change, lower tangential velocity at the tip, increased the static margin by up to 3.32% and 1.54% respectively. The linear and parabolic variations producing a higher tangential velocity at the tip, perform worse than the baseline model decreasing the static margin by 17.94% and 15.54% respectively.

Isentropic Efficiency is calculated within the CFD program. The calculation provides accurate values away from the transition to stall; however, the calculated efficiencies are 92.60%, 94.55%, 93.99% and 91.90% for all four cases respectively. These values drop to 91.28%, 91.50%, 91.43%, and 90.34% respectively near the stall region of the baseline. At the leftmost empirical data point of the baseline rotor, the efficiency has changed to, 96.11%, 75.22%, 96.31%, and 74.97% respectively. The two cases with the efficiency lower than 80% are considered within the stall region and further data in that region.
is ignored. The 95+\% efficiency of case 1 and case 3 cause the doubt of the data collected. The efficiency calculations need to be further investigated to make a firm claim about the validity of the data collected.

5.2 Recommendations

Due to the preliminary design aspect of this paper, I have a few recommendations for future research into this topic. As a note of caution, all cases exhibit a dip in efficiency at approximately 32.25 kg/s, this as well as the location of the stall location marked in NASA-TP-2879, leads me to believe that the rotor is stalled after this point and the CFD fails to capture this in a steady state analysis.

The first recommendation is an improvement in computational power, on average each 3D CFD run of each case took approximately 12 hours on the available hardware and up to 28 hours on the standard lab computer. With each point along the characteristic line requiring multiple runs the computational time used for the 3D CFD section of this paper is upwards of 1500 hours.

The second recommendation is to investigate the effect each vortex model has along the span of the blade. The free vortex might be best in the mid-span section of the blade while the hub and tip due to losses are best modeled using a forced vortex. This paper assumed a combination of multiple vortex models along the entire span and further improvements might be attained by only focusing on key areas of the blade.

A third recommendation for future research is the effect of this vortex model on the leading edge as well as the rotor/stator interactions. The baseline model was assumed to have purely axial inlet flow thus eliminating a leading edge blade angle and the associated flow vortex. My fourth recommendation
is the increased resolution within the characteristic line. A separation of 0.5 kg/s is not a high enough resolution to fully capture the transition past the stall line.

As a final recommendation presented in this paper is the continuation of the linear and parabolic variations. Increasing and decreasing the coefficients will have an effect on the blade performance, but the exact relationship is unknown at this time. A good starting point are the cases 1 and 3 as described in this paper. These two cases showed the most promise in 3D CFD analysis with case 1 displaying an instant improvement when conducting the 2D redesign used in this paper. Cases 2 and 4 are not recommended as a starting point due to the premature transition to stall for both cases.
6. REFERENCES


### APPENDIX A – MEANLINE RESULTS

#### Table 4: Meanline Code Results at 0% Span

<table>
<thead>
<tr>
<th>Station</th>
<th>Rotor 67</th>
<th>Case 1</th>
<th>Case 2</th>
<th>Case 3</th>
<th>Case 4</th>
</tr>
</thead>
<tbody>
<tr>
<td>Axial Velocity</td>
<td>189.58</td>
<td>189.58</td>
<td>189.58</td>
<td>189.58</td>
<td>189.58</td>
</tr>
<tr>
<td></td>
<td>159.27</td>
<td>159.27</td>
<td>159.27</td>
<td>159.27</td>
<td>159.27</td>
</tr>
<tr>
<td>Absolute Velocity</td>
<td>189.58</td>
<td>313.00</td>
<td>189.58</td>
<td>276.28</td>
<td>189.58</td>
</tr>
<tr>
<td></td>
<td>351.32</td>
<td>258.73</td>
<td>276.28</td>
<td>258.73</td>
<td>370.93</td>
</tr>
<tr>
<td>Rotational Velocity</td>
<td>-162.19</td>
<td>-162.19</td>
<td>-162.19</td>
<td>-162.19</td>
<td>-162.19</td>
</tr>
<tr>
<td>Absolute Flow Angle</td>
<td>0.00</td>
<td>-59.41</td>
<td>0.00</td>
<td>-54.80</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td>-59.41</td>
<td>-63.04</td>
<td>0.00</td>
<td>-54.80</td>
<td>-64.57</td>
</tr>
<tr>
<td>Absolute Tangential Velocity</td>
<td>0.00</td>
<td>-269.45</td>
<td>0.00</td>
<td>-225.75</td>
<td>0.00</td>
</tr>
<tr>
<td></td>
<td>-269.45</td>
<td>-313.15</td>
<td>0.00</td>
<td>-225.75</td>
<td>-335.00</td>
</tr>
<tr>
<td>Relative Tangential Velocity</td>
<td>-162.19</td>
<td>-162.19</td>
<td>-162.19</td>
<td>-162.19</td>
<td>-162.19</td>
</tr>
<tr>
<td></td>
<td>77.14</td>
<td>120.84</td>
<td>33.44</td>
<td>11.59</td>
<td>142.69</td>
</tr>
<tr>
<td>Relative Flow Velocity</td>
<td>189.58</td>
<td>176.97</td>
<td>189.58</td>
<td>162.74</td>
<td>189.58</td>
</tr>
<tr>
<td></td>
<td>199.92</td>
<td>159.69</td>
<td>162.74</td>
<td>159.69</td>
<td>213.84</td>
</tr>
<tr>
<td>Relative Flow Angle</td>
<td>-40.55</td>
<td>25.84</td>
<td>-40.55</td>
<td>11.86</td>
<td>-40.55</td>
</tr>
<tr>
<td>Absolute Flow Mach</td>
<td>0.56</td>
<td>0.89</td>
<td>0.56</td>
<td>0.78</td>
<td>0.56</td>
</tr>
<tr>
<td></td>
<td>0.99</td>
<td>0.73</td>
<td>0.56</td>
<td>0.73</td>
<td>1.05</td>
</tr>
<tr>
<td>Relative Flow Mach</td>
<td>0.56</td>
<td>0.50</td>
<td>0.56</td>
<td>0.57</td>
<td>0.56</td>
</tr>
<tr>
<td></td>
<td>0.57</td>
<td>0.46</td>
<td>0.56</td>
<td>0.45</td>
<td>0.60</td>
</tr>
<tr>
<td>Vortex Constant</td>
<td>0.00</td>
<td>-30.83</td>
<td>0.00</td>
<td>-25.83</td>
<td>0.00</td>
</tr>
</tbody>
</table>

**Units:**
- Axial Velocity, Absolute Velocity, Relative Flow Velocity, and Absolute Flow Velocity: m/s
- Rotational Velocity: rad/s
- Absolute Flow Angle, Relative Flow Angle, and Vortex Constant: deg
- Relative Flow Mach: -
<table>
<thead>
<tr>
<th>Station</th>
<th>ROTOR 67</th>
<th>CASE 1</th>
<th>CASE 2</th>
<th>CASE 3</th>
<th>CASE 4</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Inlet</td>
<td>Outlet</td>
<td>Inlet</td>
<td>Outlet</td>
<td>Inlet</td>
</tr>
<tr>
<td><strong>Axial Velocity</strong></td>
<td>189.58</td>
<td>159.27</td>
<td>189.58</td>
<td>159.27</td>
<td>189.58</td>
</tr>
<tr>
<td><strong>Absolute Velocity</strong></td>
<td>189.58</td>
<td>263.54</td>
<td>189.58</td>
<td>277.30</td>
<td>189.58</td>
</tr>
<tr>
<td><strong>Absolute Flow Angle</strong></td>
<td>0.00</td>
<td>-52.82</td>
<td>0.00</td>
<td>-54.94</td>
<td>0.00</td>
</tr>
<tr>
<td><strong>Absolute Tangential Velocity</strong></td>
<td>0.00</td>
<td>-209.97</td>
<td>0.00</td>
<td>-226.99</td>
<td>0.00</td>
</tr>
<tr>
<td><strong>Relative Flow Velocity</strong></td>
<td>297.77</td>
<td>163.47</td>
<td>297.77</td>
<td>160.49</td>
<td>297.77</td>
</tr>
<tr>
<td><strong>Absolute Flow Mach</strong></td>
<td>0.56</td>
<td>0.75</td>
<td>0.56</td>
<td>0.78</td>
<td>0.56</td>
</tr>
<tr>
<td><strong>Relative Flow Mach</strong></td>
<td>0.88</td>
<td>0.46</td>
<td>0.88</td>
<td>0.45</td>
<td>0.88</td>
</tr>
<tr>
<td><strong>Vortex Constant</strong></td>
<td>0.00</td>
<td>-30.83</td>
<td>0.00</td>
<td>-33.33</td>
<td>0.00</td>
</tr>
<tr>
<td>Station</td>
<td>Rotor 67</td>
<td>Case 1</td>
<td>Case 2</td>
<td>Case 3</td>
<td>Case 4</td>
</tr>
<tr>
<td>------------------</td>
<td>----------</td>
<td>--------</td>
<td>--------</td>
<td>--------</td>
<td>--------</td>
</tr>
<tr>
<td></td>
<td>Inlet</td>
<td>Outlet</td>
<td>Inlet</td>
<td>Outlet</td>
<td>Inlet</td>
</tr>
<tr>
<td>Axial Velocity</td>
<td>189.58</td>
<td>159.27</td>
<td>189.58</td>
<td>159.27</td>
<td>189.58</td>
</tr>
<tr>
<td>Absolute Velocity</td>
<td>189.58</td>
<td>234.41</td>
<td>189.58</td>
<td>234.41</td>
<td>189.58</td>
</tr>
<tr>
<td>Rotational Velocity</td>
<td>-297.06</td>
<td>-301.26</td>
<td>-297.06</td>
<td>-301.26</td>
<td>-297.06</td>
</tr>
<tr>
<td>Absolute Flow Angle</td>
<td>0.00</td>
<td>-47.20</td>
<td>0.00</td>
<td>-47.20</td>
<td>0.00</td>
</tr>
<tr>
<td>Absolute Tangential Velocity</td>
<td>0.00</td>
<td>-172.00</td>
<td>0.00</td>
<td>-172.00</td>
<td>0.00</td>
</tr>
<tr>
<td>Relative Tangential Velocity</td>
<td>-297.06</td>
<td>-129.27</td>
<td>-297.06</td>
<td>-129.27</td>
<td>-297.06</td>
</tr>
<tr>
<td>Relative Flow Velocity</td>
<td>189.58</td>
<td>205.12</td>
<td>189.58</td>
<td>205.12</td>
<td>189.58</td>
</tr>
<tr>
<td>Absolute Flow Mach</td>
<td>0.56</td>
<td>0.66</td>
<td>0.56</td>
<td>0.66</td>
<td>0.56</td>
</tr>
<tr>
<td>Relative Flow Mach</td>
<td>0.56</td>
<td>0.58</td>
<td>0.56</td>
<td>0.58</td>
<td>0.56</td>
</tr>
<tr>
<td>Vortex Constant</td>
<td>0.00</td>
<td>-30.83</td>
<td>0.00</td>
<td>-30.83</td>
<td>0.00</td>
</tr>
</tbody>
</table>

Table 6: Meanline Code Results at 50% Span
<table>
<thead>
<tr>
<th>Station</th>
<th>ROTOR 67</th>
<th>CASE 1</th>
<th>CASE 2</th>
<th>CASE 3</th>
<th>CASE 4</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Inlet</td>
<td>Outlet</td>
<td>Inlet</td>
<td>Outlet</td>
<td>Inlet</td>
</tr>
<tr>
<td><strong>Axial Velocity</strong></td>
<td>189.58</td>
<td>159.27</td>
<td>189.58</td>
<td>159.27</td>
<td>189.58</td>
</tr>
<tr>
<td><strong>Absolute Velocity</strong></td>
<td>189.58</td>
<td>215.83</td>
<td>189.58</td>
<td>208.04</td>
<td>189.58</td>
</tr>
<tr>
<td><strong>Rotational Velocity</strong></td>
<td>-364.50</td>
<td>-355.74</td>
<td>-364.50</td>
<td>-355.74</td>
<td>-364.50</td>
</tr>
<tr>
<td><strong>Absolute Flow Angle</strong></td>
<td>0.00</td>
<td>-42.44</td>
<td>0.00</td>
<td>-40.04</td>
<td>0.00</td>
</tr>
<tr>
<td><strong>Absolute Tangential Velocity</strong></td>
<td>0.00</td>
<td>-145.66</td>
<td>0.00</td>
<td>-133.85</td>
<td>0.00</td>
</tr>
<tr>
<td><strong>Relative Tangential Velocity</strong></td>
<td>-364.50</td>
<td>-210.08</td>
<td>-364.50</td>
<td>-221.89</td>
<td>-364.50</td>
</tr>
<tr>
<td><strong>Relative Flow Velocity</strong></td>
<td>410.85</td>
<td>263.63</td>
<td>410.85</td>
<td>273.14</td>
<td>410.85</td>
</tr>
<tr>
<td><strong>Absolute Flow Mach</strong></td>
<td>0.56</td>
<td>0.61</td>
<td>0.56</td>
<td>0.59</td>
<td>0.56</td>
</tr>
<tr>
<td><strong>Relative Flow Mach</strong></td>
<td>1.21</td>
<td>0.75</td>
<td>1.21</td>
<td>0.77</td>
<td>1.21</td>
</tr>
<tr>
<td><strong>Vortex Constant</strong></td>
<td>0.00</td>
<td>-30.83</td>
<td>0.00</td>
<td>-28.33</td>
<td>0.00</td>
</tr>
<tr>
<td>Station</td>
<td>ROTOR 67</td>
<td>CASE 1</td>
<td>CASE 2</td>
<td>CASE 3</td>
<td>CASE 4</td>
</tr>
<tr>
<td>------------------</td>
<td>------------</td>
<td>------------</td>
<td>------------</td>
<td>------------</td>
<td>------------</td>
</tr>
<tr>
<td></td>
<td>Inlet</td>
<td>Outlet</td>
<td>Inlet</td>
<td>Outlet</td>
<td>Inlet</td>
</tr>
<tr>
<td>Axial Velocity</td>
<td>189.58</td>
<td>159.27</td>
<td>189.58</td>
<td>159.27</td>
<td>189.58</td>
</tr>
<tr>
<td>Absolute Velocity</td>
<td>189.58</td>
<td>203.28</td>
<td>189.58</td>
<td>191.22</td>
<td>189.58</td>
</tr>
<tr>
<td>Rotational Velocity</td>
<td>-431.94</td>
<td>-410.22</td>
<td>-431.94</td>
<td>-410.22</td>
<td>-431.94</td>
</tr>
<tr>
<td>Absolute Flow Angle</td>
<td>0.00</td>
<td>-38.42</td>
<td>0.00</td>
<td>-33.60</td>
<td>0.00</td>
</tr>
<tr>
<td>Absolute Tangential Velocity</td>
<td>0.00</td>
<td>-126.32</td>
<td>0.00</td>
<td>-105.83</td>
<td>0.00</td>
</tr>
<tr>
<td>Relative Tangential Velocity</td>
<td>-431.94</td>
<td>-283.91</td>
<td>-431.94</td>
<td>-304.39</td>
<td>-431.94</td>
</tr>
<tr>
<td>Relative Flow Velocity</td>
<td>471.71</td>
<td>325.53</td>
<td>471.71</td>
<td>343.54</td>
<td>471.71</td>
</tr>
<tr>
<td>Absolute Flow Mach</td>
<td>0.56</td>
<td>0.57</td>
<td>0.56</td>
<td>0.54</td>
<td>0.56</td>
</tr>
<tr>
<td>Relative Flow Mach</td>
<td>1.39</td>
<td>0.92</td>
<td>1.39</td>
<td>0.97</td>
<td>1.39</td>
</tr>
<tr>
<td>Vortex Constant</td>
<td>0.00</td>
<td>-30.83</td>
<td>0.00</td>
<td>-25.83</td>
<td>0.00</td>
</tr>
</tbody>
</table>
Figure 54: Case 1 Mesh at 0% Span
Figure 55: Case 1 Mesh at 25% Span
Figure 56: Case 1 Mesh at 75% Span
Figure 57: Case 1 Mesh at 100% Span
Figure 58: Case 2 Mesh at 0% Span
Figure 59: Case 2 Mesh at 25% Span
Figure 60: Case 2 Mesh at 75% Span
Figure 61: Case 2 Mesh at 100% Span
Figure 62: Case 3 Mesh at 0% Span
Figure 63: Case 3 Mesh at 25% Span
Figure 64: Case 3 Mesh at 75% Span
Figure 65: Case 3 Mesh at 100% Span
Figure 66: Case 4 Mesh at 0% Span
Figure 67: Case 4 Mesh at 25% Span
Figure 68: Case 4 Mesh at 75% Span
Figure 69: Case 4 Mesh at 100% Span
Figure 70: Rotor 67 Mesh at 0% Span
Figure 71: Rotor 67 Mesh at 25% Span
Figure 72: Rotor 67 Mesh at 50% Span
Figure 73: Rotor 67 Mesh at 75% Span
Figure 74: Rotor 67 Mesh at 100% Span
Figure 75: Case 1 Density Contour at 0% Span
Figure 76: Case 1 Density Contour at 25% Span
Figure 77: Case 1 Density Contour at 75% Span
Figure 78: Case 1 Density Contour at 100% Span
Figure 79: Case 2 Density Contour at 0% Span
Figure 80: Case 2 Density Contour at 25% Span
Figure 81: Case 2 Density Contour at 75% Span
Figure 82: Case 2 Density Contour at 100% Span
Figure 83: Case 3 Density Contour at 0% Span
Figure 84: Case 3 Density Contour at 25% Span
Figure 85: Case 3 Density Contour at 75% Span
Figure 86: Case 3 Density Contour at 100% Span
Figure 87: Case 4 Density Contour at 0% Span
Figure 88: Case 4 Density Contour at 25% Span
Figure 89: Case 4 Density Contour at 75% Span
Figure 90: Case 4 Density Contour at 100% Span
Figure 91: Rotor 67 Density Contour at 0% Span
Figure 92: Rotor 67 Density Contour at 25% Span
Figure 93: Rotor 67 Density Contour at 50% Span
Figure 94: Rotor 67 Density Contour at 75% Span
Figure 95: Rotor 67 Density Contour at 100% Span