Numerical Study Of A High-speed Miniature Centrifugal Compressor

Xiaoyi Li
University of Central Florida

Part of the Mechanical Engineering Commons

Find similar works at: https://stars.library.ucf.edu/etd

This Doctoral Dissertation (Open Access) is brought to you for free and open access by STARS. It has been accepted for inclusion in Electronic Theses and Dissertations, 2004-2019 by an authorized administrator of STARS. For more information, please contact STARS@ucf.edu.

STARS Citation
https://stars.library.ucf.edu/etd/464
NUMERICAL STUDY OF A HIGH-SPEED MINIATURE CENTRIFUGAL COMPRESSOR

by

XIAOYI LI
M.S. University of New South Wales, 1998

A dissertation submitted in partial fulfillment of the requirements for the degree of Doctor of Philosophy in the Department of Mechanical, Materials and Aerospace Department in the College of Engineering & Computer Science at the University of Central Florida Orlando, Florida

Summer Term
2005

Major Professors: Dr. Jayanta Kapat and Dr. Louis C. Chow
A miniature centrifugal compressor is a key component of a reverse Brayton cycle cryogenic cooling system. The system is commonly used to generate a low cryogenic temperature environment for electronics to increase their efficiency, or generate, store and transport cryogenic liquids, such as liquid hydrogen and oxygen, where space limit is also an issue.

Because of space limitation, the compressor is composed of a radial inlet guide vane, a radial impeller and an axial-direction diffuser (which reduces the radial size because of smaller diameter). As a result of reduction in size, in order to obtain the required static pressure ratio/rise, the rotating speed of the impeller is as high as 313 KRPM, if Helium is used as the working fluid. Two main characteristics of the compressor – miniature and high-speed, make it distinct from conventional compressors.

Higher compressor efficiency is required to obtain a higher COP (coefficient of performance) system. Even though miniature centrifugal compressors start to draw researchers’ attention in recent years, understanding of the performance and loss mechanism is still lacking. Since current experimental techniques are not advanced enough to capture details of flow at miniature scale, numerical methods dominate miniature turbomachinery study.

This work numerically studied a high speed miniature centrifugal compressor. The length and diameter are 7 cm and 6 cm, respectively. The study was done on the same physical compressor but with three different combinations of working fluid and operating speed combinations: air and 108 KRPM, helium and 313 KRPM, and neon and 141
KRPM. The overall performance of the compressor was predicted with consideration of interaction between blade rows by using a sliding mesh model. It was found that the specific heat ratio needs to be considered when similarity law is applied. But Reynolds number effect can be neglected. The maximum efficiency observed without any tip leakage was 70.2% for air 64.8% for helium 64.9% for neon.

The loss mechanism of each component was analyzed. Loss due to turning bend was found to be significant in each component, even up to 30%. Tip leakage loss of small scale turbomachines has more impact on the impeller performance than that of large scale ones. Use of 10% tip gap was found to reduce impeller efficiency from 99% to 90%. Because the splitter was located downstream of the impeller leading edge, any incidence at the impeller leading edge leads to poorer splitter performance. Therefore, the impeller with twenty blades had higher isentropic efficiency than the impeller with ten blades and ten splitters. Based on numerical study, a four-row vaned diffuser was used to replace a two-row vaned diffuser. It was found that the four-row vaned diffuser had much higher pressure recovery coefficient than the two-row vaned diffuser. However, most of pressure is found to be recovered at the first two rows of diffuser vanes.

Consequently, the following suggestions were given to further improve the performance of the miniature centrifugal compressor.

1. Redesign inlet guide vane based on the numerical simulation and experimental results.
2. Add de-swirl vanes in front of the diffuser and before the bend.
3. Replace the current impeller with a twenty-blade impeller.
4. Remove the last row of diffuser.
ACKNOWLEDGMENTS

I would like to thank my advisors Dr. Jay Kapat and Dr. Louis C. Chow. Without their encouragement, patience, care, and advice, I would not have completed this dissertation.

Other members of my dissertation committee, Dr. Quanfang Chen, Dr. Roy S. Choudhury, Dr. Alain Kassab and Dr. Dan Rini have generously given their time and expertise to better my work. I thank them for their contribution and their good-natured support.

I must acknowledge my parents, who are taking care of my lovely daughter. Without their support, I would not be able to concentrate on my work.

I am grateful to my sister and brother-in-law, who support me in my hard time.
# TABLE OF CONTENTS

LIST OF FIGURES ........................................................................................................... ix

LIST OF TABLES ............................................................................................................... xii

LIST OF NOMENCLATURE ............................................................................................. xiii

CHAPTER ONE: INTRODUCTION ...................................................................................... 1

1.1 Centrifugal compressor background and development ............................................. 1

1.2 High speed miniature centrifugal compressor .......................................................... 4

1.3 Scope of this work .................................................................................................... 7

CHAPTER TWO: LITERATURE REVIEW ........................................................................ 9

2.1 CFD & turbomachinery ............................................................................................ 9

2.2 Study of interaction between blade rows ............................................................... 13

2.3 Tip leakage effect .................................................................................................... 22

2.4 Effect of size on the performance .......................................................................... 23

CHAPTER THREE: NUMERICAL PROCEDURE .......................................................... 25

3.1 FLUENT and turbomachinery applications ............................................................ 25

3.2 Turbulent modeling ............................................................................................... 27

3.2.1 Realizable $k-\varepsilon$ model .............................................................................. 28

3.2.2 Realizable $k-\varepsilon$ model transport equations ............................................. 29

3.2.3 Modeling the turbulent viscosity ................................................................... 30

3.2.4 Model constants ............................................................................................... 30

3.3 Near-Wall treatment ............................................................................................. 31

3.3.1 Standard wall function .................................................................................. 32
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.3.2 Non-equilibrium wall functions</td>
<td>35</td>
</tr>
<tr>
<td>3.3.3 Near-wall modeling</td>
<td>37</td>
</tr>
<tr>
<td>3.4 Computation with rotating elements</td>
<td>38</td>
</tr>
<tr>
<td>3.5 Settings</td>
<td>41</td>
</tr>
<tr>
<td>3.5.1 Boundary conditions</td>
<td>41</td>
</tr>
<tr>
<td>3.5.2 Control of solution: under-relaxation</td>
<td>42</td>
</tr>
<tr>
<td>3.5.3 Control of solution: discretization</td>
<td>42</td>
</tr>
<tr>
<td>3.6 Grid generation</td>
<td>43</td>
</tr>
<tr>
<td>3.6.1 Structured grid</td>
<td>43</td>
</tr>
<tr>
<td>3.6.2 Unstructured grid</td>
<td>46</td>
</tr>
<tr>
<td>CHAPTER FOUR: RESULTS AND DISCUSSION</td>
<td>49</td>
</tr>
<tr>
<td>4.1 Inlet guide vane</td>
<td>49</td>
</tr>
<tr>
<td>4.1.1 IGV loss analysis</td>
<td>52</td>
</tr>
<tr>
<td>4.1.2 Exit flow angle</td>
<td>61</td>
</tr>
<tr>
<td>4.1.3 Interaction with Impeller</td>
<td>65</td>
</tr>
<tr>
<td>4.1.4 Summary</td>
<td>65</td>
</tr>
<tr>
<td>4.2 Impeller</td>
<td>65</td>
</tr>
<tr>
<td>4.2.1 Impeller Inlet</td>
<td>66</td>
</tr>
<tr>
<td>4.2.2 Inside of the impeller</td>
<td>74</td>
</tr>
<tr>
<td>4.2.3 Impeller exit</td>
<td>78</td>
</tr>
<tr>
<td>4.2.4 Tip leakage and secondary flow</td>
<td>81</td>
</tr>
<tr>
<td>4.2.5 Loss analysis</td>
<td>87</td>
</tr>
<tr>
<td>4.2.6 Summary</td>
<td>88</td>
</tr>
</tbody>
</table>
4.3 Diffuser ................................................................................................................... 89

4.3.1 Diffuser performance ....................................................................................... 89

4.3.2 Summary ........................................................................................................ 100

4.4 Overall Performance of the compressor ............................................................... 101

4.4.1 Performance of the compressor ..................................................................... 101

4.4.2 Dimensional analysis ..................................................................................... 102

CHAPTER FIVE: CONCLUSION................................................................................. 107

APPENDIX: AIR, NEON, AND HELIUM PROPERTIES........................................... 109

LIST OF REFERENCES................................................................................................ 110
<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Centrifugal compressor with a radial diffuser (Japikse, 1996)</td>
<td>2</td>
</tr>
<tr>
<td>2</td>
<td>Miniature centrifugal compressor (Pro/E drawing)</td>
<td>6</td>
</tr>
<tr>
<td>3</td>
<td>Miniature centrifugal compressor (Cross-section view)</td>
<td>6</td>
</tr>
<tr>
<td>4</td>
<td>Miniature centrifugal compressor (Disassembled)</td>
<td>7</td>
</tr>
<tr>
<td>5</td>
<td>HVAC fan simulation with FLUENT</td>
<td>26</td>
</tr>
<tr>
<td>6</td>
<td>Centrifugal pump simulation with FLUENT</td>
<td>26</td>
</tr>
<tr>
<td>7</td>
<td>Sliding mesh model theory</td>
<td>40</td>
</tr>
<tr>
<td>8</td>
<td>Sliding mesh model time-periodic solution</td>
<td>41</td>
</tr>
<tr>
<td>9</td>
<td>Structured grid (tip leakage study)</td>
<td>45</td>
</tr>
<tr>
<td>10</td>
<td>Structured grid (impeller only)</td>
<td>46</td>
</tr>
<tr>
<td>11</td>
<td>Geometry for unstructured grid</td>
<td>48</td>
</tr>
<tr>
<td>12</td>
<td>Unstructured grid</td>
<td>48</td>
</tr>
<tr>
<td>13</td>
<td>IGV (vaned) + impeller (bladed) + diffuser (vaned)</td>
<td>51</td>
</tr>
<tr>
<td>14</td>
<td>IGV (vaned) +impeller (without blade)</td>
<td>51</td>
</tr>
<tr>
<td>15</td>
<td>IGV (without vane) + impeller (without blade)</td>
<td>52</td>
</tr>
<tr>
<td>16</td>
<td>Contour of total pressure (IGV)</td>
<td>53</td>
</tr>
<tr>
<td>17</td>
<td>Velocity vector (IGV)</td>
<td>54</td>
</tr>
<tr>
<td>18</td>
<td>Total pressure (IGV)</td>
<td>55</td>
</tr>
<tr>
<td>19</td>
<td>Flow separation at the turning bend</td>
<td>56</td>
</tr>
<tr>
<td>20</td>
<td>Total pressure (from simulation iii)</td>
<td>58</td>
</tr>
<tr>
<td>21</td>
<td>Meridional velocity (from simulation iii)</td>
<td>59</td>
</tr>
</tbody>
</table>
Figure 22 Contour of tangential velocity (IGV exit) ........................................................ 62
Figure 23 Contour of meridional velocity (IGV exit) ....................................................... 63
Figure 24 Definition of flow angle ................................................................................... 63
Figure 25 Flow angle at the IGV exit ............................................................................... 64
Figure 26 Comparison of current impeller to conventional impeller ............................... 67
Figure 27 Relative flow angle (Impeller inlet) (velocity units: m/s) ................................. 68
Figure 28 Relative total Pressure (Impeller eye) ............................................................... 69
Figure 29 Relative tangential velocity (Impeller eye) ...................................................... 70
Figure 30 Velocity triangle (impeller-splitter, uniform inlet, zero flow angle) ............... 71
Figure 31 Relative total pressure at different meridional locations (impeller-splitter,
    uniform flow, zero flow angle) .................................................................................... 73
Figure 32 Relative total pressure (Impeller) ................................................................... 75
Figure 33 Mass (Impeller) ............................................................................................. 76
Figure 34 Relative velocity vector (Colored by relative tangential velocity) ................... 77
Figure 35 Relative tangential velocity (Impeller) ........................................................... 78
Figure 36 Velocity vector at the impeller exit ................................................................. 80
Figure 37 Mass-averaged static pressure in impeller ....................................................... 80
Figure 38 Relative total pressure (with tip leakage) ....................................................... 83
Figure 39 Relative total pressure (without tip leakage) .................................................... 84
Figure 40 Total pressure (Friction study) ....................................................................... 87
Figure 41 Comparison of two diffuser designs ............................................................... 91
Figure 42 Tangential velocity (impeller exit and diffuser inlet) ....................................... 93
Figure 43 Tangential velocity (diffuser inlet) ................................................................. 93
Figure 44 Mass-averaged static pressure (diffuser) .......................................................... 94
Figure 45 Mass-averaged total pressure (diffuser) ........................................................... 94
Figure 46 Velocity magnitude with time (impeller exit to diffuser inlet) ....................... 98
Figure 47 Mass-averaged static pressure (1st and 2nd rows of diffuser vanes) .......... 99
Figure 48 Performance chart (air) ................................................................................ 101
Figure 49 Dimensional analysis .................................................................................. 103
Figure 50 Velocity contour (helium 313K corrected) .................................................. 105
Figure 51 Velocity contour (air 108K) ........................................................................ 105
Figure 52 Comparison of Reynolds number ............................................................... 106
LIST OF TABLES

Table 1 Design parameters of the compressor ............................................................... 5
Table 2 Studies of interaction in centrifugal compressors ............................................. 18
Table 3 IGV loss analysis ............................................................................................ 60
Table 4 Isentropic efficiency in meridional direction (with and without tip leakage)..... 85
Table 5 Comparison of compressor performance with two diffusers ........................... 92
LIST OF NOMENCLATURE

\( A \) Area

\( C_m \) Meridional velocity

\( C_{p,\text{loss,IGV}} \) IGV pressure loss coefficient

\( C_{p,\text{loss,impeller}} \) Impeller pressure loss coefficient

\( C_{p,\text{loss,diffuser}} \) Diffuser pressure loss coefficient

\( \theta \) Tangential velocity

\( C_{\theta,\text{rel}} \) Relative tangential velocity of flow

\( C_{\theta,i} \) Tangential velocity at cell

\( \mu \) Constant

\( D \) Diameter of impeller

\( G_k \) Generation of turbulence kinetic energy due to the mean velocity gradient

\( G_b \) Generation of turbulence kinetic energy due to buoyancy

\( m \) Mass flow rate

\( N \) Rotating speed

\( k \) Turbulent kinetic energy

\( P_{00} \) Total pressure at compressor inlet

\( P_{03} \) Total pressure at compressor exit
\( Pr \)  
Total pressure ratio between inlet and outlet

\( P_t \)  
Total pressure

\( P_{rt} \)  
Total pressure ratio

\( P_s \)  
Static pressure

\( P_{ri} \)  
Total pressure at inlet

\( P_{re} \)  
Total pressure at exit

\( P_{si} \)  
Static pressure at inlet

\( P_{se} \)  
Static pressure at exit

\( P_{s,3} \)  
Static pressure at diffuser exit

\( \overline{P} \)  
Power coefficient

\( R \)  
Gas constant

\( R_i \)  
Radius of interface

\( \text{Re} \)  
Reynolds number

\( r \)  
Radius

\( r_i \)  
Radius at cell

\( S \)  
Source term

\( S_k, S_c \)  
User defined source term

\( \Delta S \)  
Size of grid at interface

\( \Delta \overline{S} \)  
Displacement vector from the cell centroid

\( T_{t0} \)  
Total temperature at compressor inlet
\( T_{03} \) \hspace{1cm} \text{Total temperature at compressor exit}

\( T_t \) \hspace{1cm} \text{Total temperature}

\( T_s \) \hspace{1cm} \text{Static temperature}

\( Tr \) \hspace{1cm} \text{Total temperature ratio between inlet and outlet}

\( \Delta t \) \hspace{1cm} \text{Time step}

\( u \) \hspace{1cm} \text{Velocity component of flow}

\( \bar{u} \) \hspace{1cm} \text{Mean velocity components}

\( u' \) \hspace{1cm} \text{Fluctuating velocity components}

\( U \) \hspace{1cm} \text{Blade velocity}

\( x \) \hspace{1cm} \text{Coordinate component}

\( x/X \) \hspace{1cm} \text{Meridional distance from inlet over from inlet to outlet}

\( Y_M \) \hspace{1cm} \text{Contribution of the fluctuation dilatation in compressible turbulence to the overall dissipation rate}

\( y^*, y^+ \) \hspace{1cm} \text{Non-dimensional distance from wall}

\( \beta \) \hspace{1cm} \text{Thermal expansion coefficient}

\( \varepsilon \) \hspace{1cm} \text{Turbulent energy dissipation rate}

\( \eta_{isen} \) \hspace{1cm} \text{Isentropic efficiency}

\( \eta_{imp} \) \hspace{1cm} \text{Isentropic efficiency of impeller}

\( \eta_c \) \hspace{1cm} \text{Compressor efficiency}

\( \eta \) \hspace{1cm} \text{Constant}

\( \rho_{00} \) \hspace{1cm} \text{Density at compressor inlet}
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\gamma$</td>
<td>Specific heat ratio</td>
</tr>
<tr>
<td>$\omega$</td>
<td>Rotating speed</td>
</tr>
<tr>
<td>$\mu_t$</td>
<td>Turbulent viscosity</td>
</tr>
<tr>
<td>$\phi$</td>
<td>Cell-center value of scalar quantities</td>
</tr>
<tr>
<td>$\phi_f$</td>
<td>Face value of scalar quantities</td>
</tr>
<tr>
<td>$\nabla \phi$</td>
<td>Gradient of scalar quantities</td>
</tr>
</tbody>
</table>
CHAPTER ONE: INTRODUCTION

1.1 Centrifugal compressor background and development

Dixon (1998) defined a turbomachine as a rotating blade row. An impeller, also called a rotor, changes the stagnation enthalpy of the fluid moving through it by adding or extracting angular momentum, depending on the effect required of the machine. These enthalpy changes are intimately linked with the total pressure changes occurring in the fluid.

A centrifugal compressor is a type of turbomachine through which work is added to gases by a radial-flow impeller, as shown in Figure 1. Due to the nature of the radial geometry, the cross-sectional area of the impeller increases along the flow direction. Therefore, the impeller also serves as a rotating diffuser, which makes flow through a centrifugal compressor even more complex. Sometimes, static pressure is further recovered from the velocity in a vaneless or vaned diffuser. The diffuser can be in the radial or axial direction.

Compared to a centrifugal compressor, an axial-flow compressor has a higher efficiency with the same duty, and smaller frontal area and drag. However, the efficiency of an axial compressor drops sharply at very low air mass flow rates, thus the blading requiring to be small which is difficult to accurately manufacture. It is advantageous to use centrifugal compressors at low mass flow rates.
According to Krain’s (2005) review of centrifugal compressors, the first documented turbomachinery application dates back to about 60 A.D. Heron of Alexandria designed the first radial-flow reaction steam turbine. Around 1750, Leonhard and Albert Euler analyzed Heron’s steam turbine and did several basic experiments which resulted in 1754’s “Newton’s Law on Turbomachinery,” also simply called “Euler’s Equation,” $W = \Delta (UC_\theta)$.

A widespread industrial application of centrifugal compressors evolved at the end of the 19th century when industry began to require a lot of pressurized air. Later on, centrifugal compressors were used for a variety of technical applications: in metallurgy,
chemical and petrochemical engineering and refineries, in the gas and pipeline industry, for refrigeration and in armored vehicles. Strong impetus for further development of centrifugal compressors came from the introduction of aircraft propulsion. Frank Whittle (England) and Hans Joachim Pabst von Ohain (Germany) developed almost independently of one another the world’s first jet engine (1928-1941). Both pioneers used the same principle of air compression: the centrifugal compressor. Frank Whittle already aimed for stage pressure ratios of about 4:1, which was beyond all previous engineering experience. Centrifugal compressors still find widespread applications in the aviation field for small aircraft engines, most significantly for helicopters, where the performance is critically dependent on the engine’s weight plus the fuel.

During the last five decades, centrifugal compressor stage pressure ratios were boosted more and more so that single stage compressors with pressure ratios larger than 8:1 came into practical use. Building compressors with higher efficiencies is in strong demand. Opportunities for further efficiency improvements lay with carefully controlled diffusion in rotors and diffusers at rather high Mach number levels. Consequently, there is a lot of interest to investigate the internal flow field of centrifugal compressors at real operating conditions, and to clarify the problem of separation onset, jet/wake development and secondary flow. To study high speed compressors, numerical simulation comes into the picture.

Most recently, small compressors have begun to draw researchers’ attention. Miniature and micro-turbomachines are built for various applications.
1.2 High speed miniature centrifugal compressor

As common fuel for spacecraft and rockets, liquid hydrogen and oxygen need to be generated, stored and transported at cryogenic temperatures. Cooling to cryogenic temperatures (less than 120K) is also necessary for many of the electronic sensors and systems used for various military, civil, scientific, and medical purposes. Some of this equipment works better, faster and more effectively at low temperatures. For other applications, cryogenic cooling is essential to utilize phenomena, which exist at very low temperatures, sometimes within a narrow or precise temperature range. Superconducting devices fall in this category. Linde-Hampson cryocoolers and stirling cryocoolers are conventional type cryocoolers. Recently, reverse Brayton cycle cryogenic cooling systems came into existence. Compared to convectional cryocoolers, the Brayton cycle cryogenic cooler has the advantages of simplicity, reliability, low vibration, and higher thermal efficiency.

A miniature high efficiency cryogenic cooling system was designed based on the Brayton cycle, detailed in Zhou (2003). The system is slightly longer and slimmer than a Coca-Cola can. It has the ability to remove 20 W of heat from a 70K source to a 320K source, with a proposed coefficient of performance (COP) of around 0.1.

High compressor efficiency is desirable to reach a high COP of the cooling system. The preliminary design of the compressor was based on a one-dimensional analysis. The compressor is air-based instead of helium-based for test purposes. (See Appendix for air and helium properties).
Some parameters of the compressor are listed in Table 1.

Table 1 Design parameters of the compressor

<table>
<thead>
<tr>
<th></th>
<th>Neon</th>
<th>Air</th>
<th>Helium</th>
</tr>
</thead>
<tbody>
<tr>
<td>( m ) (g/s)</td>
<td>6.67</td>
<td>7.33</td>
<td>1.62</td>
</tr>
<tr>
<td>( Pr\pi )</td>
<td>1.7</td>
<td>1.551</td>
<td>1.7</td>
</tr>
<tr>
<td>( N ) (RPM)</td>
<td>141000</td>
<td>108000</td>
<td>313000</td>
</tr>
<tr>
<td>( T_0 ) (K)</td>
<td>300</td>
<td>300</td>
<td>300</td>
</tr>
<tr>
<td>( R ) (J/kg*K)</td>
<td>416</td>
<td>287</td>
<td>2079</td>
</tr>
<tr>
<td>( \gamma )</td>
<td>1.67</td>
<td>1.4</td>
<td>1.67</td>
</tr>
<tr>
<td>( \rho_0 ) (kg/m³)</td>
<td>0.8086</td>
<td>1.16</td>
<td>0.08</td>
</tr>
<tr>
<td>( \overline{P} ) (W)</td>
<td>672</td>
<td>424</td>
<td>198</td>
</tr>
</tbody>
</table>

As shown in Figure 2, Figure 3 and Figure 4, the compressor is composed of Inlet Guide Vanes (IGV-preswirl), an impeller (radial) and a compact diffuser (axial). The diameter is about 6 cm and the blade height at the rotor exit is 2 mm to achieve the correct mass flow rate.
Figure 2 Miniature centrifugal compressor (Pro/E drawing)

Figure 3 Miniature centrifugal compressor (Cross-section view)
An axial compact vaned diffuser, instead of a conventional radial diffuser, is applied to reduce the outer radius of the compressor. However, due to the lack of research on this kind of geometrical arrangement, it is difficult to match the angles between the impeller and diffuser and to understand the loss mechanism.

1.3 Scope of this work

Analytical and empirical design rules that have been employed in obtaining the preliminary design cannot predict the performance exactly, nor can they be used for design optimization. To provide the same enthalpy increase, miniature compressors rotate much faster than large scale compressors.
Therefore, the miniature compressor is different from conventional compressors by its size, unique geometry and high rotating speed. For a small scale with such a high rotating speed, the experiment measurement is limited by the current available techniques. Numerical methods are the only way to study the details of the flow in a miniature centrifugal compressor. Also, computational prediction of performance is much cheaper and less time consuming compared to direct experimental verification.

This work concentrates on comprehensive numerical analysis of interactions between blade rows, and between the tip leakage flow and the main flow inside of the miniature high-speed centrifugal compressor introduced above. Studies of the fluid flow field of the compressor during design and off-design conditions were carried out. The numerical simulations also predicted the performance of the centrifugal compressor with different working fluids to validate dimensional analysis in a small scale. The diffuser design was modified based on the preliminary CFD simulation. The interaction was studied with two designs of the diffuser. Eventually, a loss analysis was carried out for each component.
CHAPTER TWO: LITERATURE REVIEW

2.1 CFD & turbomachinery

Computational Fluid Dynamics (CFD) is the science of predicting fluid flow, heat and mass transfer, chemical reactions, and related phenomena by numerically solving the set of governing mathematical equations, such as conservation of mass, momentum, energy, species, and so on. CFD analysis complements testing and experiments. It reduces the total effort required in the experiment design and data acquisition.

For many years the design of modern turbomachines has relied heavily on CFD to develop three-dimensional blade sections. The benefits of CFD range from shorter design cycles to better performance and reduced costs and weight. Also, details of aerodynamic data that are now accessible during the design process and increased compressor performance have without a doubt been made possible. This section introduces the history and development of CFD in turbomachinery applications.

The application of numerical methods to turbomachinery dates back to the 1940s, even before the advent of digital computers. In the early years, compressor design substantially depended on empirical correlations of data. Data came from cascade tests of blades with prescribed profiles coupled with some simple analytical methods. The mean line design of axial compressors relied on cascade data for flow deviation and for limiting deflections. There was a limited understanding of the physics of the flow.
Analytical research began to have a greater impact on this largely practical design approach towards the end of the 1950s and the beginning of the 1960s. There were four particular areas of interest: blade to blade flows, secondary flow, through-flow methods, and clearance flow. The analytical/empirical methods from the period of 1950-1970 can be looked back upon in light of the powerful computational methods now available. Horlock and Denton (2005) listed four examples: flow deviation, annulus wall blockage, tip clearance flows, and secondary flows. Interested readers who want to know about the details of these analytical methods can refer to their paper.

Till the early 1980s when fully three-dimensional (3D) methods first became available, the definition of blade-to-blade (S1) and hub-to-tip (S2) stream surfaces dominated the subject (quasi-three-dimensional Q3D). This method was first introduced by Wu (1951). The axisymmetric hub-to-tip (S2) calculation is often called the “throughflow calculation” and has become the backbone of turbomachinery design, while the blade-to-blade (S1) calculation remains the basis for defining the detailed blade shape. Q3D flow calculations predict flow on the stream surface using the assumption that the surface is a surface of revolution.

For turbomachinery, many phenomena can be studied and predicted by two-dimensional (2D) or quasi-three-dimensional (Q3D). However, some aspects of the flow must be simulated by three-dimensional (3D), and cannot be understood by the Q3D method. For instance, secondary flow, blade sweep, blade lean and interaction of blade rows are fully 3D effects. The 3D method replaces the stream surface calculations by a single calculation for all the blade rows. This removes the modeling assumptions of the quasi-three-dimensional (Q3D) approach but requires far greater computer power and so
was not usable as a routine design tool until the late 1980s (Denton and Dawes, 1999). Early methods had to use coarser grids that introduced larger numerical errors than in the Q3D approach. As computer technology has been rapidly developing in recent years, CFD simulations which involve a large quantity of grid points have become feasible. For instance, 12 blade rows, with 70,000 grid points used in each row, can be calculated on a modern workstation in run times on the order of 24 hours.

What CFD really provides is the ability to exploit the three-dimensional nature of the flow to control undesirable features such as corner separations in compressors. Since loss predictions by CFD are still not accurate, interpretation of the results requires considerable skill and experience. Good physical understanding and judgment of when the flow has been improved remain very important.

Some successful uses of CFD to improve designs include the use of bowed blades to control secondary loss in turbines and the use of sweep and bow to reduce corner separation in compressors; both of these techniques are now routinely used in production machines.

The current trend is to move to multistage and unsteady predictions, both of which require increased computer power. The unsteady interaction between blade rows can be modeled by steady multistage calculation using mixing planes or deterministic stresses. These models are approximations and their limitations need to be explored by detailed flow measurements on multistage machines.

Unsteady calculations are very time consuming to be used for routine design, but are good for research investigations. Their results can be used to obtain the unsteady blade loading and hence to assess the mechanical limitations of blading. The implications of
unsteady flow on the loss generation are just starting to be explored. Questions such as whether the entropy generation is more or less when wakes and vortices mix out unsteadily in a downstream blade row, rather than in a steady flow, are starting to be answered by CFD rather than by experiment.

The interaction of leakage flows with the main gas path flow has usually been neglected in the past, but its importance is starting to be realized. Such flows can interact with the main stream flow to produce shear layers which contain vorticity and hence generate additional secondary flows and losses. These flows are highly turbulent. Experimental measurement of these flows is difficult because their length scale is often very small. CFD prediction is very dependent on the turbulence modeling in the codes.

Regarding the practice of relating CFD to various important aerodynamic criteria in turbomachinery, Casey (2003) concluded eight points: avoid poor incidence onto blading, reduce friction on surfaces, avoid kinetic energy loss, avoid flow separation, provide a uniform distribution onto downstream blades, examine classical blade loading criteria, and minimize the pre-shock Mach number.

Some critical issues related to the current state of turbomachinery CFD, such as validation and quality assurance requirement, were addressed by Hirsch (2003). Comparing structured and unstructured CFD technology, he pointed out that structured-grid methods take full advantage of high quality stretched grids for viscous flows at high Reynolds numbers. However, the requirements for geometries beyond simple blade passages lead to high costs in engineering time for the structured meshing process, where unstructured grid generators can generate complex grids nearly automatically and in a cost effective way, with short turnaround times. Many sources of uncertainty are still
present in turbomachinery validation, in addition to the numerical error sources. Some of these possible sources are:

- Geometrical uncertainties, such as tip clearance gap height, errors in the periodic blade settings, influence of fillets, hub/shroud gaps.
- Secondary flow effects, connected to cavities, labyrinths, etc…

Uncertainties in the inlet flow conditions, where small errors, on swirl for instance, could have a severe effect on CFD predictions.

### 2.2 Study of interaction between blade rows

As discussed in the previous section, small errors on swirl of inlet flow conditions could have a severe effect on CFD predictions. Therefore, it is necessary to consider the interactions in order to capture inlet conditions more accurately for the downstream components.

Before discussing this interaction, it is helpful to have a brief explanation of “stall”. Whenever a diffusing (decelerating) flow is found along a physical surface, the possibility exists for the flow to be retarded so severely that it can no longer follow the surface. The streamlines adjacent to the wall will leave the wall and a region of reverse flow will develop from that point along the wall surface. In other words, the momentum in the streamline adjacent to the wall is insufficient to overcome the adverse pressure gradient and the viscous shear stresses along the wall. Then the flow is forced to deviate from the surface, and is said to be stalled. The terms stall and separation are virtually interchangeable.
A rotating stall can initiate some unsteady flow patterns, such as surge, periodic pressure and flow variations. Unsteadiness of the flow not only results in a deterioration of compressor performance, but also constitutes a source of mechanical excitation of the blades and shaft, which may limit the range of operation because of vibration problems.

The unsteady interaction between the impeller and diffuser can generate a rotating stall (Japikse, 1996). The relative motion of the stationary and rotating components in a turbomachine will give rise to unsteady interactions. These interactions are generally classified as follows:

- Potential interactions- flow unsteadiness due to pressure waves which propagate both upstream and downstream.
- Wake interactions- flow unsteadiness due to wakes from upstream blade rows convecting downstream.
- Shock interactions- for transonic/supersonic flows, unsteadiness due to shocks waves striking downstream blade row.

Shock interactions are not included in this study, since the compressor is mainly operated at subsonic range.

Potential effects propagate upstream and downstream. Potential effects are geometry driven pressure unsteadiness due to the relative motion of the blades. A simple demonstration is where the pressure field can be approximately decomposed into a steady uniform part, a non-uniform part that is steady in the stator frame and a non-uniform part that is steady in the rotor frame. The motion of the non-uniform pressure field, that is steady in the rotor frame, causes the potential unsteady effects in the stator. This unsteady pressure observed in the stator is like an unsteady variation recorded by a pressure probe.
moving uniformly across a non-uniform, steady, pressure field generated by an isolated airfoil. Similarly, the potential effects observed in the rotor are due to the non-uniform pressure field that is steady in the stator frame.

Unsteady effects generated by wake/blade interactions are due to the slicing in pieces of the wakes, issued from the front blades, by the downstream blade row. Wakes are created by viscosity. Flow leaving a rotor is fully three-dimensional and presents non-uniformities between the hub and the shroud as well as in the circumferential direction. These non-uniformities result from the secondary flow that developed in the rotor. Under the action of the centrifugal and Coriolis forces, low momentum fluid accumulated in the shroud-suction side corner. This migration of the low momentum fluid leads to the so-called jet-wake model at the rotor exit.

Study of interaction is crucial to the optimization of diffuser design. Energy transfer in turbomachinery involves the exchange of significant levels of kinetic energy in order to accomplish the intended purpose. As a consequence, very large levels of residual kinetic energy frequently accompany the work input and work extraction process, sometimes as much as 50% of the total energy transferred. With kinetic energies of this magnitude, it is not hard to appreciate that the performance of the diffuser directly and often strongly influences the overall efficiency of the turbomachine. A change of 0.01 in pressure recovery can be equivalent to a few tenths of a point of stage efficiency or can be as much as two to four points of stage efficiency (Japikse, 1998). Distortion in the inlet parameters of the diffuser, such as velocity distribution or total pressure distribution, may be expected to have a significant effect on diffuser performance, which strongly relies on the outflow from the impeller.
The blade row interaction has drawn more research attention in recent years. Studies of this interaction were carried out numerically and experimentally.

Kirtley and Beach (1992) numerically studied blade row interactions in a low-speed centrifugal compressor stage using average passage Navier-Stokes equations, which were derived by Adamczyk (1985). The isolated impeller calculation shows a jet/wake discharge flow, with the wake confined to the shroud near the pressure side. This wake flow is coincident with the tip leakage vortex. In a stage calculation, the impeller flow is found to be similar to the isolated impeller, except for the reduction of shroud separation due to the additional blockage that the vane provides. The large secondary flows generated in the impeller give rise to deviations in the exit flow angle, which cause the vane to operate off design incidence.

The unsteady aerodynamics response of the impeller blade to the vaned diffuser potential field was studied numerically by Gottfried and Fleeter (2002). A small perturbation model was developed that considers the number of impeller blades and diffuser vanes, the impeller blade backswept angle, the impeller rotational speed, and the mass flow rate. The model was applied to a high-speed impeller-vaned diffuser configuration. It was found that increasing back-sweep angle caused the unsteady pressure differential magnitude along the entire impeller blade to decrease. Changing the number of diffuser vanes can significantly change the impeller blade unsteady pressure levels. The phase changes more rapidly along the blade as the number of diffuser vanes increase. Increasing the radial spacing between the impeller and diffuser is suggested when facing an aerodynamically-forced impeller vibration problem. The mass flow rate
decreases; the unsteady pressure differential magnitude increases. Also unsteady pressure differential magnitudes increase as the compressor speed increases.

Ziegler (2002) performed an experimental study of impeller-diffuser interaction. The attention is mainly directed to the radial gap, as it determines the intensity of the impeller and wedge type diffuser interaction. The results show that in most cases smaller radial gaps lead to a more homogeneous flow field at the diffuser vane exit and to a higher diffuser pressure recovery resulting in higher compressor efficiency. On the other hand, impeller efficiency is hardly affected by the radial gap. It was also found that in most cases the flow field at the diffuser vane exit is more homogeneous at smaller radial gaps indicating a much better diffusion. The reason is that decrease of the radial gap leads to an unloading of the typically highly loaded vane pressure side.

The stator-rotor interaction in a compressor attracts little attention. In this area, most studies are about transonic compressors. Gorrel (2003) did a numerical analysis on a high-speed, highly loaded compressor consisting of three blade rows: wake generation, rotor and stator. He concluded that additional loss occurs when the two blade rows are spaced closer together axially.

Table 2 includes most of the studies of interaction in centrifugal compressors. All of the interaction studies concentrated on the interaction between the radial impeller and radial diffuser where the compressors are large scale.
Table 2 Studies of interaction in centrifugal compressors

<table>
<thead>
<tr>
<th>Author</th>
<th>Study Method</th>
<th>Type of Machine</th>
<th>Conclusion</th>
</tr>
</thead>
<tbody>
<tr>
<td>Benini, E., Toffolo, A. (2003)</td>
<td>Numerical-CFD</td>
<td>Single Stage, Axial Vaned Diffuser</td>
<td>Unsteady simulation found that the flow field fluctuation is remarkable only in the semi-vaneless gap, whereas the diffuser blade channel is not substantially affected by this phenomenon.</td>
</tr>
<tr>
<td>Bonaituti, D., Arnone, A. Milani, A. (2003)</td>
<td>Numerical-CFD</td>
<td>Multistage, Vaneless Diffuser</td>
<td>Simplest assumption of axial uniform inlet flow leads to overestimation of the impeller efficiency by about 1~1.5% and does not provide the correct positioning of the operating envelope.</td>
</tr>
<tr>
<td>Filipenco, V.G., Deniz, S., Johnston J.M., Greitzer, E.M., and Cumpsty, N.A., (2000)</td>
<td>Experimental Measurement</td>
<td>Single Stage, Vaned Diffuser</td>
<td>The overall diffuser pressure recovery coefficient, based on suitably averaged inlet total pressure, was found to correlate well with momentum-averaged flow angle into the diffuser. The pressure recovery coefficient was found to be essentially independent of the axial distortion at diffuser inlet and the Mach number, over the wide flow range investigated. The generally accepted sensitivity of diffuser pressure recovery performance to inlet flow distortion and boundary layer blockage can be largely attributed to inappropriate quantification of the dynamics pressure at the inlet. The two types of diffusers, discrete-passage and straight-channel diffusers, showed similar behavior regarding the dependence of pressure recovery on diffuser inlet flow angle and the insensitivity of the performance to inlet flow field axial distortion and Mach number.</td>
</tr>
<tr>
<td>Author(s)</td>
<td>Type</td>
<td>Configuration</td>
<td>Details</td>
</tr>
<tr>
<td>-----------</td>
<td>-----------------------</td>
<td>---------------</td>
<td>---------</td>
</tr>
<tr>
<td>Fletter, S., Jay, R.L., Bennett, W.A. (1978)</td>
<td>Experimental Measurement</td>
<td>Single Stage, Large Scale, Low Speed</td>
<td>The experimental measurement studied the chordwise distribution of the dynamic pressure coefficient changing with the rotor wake frequency. A correlation was formed for the aerodynamic cascade transverse gust analysis. The correlation was quite good for all reduced frequency values for small values of incidence.</td>
</tr>
<tr>
<td>Hillewaert, K., Braembussche, R.A.V.D. (1999)</td>
<td>Numerical-Three-D N-S</td>
<td>Vaneless Diffuser Impeller with splitter</td>
<td>Due to pressure waves generated in the impeller by the sudden pressure rise at the volute tongue and reflected at the impeller inlet, the wavy variations were observed. The blade forces changed twice during each rotation. The weaker wave with four periods per rotation, visible at the diffuser inlet, resulted from the reflection of pressure waves on the leading edge plane of the splitter vane.</td>
</tr>
<tr>
<td>Inoue, M. and Cumpsty, N.A. (1984)</td>
<td>Experimental Measurement</td>
<td>Single Stage, Large Scale, Low Speed, Vaneless and Vaned Diffuser</td>
<td>The core of the wake at the impeller outlet changed its position from the shroud to the hub past the suction surface as the throughflow rate was decreased. The amount of reversed flow for vaned diffusers with a small vaneless ratio was significantly greater than that for the vaneless diffuser, but the strength of the reversed flow was weakened as the vane number increased. The circumferential distortion from the impeller was attenuated very rapidly in the entrance region of the diffuser vanes and surprisingly had only minor effects on the flow inside the vaned diffuser passage.</td>
</tr>
<tr>
<td>Kirtley, K.R., Beach, T.A. (1992)</td>
<td>Numerical-Deterministic</td>
<td>Single Stage Radial Vaned Diffuser D=1.524m Low Speed: 1869RPM</td>
<td>The potential interactions between blade rows are small. The large secondary flows generated in the impeller give rise to deviations in the exit flow angle at the impeller exit, which cause the vane to operate off design angle. There is negative pressure recovery up to the throat of the vane and a large suction side boundary layer which eventually detaches.</td>
</tr>
<tr>
<td>Author(s)</td>
<td>Method</td>
<td>Type</td>
<td>Description</td>
</tr>
<tr>
<td>----------</td>
<td>--------</td>
<td>------</td>
<td>-------------</td>
</tr>
<tr>
<td>Krain, H. (1981)</td>
<td>Experimental Measurement</td>
<td>Single Stage Radial Vaned Diffuser</td>
<td>The compressor has a radially ending splitter blade impeller. Separately performed measurements within adjacent channels of the splitter blade impeller reveal different flow patterns aft of the splitter blade leading edge. The lengthened flow path along the suction side of the main blade causes an increased boundary layer development, accompanied by a premature tendency to separation. A highly distorted, unsteady flow character has been analyzed within the vaned diffuser entrance region.</td>
</tr>
<tr>
<td>Sideris, M. T., Braembussche, R.A.V.D. (1987)</td>
<td>Experimental Measurement Theoretical Model</td>
<td>Vaneless Diffuser</td>
<td>The outlet volute influences the flow inside the impeller through the static pressure variation at the impeller exit. The velocity variations at the impeller exit are partly due to an unsteady acceleration or deceleration of the relative flow in the blade passage and partly due to a change in outlet flow angle.</td>
</tr>
<tr>
<td>Tamaki, H., Nakao, H., and Saito, M., (1999)</td>
<td>Experimental Measurement</td>
<td>Single Stage Vaned Diffuser</td>
<td>The study observed the overall performance of the compressor by considering the interaction between blade rows. Vaned diffusers covered the impeller operating range as broadly as possible. By reducing the diffuser throat area, the compressor could be operated at a flow rate less than 40 percent of its design flow rate.</td>
</tr>
<tr>
<td>Ubaldi, M., Zunino, P., Ghiglione, A., (1998)</td>
<td>Experimental Measurement</td>
<td>Vaneless Diffuser</td>
<td>Wake is generated in the corner of the shroud and pressure side. Moving downstream this defect develops and at the impeller outlet extends midspan. Due to the lightly loaded backward-swept blades, the suction side blade boundary layer is not separated and the jet and wake flow structure is not presented. At the diffuser inlet, the flow angle variations across the blade wake are larger than 30 degrees. Flow within the impeller is characterized by the existence of an extended zone of low turbulence. On the contrary, turbulence</td>
</tr>
</tbody>
</table>
intensity is higher in the diffuser, as a result of the discharge of turbulent flow from the impeller and the turbulence production associated with the mixing-out of flow distortions.

| Ziegler, K.U., Gallus, H.E., and Neihuis, R. (2003) | Experimental | Single Stage, Vaned Diffuser | Total pressure ratio rises as radial gap decreases. The impeller has a slightly higher work input at smaller radial gaps. The flow field at the diffuser vane exit is more homogeneous at smaller radial gaps. The reason is that a decrease of the radial gap leads to an unloading of the typically highly loaded vane pressure side. At the impeller exit, a typical jet-wake structure was found, becoming more pronounced at higher rotational speeds and at higher mass flow rates. Slip is hardly influenced by the diffuser. Angle non-uniformities mix out better than differences in velocity. |
2.3 Tip leakage effect

As for any turbomachinery, leakage flow through the tip clearance of blades deteriorates the performance. According to Senoo and Ishida (1986), the tip clearance loss consists of two kinds of loss. One kind is due to flow leakage through the clearance and the other is the pressure loss to support the fluid in the annular clearance space between the shroud and the blade tip against the pressure gradient in the meridional plane without blades.

Mashimo et. al. (1979) found that the fluid loss caused by the leakage flow through the clearance depends on Reynolds number, and the loss increases as the Reynolds number increases. Not only was the efficiency of the impeller affected, but also the relative flow angle was reduced significantly by an increase in tip clearance, which was found by Ishida, Senoo, and Ueki (1990). The change of relative flow angle can cause mismatch of the flow angle to the downstream component - diffuser.

The tip clearance effect on a miniature centrifugal impeller is critical. Compared to traditional centrifugal compressors, the miniature impeller has a relatively larger ratio of tip clearance to blade height. Moshimo, Watanabe, and Ariga (1979) studied the tip leakage on large-scale impellers. They considered the relative tip clearance about 1.2% ~ 12%. The decrease of efficiency due to the tip clearance was only about one-third of a percentage point for each percent of tip clearance ratio, which was less than 4%. Bindon (1989) quantified the tip leakage loss of an impeller, where the blade length was 456 mm. The tip leakage loss was about 39% of the total loss. Weib et. al. (2003) numerically
studied the influence of the tip clearance on the wake formation inside of a centrifugal impeller. The results showed that a relatively large clearance height at the leading edge combined with a small height at the trailing edge moved the wake further to the suction side, which corresponds well with the experimental results.

Tip leakage loss analysis is not available for the miniature impeller, which is the focus of this study.

There are two ways to study the tip leakage loss: numerical simulation and experimental measurement. There are certain difficulties encountered with the experimental measurement in a miniature scale, where the gap is only about 10 ~ 25 microns. Meanwhile, the accuracy of numerical prediction is still arguable. Recently, many researchers have applied CFD methods to the tip leakage studies. Eum et. al. (2002) numerically studied the tip clearance effect on the performance of a centrifugal compressor with six different tip clearances. He suggested that CFD predicted performance and efficiency drop quite successfully.

2.4 Effect of size on the performance

Fluid mechanics are scale dependent (Jacobson, 1998). Viscous forces are more important at a small scale. As size goes down, precision loss and frictional losses go up.

Kang (2001) did a dimensionless analysis and studied scale implications for turbomachinery. He found that the length and time (period) are proportional. For example, if a gas turbine is made smaller (shorter length), it will spin faster (shorter
period), assuming the energy density of the driving gas is the same. He also concluded that the power density of turbomachinery increases as its size goes down. The maximum linear speed of a rotor blade tip is determined by the strength of the rotor material, so keeping constant surface speeds for turbomachinery at different size scales is a good idea. The maximum value of $D \times N$ is then constant for two rotors of different size but of the same shape and material. The constant rotor surface speed implies the same flow speed in turbomachinery. The blade tip speeds are similar for the turbomachine with different diameters. Since the power density of a small engine is higher than that of a larger machine, the former can generate the same power with smaller volume. Epstein (2000) pointed out that high power density implies highly stressed rotating structures, and also, high speed rotating machinery generally requires high precision manufacturing to maintain tight clearance and good balance.

Recently, miniature centrifugal compressors have drawn more attention. However, most of the results are unpublished. Toffolo (2003) studied the centrifugal compressor of a 100 KW microturbine numerically. The interaction between the impeller and diffuser was studied by using FLUENT. Transient solutions show that some of the geometrical characteristics of the diffuser are not properly matched to the flow leaving the impeller, leading to poor overall diffuser performance. There is no detailed analysis of loss presented in this paper.
CHAPTER THREE: NUMERICAL PROCEDURE

3.1 FLUENT and turbomachinery applications

In this research FLUENT, a commercial CFD code, is used as a study instrument. The FLUENT solvers are based on the finite volume method. The domain is discretized into a finite set of control volumes or cells. General conservation (transport) equations for mass, momentum, energy, etc, in the form of partial differential equations are discretized into a system of algebraic equations. All the algebraic equations are then solved numerically to render the solution field.

Recently, FLUENT has played a big role in turbomachinery study and design. Some prestigious companies, such as BMW Rolls-Royce and Honeywell Engines & System, turn to FLUENT and use it as the building blocks in the turbomachinery design process. In predicting turbomachinery performance, FLUENT has a high degree of accuracy. Some validation cases have been implemented.

Case 1: Fluent Inc. (FLUENT 2003a) did a simulation of a HVAC fan, and the solution was compared to experimental data. The FLUENT model involved a fully unstructured tetrahedral mesh using a realizable $k-\varepsilon$ model. Excellent agreement was found between CFD results and the experimental wind tunnel data (shown in Figure 5).
Case 2: Fluent Inc. performed another validation case recently. A centrifugal pump (shown in Figure 6) used in the water industry was studied using FLUENT. This case contained approximately 1.5 million unstructured test cells, and used a multiple reference frame (MRF) model and a realizable $k-\varepsilon$ turbulence model. The pump speed of 3000 RPM and airflow rate of 1632 m$^3$/hr is the same as experimental settings. The hydraulic efficiency was computed to be 0.967, which compared very favorably with the measured value of 0.964.
When using FLUENT to conduct the turbomachinery simulation, one of the most important issues is to select an appropriate turbulent model, since flow is highly three dimensional and turbulent.

### 3.2 Turbulent modeling

For modeling turbulence, FLUENT provides the following choices of turbulence models: Spalart-Allmaras model; $k-\varepsilon$ models (standard, renormalization-group, and realizable); $k-\omega$ models (standard, shear-stress transport); Reynolds stress; and large eddy simulation (LES) model. The realizable $k-\varepsilon$ model is suggested for turbomachinery applications. The $k-\varepsilon$ models have a form with transport equations for $k$ and $\varepsilon$. In this research, the realizable $k-\varepsilon$ turbulence model is applied.

The case, which studies the turning bend loss, employs both the realizable $k-\varepsilon$ model and $k-\omega$ model separately, in order to make the comparison. In this case, large separation exists, and $k-\omega$ usually predicts separation better than other models. The pressure loss coefficient of the impeller passage calculated with realizable $k-\varepsilon$ model is about 30%, and that calculated with $k-\omega$ model is about 29%. The difference between two results calculated from different turbulent model is small. The realizable $k-\varepsilon$ model is sufficient to predict separation flow.
3.2.1 Realizable $k-\varepsilon$ model

“Realizable” means that the model satisfies certain mathematical constraints on the normal stresses, consistent with the physics of turbulent flows.

Consider combining the Boussinesq relationship

$$-\rho u_i'u_j = \mu_i \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \left( \rho k + \mu_i \frac{\partial u_i}{\partial x_i} \right) \delta_{ij}$$

and the eddy viscosity definition

$$\mu_i = \rho C_\mu \frac{k^2}{\varepsilon} \ (C_\mu \text{ is constant})$$

The following expression for the normal Reynolds stress in an incompressible strained mean flow:

$$\overline{u^2} = \frac{2}{3} k \frac{\partial U}{\partial x}$$

may obtain the result that the normal stress, $\overline{u^2}$, which by definition is a positive quantity, becomes negative when the strain is large enough to satisfy

$$\frac{k \frac{\partial U}{\partial x}}{\varepsilon} > \frac{1}{3C_\mu} \approx 3.7$$

Similarly, it can also be shown that the Schwarz inequality for shear stress can be violated when the mean strain rate is large. Another weakness of the standard $k-\varepsilon$ and other traditional $k-\varepsilon$ models lies with the modeled equation for the dissipation rate ($\varepsilon$).

The realizable $k-\varepsilon$ model (Shih et. al.) addresses these deficiencies of traditional $k-\varepsilon$ models by adopting

a) a new eddy-viscosity formula involving a variable $C_\mu$

b) a new model equation for dissipation ($\varepsilon$) based on the dynamic equation of the mean-square vorticity fluctuation.
3.2.2 Realizable $k-\varepsilon$ model transport equations

The modeled transport equations for $k$ and $\varepsilon$ in the realized $k-\varepsilon$ model are as following.

$$\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho k u_j)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \mu + \frac{\mu_t}{\sigma_k} \frac{\partial k}{\partial x_j} \right] + G_k + G_b + \rho \varepsilon - Y_M + S_k$$

$$\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho \varepsilon u_j)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \mu + \frac{\mu_t}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_j} \right] + \rho C_1 S \varepsilon - \rho C_2 \frac{\varepsilon}{k + \sqrt{\varepsilon} \rho \varepsilon} + C_1 C_\varepsilon \frac{\varepsilon}{k} C_\varepsilon G_b + S_\varepsilon$$

where

$$C_1 = \max \left[ 0.43, \frac{\eta}{\eta + 5} \right]$$

$$\eta = \frac{k}{\varepsilon}$$

$$G_k = -\rho \mu_t' u_j \frac{\partial u_j}{\partial x_i}$$, which is the generation of turbulence kinetic energy due to the mean velocity gradient;

$$G_b = \beta g_i \frac{\mu_t}{Pr_t} \frac{\partial T}{\partial x_i}$$, which is the generation of turbulence kinetic energy due to buoyancy;

$$Y_M = 2 \rho \varepsilon M_i^2$$, which is the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate where $M_i = \sqrt{\frac{k}{a^2}}$ is the turbulent Mach number;
$C_1$ and $C_{ke}$ are constants;

$\sigma_k$ and $\sigma_\varepsilon$ are the turbulent Prandtl numbers for $k$ and $\varepsilon$;

$S_k$ and $S_\varepsilon$ are user-defined source terms.

### 3.2.3 Modeling the turbulent viscosity

The eddy viscosity is computed from $\mu_* = \rho C_\mu \frac{k^2}{\varepsilon}$.

$C_\mu$ is no longer a constant as in the other $k-\varepsilon$ models, but a function of the mean strain and rotation rates.

$$C_\mu = \frac{1}{A_0 + A_s \frac{kU^*}{\varepsilon}}$$

where $U^* = \sqrt{S_{ij}S_{ij} + \Omega_{ij}\Omega_{ij}}$ ; $\Omega_{ij} = \Omega_{ij} - 2\varepsilon_{ijk}\sigma_k$ ; $\Omega_{ij} = \overline{\Omega_{ij}} - \varepsilon_{ijk}\sigma_k$

where $\overline{\Omega_{ij}}$ is the mean rate-of-rotation tensor viewed in a rotating reference frame with the angular velocity $\sigma_k$. $A_0 = 4.04$ , $A_s = \sqrt{6} \cos \phi$ , $\phi = \frac{1}{3} \cos^{-1} \left( \sqrt{6}W \right)$ ,

$$W = \frac{S_{ij}S_{jk}S_{kl}}{S} , \quad S = \sqrt{S_{ij}S_{ij}} , \quad S_{ij} = \frac{1}{2} \left( \frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right).$$

### 3.2.4 Model constants

$C_{ke} = 1.44$

$C_2 = 1.9$
\[ \sigma_k = 1.0 \]
\[ \sigma_\varepsilon = 1.2 \]

### 3.3 Near-Wall treatment

Turbulent flows are significantly affected by the presence of the wall, because of the no-slip condition at the wall. Near the wall region, viscous damping reduces the tangential velocity fluctuations, while kinematics blocking reduces the normal fluctuations. The turbulence is rapidly increased by the production of turbulence kinetic energy due to the large gradient in mean velocity in the direction outward to the near-wall region. The near-wall modeling significantly affects the fidelity of numerical solutions, because the walls are the main source of mean vorticity and turbulence. Accurate representation of the flow in the near-wall region is important.

Near-wall region definition was derived from numerous experiments. The region can be divided into three layers: viscous sublayer (almost laminar), buffer layer and outer layer (fully turbulent layer).

There are two approaches to modeling the near-wall region, wall functions and near-wall model.

1. Wall functions are semi-empirical formulas used to bridge the viscosity-affected region between the wall and the fully turbulent region. For high Reynolds number flow, the wall function approach substantially saves computational resources.

2. The near-wall modeling approach solves the viscosity-affected region with mesh
all the way to the wall, which has advantages in solving the problem for low Reynolds number flow.

Fluid flow in turbomachinery passages is highly turbulent. Therefore, the wall function approach is suitable for most simulations. However, when the tip clearance is considered, only the near-wall modeling, also called two-layer modeling, is able to capture the leakage flow.

Wall functions are comprised of laws-of-the-wall for mean velocity and temperature; formulas for near-wall turbulent quantities. FLUENT offers two choices of wall function approaches: standard wall functions, and non-equilibrium wall functions. Non-equilibrium wall functions are recommended for use in complex flows involving separation, reattachment, and impingement where the mean flow and turbulence are subjected to severe pressure gradients and change rapidly.

### 3.3.1 Standard wall function

The standard wall function is based on the proposal of Lauder and Spalding.

\[ U^* = \frac{1}{\kappa} \ln(Ey^*) \]

where

\[ U^* \equiv \frac{\rho C^{1/4} \mu^{1/2} k_{p}^{1/2}}{\tau_{w}/P} \]

\[ y^* \equiv \frac{\rho C^{1/4} \mu^{1/2} y_{p}}{\mu} \]
and \( \kappa = \) Von Karman constant (0.42); \( E = \) empirical constant (9.793); \( U_p = \) mean velocity of the fluid at point P; \( k_p = \) turbulence kinetic energy at point P; \( y_p = \) distance from point P to the wall; \( \mu = \) dynamic viscosity of the fluid.

The law is valid for \( y' > 30 \sim 60 \). In FLUENT, the log-law is employed when \( y' > 11.225 \). When the mesh is such that \( y' < 11.225 \) at the wall-adjacent cells, FLUENT applies the laminar stress-strain relationship \( U' = y' \). In FLUENT, the laws-of-the-wall for mean velocity and temperature are based on the wall unit, \( y' \) rather than \( y' = \rho u_{\tau} y / \mu \).

**Energy**

Reynolds’ analogy between momentum and energy transport gives a similar logarithmic law for mean temperature. FLUENT comprises the following two different ways for temperature’s law-for-wall.

1. Linear law for the thermal conduction sublayer where conduction is important.
2. Logarithmic law of the turbulent region where effects of turbulence dominate conduction.

Thickness of the thermal conduction layer is different from the thickness of the viscous sublayer.
\[ T^* = \frac{(T_w - T_p) \rho c_p C_{\mu}^{1/4} k_p^{1/2}}{q} \]

\[ \begin{cases} \text{Pr} y^* + \frac{1}{2} \rho \text{Pr} \frac{C_{\mu}^{1/4} k_p^{1/2}}{q} U_p^2 \quad (y^* < y_T^*) \\
\text{Pr} \left[ \frac{1}{\kappa} \ln(E y^*) + P \right] + \\
\frac{1}{2} \rho \frac{C_{\mu}^{1/4} k_p^{1/2}}{q} \left\{ \text{Pr}_t U_p^2 + (\text{Pr}_t - \text{Pr}_m) U_c^2 \right\} \quad (y^* > y_T^*) \end{cases} \]

where \( P = 9.24 \left[ \left( \frac{\text{Pr}}{\text{Pr}_t} \right)^{3/4} - 1 \right] \left[ 1 + 0.28 e^{-0.007 \text{Pr}/\text{Pr}_t} \right]; \)

\( k_f \) = thermal conductivity of fluid;
\( \rho \) = density of fluid;
\( c_p \) = specific heat of fluid;
\( q \) = wall heat flux;
\( T_p \) = temperature at the cell adjacent to the wall;
\( T_w \) = temperature at the wall;
\( \text{Pr} \) = molecular Prandtl number (\( \mu c_p / k_f \));
\( \text{Pr}_t \) = turbulent Prandtl number (0.85 at the wall);
\( A = 26 \) (Van Direst constant);
\( \kappa = 0.4187 \) (Von Karman constant);
\( E = 9.793 \) (wall function constant);
\( U_c \) = mean velocity magnitude at \( y^* = y_T^* \).

**Turbulence**
The boundary condition of $k$ imposed at the wall is $\frac{\partial k}{\partial n} = 0$. The production of $k$ is computed from $G_k \approx \tau_w \frac{\partial U}{\partial y} = \tau_w \frac{\tau_w}{\kappa \rho C^{1/4}_\mu k^{1/2}_p y_p}$, and $\varepsilon_p = \frac{C^{3/4}_\mu k^{3/2}_p}{\kappa y_p}$.

Standard wall functions work reasonably well for a broad range of wall-bounded flows.

### 3.3.2 Non-equilibrium wall functions

Non-equilibrium wall functions keep mean temperature equations the same as the standard wall function, but use log-law for mean velocity sensitized to pressure gradients.

$$\frac{\bar{U} C^{1/4}_\mu k^{1/2}}{\tau_w \rho} = \frac{1}{\kappa} \ln \left( \frac{E \rho C^{1/4}_\mu k^{1/2}_p y}{\mu} \right)$$

where $\bar{U} = U - \frac{1}{2} \frac{dp}{dx} \left[ \frac{y_v}{\rho \kappa \sqrt{k}} \ln \left( \frac{y}{y_v} \right) + \frac{y - y_v}{\rho \kappa \sqrt{k}} + \frac{y_v^2}{\mu} \right]$;

$y_v \equiv \frac{\mu y_v^*}{\rho C^{1/4}_\mu k^{1/2}_p}$, which is the physical viscous sublayer thickness;

$y_v^* = 11.225$.

The non-equilibrium wall function employs the two-layer concept in computing the budget of turbulence kinetic energy at the wall-adjacent cells, which are needed to solve the $k$ equation at the wall-neighboring cells. The wall-neighboring cells are assumed to
consist of a viscous sublayer and a fully turbulent layer. The following profile assumptions for turbulence quantities are made:

\[
\tau_r = \begin{cases} 
0 & y < y_v \\
\tau_w & y > y_v 
\end{cases}
\]

\[
k = \begin{cases} 
\left(\frac{y}{y_v}\right)^2 k_p & y < y_v \\
k_p & y > y_v 
\end{cases}
\]

\[
\varepsilon = \begin{cases} 
\frac{2\nu k}{y_v^2} & y < y_v \\
\frac{k^{2/3}}{C_l y} & y > y_v 
\end{cases}
\]

where \( C_l = \kappa C_{\mu}^{-3/4} \).

Using these profiles, the cell-averaged production of \( k \) and the cell-averaged dissipation rate \( \overline{\varepsilon} \) can be computed from the volume average of \( G_k \) and \( \varepsilon \) of the wall-adjacent cells.

The standard wall functions give reasonably accurate predictions for the majority of high Reynolds number, wall-bounded flow. The non-equilibrium wall functions further extend the applicability of the wall function approach by including the effects of pressure gradient and strong non-equilibrium. However, the wall function approach becomes less reliable when the flow conditions depart too much from the ideal conditions underlying the wall functions.
3.3.3 Near-wall modeling

In the near-wall model, the viscosity-affected near-wall region is completely solved all the way to the viscous sublayer. In this approach, the whole domain is subdivided into a viscosity-affected region and a fully-turbulent region. The demarcation of the two regions is determined by a wall-distance-based, turbulent Reynolds number $Re_y$, defined as

$$Re_y \equiv \frac{Dy\sqrt{k}}{\mu}$$

where $y$ is the normal distance from the wall at the cell centers.

In the fully turbulent region ($Re_y > Re_y^*$, $Re_y^* = 200$), the $k-\varepsilon$ models or the Reynolds stress model are employed. In the viscosity-affected near-wall region ($Re_y < Re_y^*$), the one-equation model of Wolfstein is employed. In the one-equation model, the momentum equations and the $k$ equation are retained as described above. However, the turbulent viscosity is computed from

$$\mu_{r,2layer} = \rho C_{r} l_\mu \sqrt{k}$$

where the length scale is computed from

$$l_\mu = yC_{l}(1 - e^{-Re_y/A_\mu}).$$

The two-layer model requires that the near-wall mesh is fine enough to be able to resolve the laminar sublayer, typically $y^+ = 1$. The restriction that the near-wall mesh must be sufficiently fine everywhere might impose too large a computational requirement.
3.4 Computation with rotating elements

The solution of flows in moving reference frames requires the use of moving cell zones, which are interpreted as the motion of a reference frame to which the cell zone is attached. In the case where both the stator and rotor are present, FLUENT has three solutions for the multi-moving zone cases: Multiple Reference Frame (MRF) model, Mixing Plane Model (MPM) and Sliding Mesh Model (SMM). In this study, only the SMM is employed, which is able to predict interaction between blade rows.

The SMM assumes the flow field is unsteady, models the interaction with complete fidelity, and hence is the most accurate model for simulating flows in multiple moving reference frames. However, in many situations it is not practical to employ the SMM. The SMM interfaces can be rotationally periodic, but adjacent zones must have equal periodic angles. Therefore, in the cases such as the ones in this study where the number of blades is different for each blade row, a large number of blade passages are needed in order to maintain circumferential periodicity. Since the SMM can well capture the interaction between blade rows, in this research it is used as a main tool regardless of the tremendous meshes required.

When rotor-stator interactions are strong and where a time-accurate solution is desired, the SMM is necessary for an unsteady flow field. In the sliding mesh technique, two or more cell zones are used. Each cell zone is bounded by at least one “interface zone” where it meets the opposing cell zone. The interface zones of adjacent cell zones are associated with one another to form a “grid interface”. The two cell zones will move relative to each other along the grid interface. During the calculation, the cell zones slide
relative to one another along the grid interface in discrete time steps. Grid interface could be any shape. The SMM (moving mesh) formulation assumes that the computational domain moves relative to the stationary frame, so there is no reference frame attached to the computational domain.

Theory

To compute the interface flux, the intersection between the interface zones is determined at each new time step. The resulting intersection produces one interior zone (a zone with fluid cells on both sides) and one or more periodic zones. The resultant interior zone corresponds to where the two interface zones overlap; the resultant periodic zone corresponds to where they do not. The number of faces in these intersection zones will vary as the interfaces move relative to one another. Principally, fluxes across the grid interface are computed using the faces resulting from the intersection of the two interface zones, rather than from the interface zone faces themselves.

In the example (as shown in Figure 7), the interface zones are composed of faces A-B and B-C, and faces D-E and E-F. The intersection of these zones produces the faces a-d, d-b, b-e, etc. Faces produced in the region where the two cell zones overlap (d-b, b-e, and e-c) are grouped to form an interior zone, while the remaining faces (a-d and c-f) are paired up to form a periodic zone. To compute the flux across the interface into cell IV, for example, face D-E is ignored and face d-b and b-e are used instead, bringing information into cell IV from cells I and III, respectively.
Time step for the SMM

Since the SMM is transient, the time step is needed. The recommended time step size is based on the principal that the time step should be no larger than the time it takes for a moving cell to advance past a stationary point. An estimate for the time step can thus be calculated as:

$$\Delta t \approx \frac{\Delta s}{\omega R}$$

where $\Delta s$ = mesh spacing at sliding interface; $\omega$ = rotational speed; $R$ = radius of interface.

Time-periodic

The SMM calculation terminates whenever the solution reaches steady time-periodic. The final time-periodic solution is independent of the time steps taken during the initial
stages of the solution procedure. To determine how the solution changes from one period to the next, it is needed to compare the solution at the same point in the flow field over two periods. Some global quantities can be tracked, such as lift and drag coefficient, mass flow rate, and so on, as shown in Figure 8

![Figure 8 Sliding mesh model time-periodic solution](image)

3.5 Settings

3.5.1 Boundary conditions

FLUENT has several options of input boundary conditions for turbomachinery applications.
a. pressure-inlet/pressure-outlet
b. velocity-inlet/pressure-outlet (incompressible flows only)
c. mass-flow-inlet/pressure-outlet

For most of the simulations, the pressure-inlet/pressure-outlet option is preferable. In this study, the IGV inlet condition is a total pressure of 1 atm and the diffuser outlet condition is a static pressure of 1.6 atm. Other values depend on the operating conditions.

A periodical boundary condition is used to reduce the computational requirement. The IGV, impeller, and diffuser have a common periodic angle of thirty six degrees.

3.5.2 Control of solution: under-relaxation

The sliding mesh model is quite stable. Therefore, default under-relaxation is used for each of the parameters.

3.5.3 Control of solution: discretization

A second-order upwind scheme is used for all the parameters for better accuracy. Quantities at cell faces are computed using a multidimensional linear reconstruction approach. In this approach, higher-order accuracy is achieved at cell faces through a Taylor series expansion of the cell-centered solution about the cell centroid. The face value $\phi_f$ is computed from $\phi_f = \phi + \nabla \phi \cdot \Delta \vec{S}$, where $\phi = \text{cell-centered value}$, $\nabla \phi = \text{gradient}$, and $\Delta \vec{S} = \text{displacement vector from the cell centroid to the face centroid}$. 
\[ \nabla \phi = \frac{1}{V} \sum_{f}^{N_{\text{faces}}} \phi_{f} \cdot A, \]

where \( \phi_{f} \) are computed by averaging \( \phi \) from the two cells adjacent to the face. Then \( \nabla \phi \) is limited so that new maxima or minima are introduced.

### 3.6 Grid generation

The generation of an appropriate grid is an essential element of an accurate flow solution process along with the turbulent model. Adequate resolution of important flow phenomena in turbomachinery flow, such as boundary layer jet and wakes, are crucial to obtaining physically representative flow solutions. Numerical study of tip leakage relies heavily on the grid density and quality. The results can be highly grid dependent.

#### 3.6.1 Structured grid

Structured grids have the advantage that connectivity arrays are trivial, thus simplifying the calculation of higher derivatives encountered in viscous flow computations. The generation of structured grids for the computation of viscous flow in the centrifugal compressor blade is desirable. To maintain the accuracy of difference approximations of the derivatives in the solver, structured grids have to be smooth, i.e., the local radius of curvature of a grid line must be large compared to the local length of the grid line which must be large compared to the local length of grid segments. But the smoothest grids will, in general, not provide sufficient grid resolution in regions of large gradients of flow properties, e.g., at leading and trailing edges, and in boundary layers.
and wakes. Another consideration is orthogonality. In turbulent flow analyses, the application of turbulence models is greatly simplified when the grid lines intersect the solid boundaries at right angles. Although orthogonality of the grid over the whole domain would be advantageous, it is not possible to enforce orthogonality in periodic grids for blade rows with nonzero inlet or outlet angles. The boundary-value problem nature of grid generation for internal flows (the domain is completely enclosed by the boundaries) and the smoothness requirement make control of the mesh density important. For turbomachinery geometry, the leading and trailing edges are particular areas where control over the distribution of points is required. Highly cambered cascades also require control over the spacing of points over the remainder of the blade profile to resolve large gradients that occur there.

A typical H-Grid for turbomachinery was presented and modified by Basson (1993). GAMBIT (pre-processor) has the add-on G-Turbo, which is designated to produce H, H-O, and H-C type grids. However, the grid automatically generated has very poor orthogonality. The skew in some regions reaches 0.8. Some modifications are needed for a better-structured grid.
In this study, the performance of the impeller with and without tip leakage is predicted using a structured grid. With tip leakage, the case is very much grid-dependent. Tip clearance is 0.0025 m uniformly from inlet to outlet of the impeller. To capture the friction effect, there are more than 30 grids needed in the tip clearance region. More than 1.5 million grids are required in a tip leakage study, as shown in Figure 9 and Figure 10. The geometry, shown in Figure 9, is an impeller with twenty blades, and the tip leakage flow is considered. More grids are used at the regions, where walls exist, in order to satisfy $y^+ \sim 1$. Figure 10 shows an impeller with ten blades and ten splitters. This case is used to compare with the case of an impeller with twenty blades to study blade number effect on the impeller performance.

Figure 9 Structured grid (tip leakage study)
3.6.2 Unstructured grid

Compared to structured grids, uniform unstructured grids are relatively easier to generate with GAMBIT. Fully three-dimensional test grids can be generated in TGRID. Besides, GAMBIT can also generate unstructured volume meshes automatically. Boundary layer (grid stretching) and size function (using source terms) methods are provided to create meshes with higher resolution in the regions of interest. Additionally, in FLUENT an adaptive grid method can be used to obtain the required resolution in regions of large flow gradients.

The majority of initial CFD development favored structured grids. The requirements for geometries beyond simple blade passages cause high costs in the time of the
structured meshing process. Unstructured grid methodologies have progressively taken over, since the unstructured grid generators can generate complex grids nearly automatically and in a cost effective way, with short turnaround time. However, the block structured grids still have advantages in terms of accuracy and grid quality.

In the current study, the diffuser vane has a very large angle at the inlet, as shown in Figure 11. The periodic boundary conditions are posted, which requires mesh consistency at the periodic interfaces. Therefore, to create a structured mesh in a diffuser is very difficult. Therefore an unstructured mesh is employed, as shown in Figure 12. To validate the mesh, two sets of numerical experiments were performed. First, the mesh density was varied from coarse to dense. The study showed that the solution from the current set of the unstructured mesh is independent of the grid. The overall efficiency of the compressor changes 1% when the mesh is doubled. Second, the impeller was studied with the same unstructured grid for the wall function and with a structured grid for the two-layer model. The impeller efficiency varied by 3 % when the mesh density increased by ten times.

The unsteady study of the IGV-impeller-diffuser takes a much longer time to become convergent. Convergence needs to be reached at every time step, and the time step is $10^{-6}$ s when the rotating speed is 108 KRPM. Normally, more than one thousand time steps are needed in order to reach time-periodic convergence. The calculation is very time-consuming. Therefore, the grid number is critical. When using an unstructured grid, the current grid number for the interaction study reached 440 000.
Figure 11 Geometry for unstructured grid

Figure 12 Unstructured grid
CHAPTER FOUR: RESULTS AND DISCUSSION

4.1 Inlet guide vane

The rotating speed of the miniature centrifugal compressor is very high, as a result, the blade velocity at the eye of the impeller ($U$) is much higher than that of a conventional impeller.

$$U = \omega R$$

To reduce the incidence, which is defined as the difference between the relative flow angle and the blade angle at the eye of impeller, the inlet guide vane is employed. The objective of the inlet guide vane is to bring the flow to the eye of impeller with a level of inlet swirl. By properly configuring flow fields and choosing inlet blade angles carefully, it is possible to match the incidence levels along the radial extent of the blade, thus ensuring the best efficiency and good flow control. Besides, it is also important to ensure that the inlet flow is as uniform as possible.

There are two kinds of inlet guide vanes, radial and axial. The radial inlet requires a substantially smaller cascade setting angle in order to achieve the desired impeller eye inlet angle. For angles greater than 38 degrees but less than approximately 65 degrees, the radial inflow guide vane system exhibits lower losses. The axial guide vane begins to stall at about 30 degrees and then the blade losses increase proportionally. The radial inflow guide vane does not stall until a much higher angle, and the rise in losses at approximately 55 degrees is principally due to the development of a three-dimensional
hub separation just before the eye of the impeller. This separation is a consequence of the conservation of angular momentum, because at small radii the tangential velocity component $C_{\theta}$ becomes very large. Since a higher swirl angle is desired, the current centrifugal compressor employs a radial inlet guide vane.

Design analysis was carried out on a one-dimensional basis. The velocity triangle at the cascade exit was related to the velocity triangle at the eye of the impeller by the use of the conservation of mass and conservation of angular momentum equations:

$$\rho_{0.5}C_{m0.5}A_{0.5} = \rho_{1}C_{m1}A_{1}$$

$$r_{0.5}C_{\theta0.5} = r_{1}C_{\theta1}$$

where 0.5 indicates the location at the trailing edge of the IGV vanes, and 1 indicates the location at the IGV exit.

The numerical study of the IGV focuses on the loss analysis and interaction effects from downstream components. The flow inside of the IGV is studied with and without the interaction between the IGV and impeller. Numerical simulation of the following geometries, as shown in Figure 13, Figure 14, and Figure 15, are performed separately.

i. IGV (vaned) + impeller (bladed, rotating with SMM model) + diffuser (vaned, 2 sets of diffusers)

ii. IGV (vaned) + impeller (without blade)

iii. IGV (without vane) + impeller (without blade)
Figure 13 IGV (vaned) + impeller (bladed) + diffuser (vaned)

Figure 14 IGV (vaned) + impeller (without blade)
4.1.1 IGV loss analysis

Insufficient information has been published to establish clearly the quantitative performance of inlet guide vane systems. Found by other researchers, the majority of loss in the IGV is due to wall friction. It is suggested that the intake duct should be short, its surface smooth, and the flow velocity low, although the exit velocity is dictated by the impeller inlet dimensions. At the same time, every effort must be made to minimize boundary layer growth and to avoid boundary layer separation in order to ensure steady uniform flow enters into the impeller situated immediately downstream of the intake duct. The best way to achieve attached boundary layers is to avoid a significant flow deceleration throughout the intake duct. The current design has a nozzle-like configuration, and the flow accelerates during the process.
It is found that without the impeller, the flow at the IGV exit reverses due to the separation at the bend. When interaction is included, the reversed flow at the IGV exit in the previous IGV-only simulation diminishes due to the negative static pressure field in front of the eye of the impeller, which is generated by the impeller. The flow is dragged toward the eye. The separation disappears due to the potential interaction between the IGV and the impeller.

The plot of total pressure at the mid-spanwise plane, as shown in Figure 16, shows that a wake region generates at the suction side of the vane. This is a result of non-zero incidence at the IGV inlet, as shown in Figure 17. The flow is normal to the boundary at the IGV inlet. The non-zero vane angle at the IGV inlet introduces a small flow departure from the pressure side of the vane. It can be found that the total pressure loss inside of the vane passage is mainly due to the leading edge separation of the flow.

![Figure 16 Contour of total pressure (IGV)](image-url)
Figure 18 presents the plot of averaged total pressure in the meridional direction. Even though there is no apparent separation in the flow, the total pressure drops sharply at the turning bend portion of the IGV. There are two losses existing in this region. The first is mixing loss due to unloading of the IGV vanes. The second is the loss due to the turning of the IGV flow passage.
Turning bend loss is very important in this study. The discussion of turning bend loss applies to IGV, impeller and diffuser loss analysis. It is necessary to discuss the loss mechanisms in detail.

Considering an ideal fluid with a uniform energy distribution passing through a bend, as shown in Figure 19, within the bend the static pressure increases with radius to balance the centrifugal force. Since the sum of velocity and static pressures is everywhere the same, velocities decrease from the inside to the outside of the bend.
Because the velocities vary from zero at the walls to a maximum in the core, actual flows through bends involve non-uniform energy distributions. Centrifugal and pressure forces acting on the faster moving core flow cause the core to be deflected towards the outside of the bend. Fluid approaching the outside of the bend sees the adverse pressure gradient. Energy deficient near-wall fluid approaching the outer wall cannot pass through the adverse pressure gradient, so instead it moves around the walls towards the low static pressure region on the inside of the bend. The movement of low energy fluid towards the inside of the bend, combined with the deflection of the high velocity core region towards the outside of the bend, sets up two cells of secondary flow.

Curvature causes the flow on convex surfaces to be stabilized with a reduction in turbulent shear stresses and to be de-stabilized on concave surfaces with an increase in
turbulence shear stress. Therefore, turbulent energy transfer towards the wall is enhanced, energy dissipation is increased and the velocity gradient close to the wall is steep, enabling near-wall flow to withstand adverse pressure gradients. The converse situation exists at the inner wall. Energetic particles tend to be displaced away from the inner wall so that low energy fluid accumulates near the wall. Energy transfer is reduced, the velocity gradient away from the wall becomes shallow, energy dissipation is reduced and the near-wall is unable to withstand adverse pressure gradients. The effect of the secondary flows is to re-energize the inner wall regions. When the inlet flow is nearly uniform, the secondary flows may be too weak to prevent flow separation.

Some early studies showed that most of the pressure loss is associated with re-establishing developed flow after a bend. The centerline radius to diameter ratio is a key control variable, $R/d$.

As a result of the turning bend, the flow in the turning bend should have losses in both the inner and outer passages. A large amount of low momentum flow accumulates at the inner bend. Figure 20 and Figure 21 present results from simulation based on geometry arrangement iii. Figure 20 shows the total pressure contour. Total pressure loss is observed at both the inner and outer passage. The pressure loss at the outer wall (connecting to impeller hub) is due to high turbulent energy dissipation, and at the inner wall (connecting to impeller shroud) is due to the low energy flow. Figure 21 plots the meridional velocity, which is defined by $C_m^2 = \sqrt{C^2 - C_\theta^2}$, where $C$ is velocity magnitude. The velocity at the inner wall is higher than the outer wall. Secondary flow causes more low momentum flow to move from the outer to inner sides. Figure 22 shows
the tangential velocity at the IGV exit. The tangential velocity at the hub side is higher than that at the shroud side and is a result of the conservation of angular momentum.

Figure 20 Total pressure (from simulation iii)
Based on the above analyses, there are four main types of losses associated with the IGV, which are friction loss, leading edge separation loss, mixing loss and turning bend loss. In order to better understand the IGV and to improve IGV performance, it is necessary to quantify each loss.

The pressure loss coefficient of the IGV is defined by the following equation.

\[ C_{p,loss,IGV} = \frac{P_{t,inlet,IGV} - P_{t,exit,IGV}}{P_{v,exit,IGV} - P_{s,exit,IGV}} \]

where the subscript \( t \) represents total, and \( s \) represents static. The value is mass-averaged and calculated from

\[ P_t = \frac{\int \rho V p dA}{\int \rho V dA} . \]
The pressure loss coefficients of the IGV from simulation of the three geometry arrangements are listed in Table 3.

### Table 3 IGV loss analysis

<table>
<thead>
<tr>
<th>Geometry</th>
<th>Losses included</th>
<th>Pressure loss coefficient</th>
</tr>
</thead>
<tbody>
<tr>
<td>i</td>
<td>All losses and loss due to interaction between IGV and impeller.</td>
<td>18.7%</td>
</tr>
<tr>
<td>ii</td>
<td>Friction loss of the wall and the vanes, leading edge separation loss, mixing loss and turning bend loss.</td>
<td>16.8%</td>
</tr>
<tr>
<td>iii</td>
<td>Friction loss of the wall and turning bend loss,</td>
<td>13.8%</td>
</tr>
</tbody>
</table>

After comparing the results from the three geometrical arrangements, the following can be found.

Because of the interaction, the flow separation at the IGV exit vanishes. This adds some benefit to the IGV performance. Therefore, the loss due to the interaction between the IGV and impeller is a bit more than 2%, which is more than 10% of the total pressure loss. The friction loss due to the vanes, leading edge separation loss, and mixing loss is 2%, which is an additional 10% of the total pressure loss. The main loss is due to the turning bend loss and friction loss of the wall, which is 13.8%. Therefore, more than 70% of the total pressure loss of the IGV is due to the turning bend loss and friction loss of the wall. The Reynolds number of the IGV based on the inlet condition is about 3500. The flow is highly turbulent. Accordingly, the friction loss is low and the loss in the IGV is
mainly due to turning bend loss. The overall IGV pressure loss coefficient is 18.69%, which is acceptably high.

Nevertheless, if higher IGV performance is needed, it is recommended to reduce the turning bend loss by changing the IGV shape.

4.1.2 Exit flow angle

The main purpose of employing an IGV is to generate flow swirl at the impeller. The flow angle at the IGV exit is important to evaluate the performance of the IGV.

As discussed in the previous section, the flow angle at the IGV exit is characterized by the turning effect of the passage. As a result of the conservation of angular momentum, $rC_\theta = const$, and centrifugal force, the tangential velocity decreases from hub to shroud side, as shown in Figure 22. Due to the secondary flow at the turning bend, the meridional velocity decreases from shroud to hub.

The mass flow rate is controlled by the meridional velocity. Since meridional velocity increases from hub to shroud, as shown in Figure 23, the mass flow rate at the shroud side is higher than the hub side. The shroud side is different from the hub side in two aspects. First, the shroud does not move with the impeller rotation while the hub rotates. The flow close to the shroud obtains angular momentum from the impeller rotation and also suffers from the friction due to the still shroud. Second, due to the manufacturing precision, the tip clearance exists. The tip clearance is the gap between the shroud and the impeller blade tip. Some flow inside of the tip clearance may bypass the impeller with no work added. More mass flow close to the shroud side means more flow
can bypass the impeller, which has a negative impact on the impeller efficiency. In this study, the tip leakage effect on the impeller is studied but without the IGV, because huge amounts of mesh are required for the clearance, which is very time-consuming for a transient problem. Hence, the impeller efficiency discussed in the following chapter may be over-predicted.

Figure 22 Contour of tangential velocity (IGV exit)
The flow angle is defined as in Figure 24 and calculated by

\[
\alpha = \arctan \left( \frac{C_p}{C_m} \right).
\]
The plot of the flow angle at the IGV exit from shroud to hub is shown in Figure 25.

Figure 25 Flow angle at the IGV exit

The flow angle at the IGV exit increases from the shroud side to the hub side. The flow angle at the IGV governs the impeller inducer design. The impeller blade angle should match with the flow angle at the IGV exit. It is found that the mass-averaged flow angle at the eye of impeller is around 60 degrees. For all three geometries, not only the mass-averaged flow angle but also the distribution of the flow angle at the IGV exit does not vary with the boundary conditions, such as mass flow rate and pressure ratio. The flow angle at the IGV exit is decided by the vane exit angle of the IGV.
4.1.3 Interaction with Impeller

The interaction from the impeller on the IGV is mainly potential interaction. The IGV performance, which includes loss coefficient and exit flow angle, is not a function of time. Comparing the results from case i and case ii, as listed in Table 3, the IGV loss coefficient increases more than 2% due to the interaction with the impeller.

4.1.4 Summary

The performance of the IGV is studied based on three geometrical arrangements. Some of the IGV losses are quantified. The main total pressure loss of the IGV is due to the turning bend loss. In order to improve the IGV performance, future studies should focus on changing the shape of the IGV.

The flow angle at the IGV exit is in agreement with the law of conservation of angular momentum and the turning flow pattern. The flow angle increases from shroud to hub. The results can be used for the impeller inducer design.

The interaction from the impeller on the IGV is mainly potential interaction. The IGV loss coefficient increases more than 2% due to the interaction with the impeller.

4.2 Impeller

The impeller is the heart of a compressor design. It is responsible for the energy transfer process described by the Euler turbomachinery equation.
To study miniature centrifugal compressors, it is necessary to include the interaction between upstream and downstream components and the tip leakage effect. There are several different cases considered.

i. IGV (vaned) + impeller (bladed) + diffuser (vaned)

In this case, the interaction between blade rows is considered. Two diffuser designs are applied.

ii. Impeller only (10 blades with 10 splitters).

The flow is uniform and perpendicular to the boundary at the impeller inlet.

iii. Impeller (20 blades without splitter) with and without tip leakage.

4.2.1 Impeller Inlet

The flow is drawn toward the impeller. An essential function of the impeller is to decrease the static pressure at the face of the impeller. With the IGV, the flow approaches the eye of the impeller with an angle, which was discussed in the previous section. As soon as the flow approaches the leading edge of the blade, it experiences strong forces created by the impeller blades, and most fluid elements begin to follow a pseudo-helical pattern through the impeller domain.
The process of fluid entry to the impeller is very important. The incidence directly affects the performance of the impeller. Most designs changed so that the blading was bent in the direction of the relative approaching flow, allowing the entering flow to have controlled incidence level, called an inducer. If the inducer is well designed and carefully fabricated, then very good performance can be achieved. Because the IGV gives the flow certain level tangential velocity, the impeller in this design utilizes a smaller inducer than the conventional one, as shown in Figure 26. The rotating speed of the impeller at the design point is 313 KRPM when helium is employed. Because of its smaller size, the mass flow rate, Reynolds number, and Mach number are kept low. To reduce friction loss and blade loading, a smaller inducer is used.

Incidence angle is the difference between the relative flow angle and the blade angle at the impeller inlet. The relative flow angle is a function of the flow velocity and the blade velocity. Low absolute incidence angle is preferable. Due to the centrifugal force, the flow angle at the IGV exit is non-uniform. As a function of radius, the blade...
velocities also vary from shroud to hub. Figure 27 shows the relative flow angle at hub and shroud. The blade angle at the shroud side is 72 degrees and at the hub side is 54 degrees. For both locations, large amount of incidence exists. Roughly, the incidence is 2 degrees at the hub side and 60 degrees at the shroud side. The current design of the IGV creates a positive incidence at the impeller eye, especially at the shroud side. Theoretically, positive incidence is better than negative incidence, since the higher pressure on the pressure side of the blade tends to suppress the flow and prevent it from separating from the blade. However, too much positive incidence angle can still cause flow separation at the pressure side of the impeller blade.

![Figure 27 Relative flow angle (Impeller inlet) (velocity units: m/s)](image)

Figure 27 Relative flow angle (Impeller inlet) (velocity units: m/s)

Figure 28 shows the contour of relative total pressure at the leading edge of the impeller blade versus time. The relative total pressure, $P_{0,\text{ref}}$, is the stagnation pressure computed using relative velocities instead of absolute velocities, which represents the loss of total pressure in the moving components. The relative total pressure at the impeller inlet does not vary with time. Hence, the IGV effect on the impeller is not time dependent, and it is mainly due to potential interaction. There is a large amount of total
pressure loss at the leading edge of the impeller blade close to the pressure side of the blade. This loss region extends to the leading edge of the splitter, which is located downstream of the leading edge of the impeller.

The relative tangential velocity, as shown in Figure 29, is defined by the following equation.

\[ C_{\theta,rel} = C_{\theta,i} - r_i \omega \]

where \( C_{\theta,i} \) is the tangential velocity at the cell, and \( r_i \) is the radius.
The flow tangential velocity is higher than the blade tangential velocity at the inlet. Thus, the flow is faster than the blade at the eye of the impeller. A positive incidence makes flow separate at the pressure side of the blade. Due to the high pressure field on the pressure side of blade, the flow is forced to reattach to the blade. Positive incidence causes chaos in the flow state at the inlet of the impeller.

The impeller with uniform inlet flow is also simulated using a single moving reference frame model. The flow is perpendicular to the boundary at the inlet. The velocity triangle is shown in Figure 30.
It is found that positive incidence still exists, which is about 10 degrees at the hub side and 34 degrees at the shroud side. The relative flow angle is a function of the blade speed and the meridional speed of the flow, which is controlled by the mass flow rate, density and cross-sectional area. A smaller mass flow rate leads to a larger relative flow angle.

The relative total pressure is taken at each meridional location, as shown in Figure 31. The flow separates at the leading edge of the impeller at meridional location $x / X = 0.3 / 1$. A small low energy flow region generates at the pressure side of the blade as a result of the separation. A low energy flow region is also found close to the shroud due to the boundary layer effect, so called secondary flow.

The blade angle does not match the flow angle at the impeller inlet, which causes the flow to separate at the pressure side of the impeller blades. It is necessary to discover the reason behind this mismatch, in order to improve the impeller efficiency. From Figure 30, smaller meridional velocity $C$ can lead to a larger relative flow angle, and smaller
incidence. CFD simulation defines the boundary conditions as fixed pressure boundaries and the mass flow rate is a result of the simulation. Comparing the mass flow rate from CFD simulation and from one-dimensional analysis, the mass flow rate calculated by one-dimensional analysis is much smaller than that from three-dimensional simulation. The original designed mass flow rate is 5 g/s, but the simulation result is 30 g/s. As discussed in the previous section, relative flow angle is related to the mass flow rate. As the mass flow rate goes down, the relative flow angle increases, and the incidence decreases. Hence, the reason of the separation is that the mass flow rate predicted by one-dimensional analysis does not match that predicted by numerical simulation.
Figure 31 Relative total pressure at different meridional locations (impeller-splitter, uniform flow, zero flow angle)
4.2.2 Inside of the impeller

After the flow has been brought on-board the impeller, either smoothly and efficiently or with disruptions, the flow then proceeds through the impeller. Regardless of the flow state (orderly or disorderly from inlet effects), the fluid receives a fundamental pressure rise that varies strongly with $\omega^2 r^2$. The Euler equation and definition of efficiency gives:

$$pr_n = \left[1 + \frac{\eta}{C_p T_0} (U_2 C_{\theta 2} - U_1 C_{\theta 1})\right]^{\gamma/(\gamma-1)}$$

Pressure rises with the impeller speed squared. This is the inescapable centrifugal pressure rise. The pressure rise equation is directly a function of the gas constant, efficiency, work input coefficient, and the Mach number squared. The gas constant varies from one fluid type to another. Simple molecules have very high values of the gas constant, whereas more complex molecules have much lower values.

Each passage of a centrifugal compressor impeller can be thought of, generally, as a rotating diffuser. To be sure, it is a very complex diffuser with all the influences of curvature and rotation present to complicate the process. However, it has been amply demonstrated that the treatment of the impeller as a diffuser is important if high performance is to be achieved. Along any particular operating speed line, both accelerating and diffusing characteristics can be found within an impeller flow passage as operation is shifted from high flow to low flow rates.
In this impeller, to control the mass flow rate, the number of impeller blades is limited to ten. Ten splitters are used to prevent boundary layer build-up. As shown in Figure 32, the loss between splitter and blade (1), at opposing rotating directions, is higher than that between blade and splitter (2).

![Figure 32 Relative total pressure (Impeller)](image)

Figure 32 and Figure 34 present the mass distribution and the relative velocity vector inside of the impeller, respectively. Flow separations are found at the leading edge of the impeller close to the pressure side, as a result of large incidence, and at the suction side of the splitter. The leading edge of the splitter is located in the region where the flow tends to reattach to the blade due to the high pressure on the pressure side of the impeller. Therefore, a negative incidence causes flow to separate at the suction side, which is
unfavorable. Passage 1 blockage increases because of the separation. More flow passes through passage 2.

Figure 33 Mass (Impeller)
The relative tangential velocity inside of passage 2 is lower than that of passage 1, as shown in Figure 35. Going back to Figure 34, it can be found that due to the separation and reattachment of the flow, the flow in passage 1 is moving toward the suction side of the blade, which is the direction of rotation.

Similar phenomenon can be found in Figure 31. At meridional distance $x/X = 0.5/1$, the low momentum region is found close to the suction side of the splitter. When the distance varies from $x/X = 0.5/1$ to $0.7/1$, the separation, the secondary flow, and the core flow mix gradually.
Based on above analysis, one can find that splitter deteriorates the impeller performance by causing the large separation. It is found that the impeller isentropic efficiency increases by 10% when a twenty blades impeller is used instead of a ten blades and ten splitters impeller.

4.2.3 Impeller exit

Once the flow leaves the impeller blade passage, complicated flow processes are still encountered. Since the flow field has diffused substantially, one must expect to find high aerodynamic blockage at the passage exit and substantial regions of low momentum fluid, as well as a high energy core flow.
In all flow processes, such strong variations will mix out with an entropy increase, that is, a total pressure loss (unless the stream tubes are deliberately separated). In such mixing processes there is also a static pressure rise, which is generally beneficial.

The exit mixing state is important as it influences the exit flow angle, the exit total pressure, and the exit static pressure. This leads to the final important process within the impeller, namely, flow angle deviation. Consequently, the exit flow condition of the impeller directly relates to the inlet of the diffuser.

Ideally, one would expect the flow to follow the direction of the impeller blading as it leaves the impeller blading. However, experience has shown that this never happens in any type of stationary or rotating cascade. Invariably, one finds a strong tendency for the primary passage flow to migrate from the pressure towards the suction surface due to the effect of blade unloading and is an essential process with any airfoil. It is brought about by force balancing. In a centrifugal compressor impeller, it is traditionally called slip, as it appears that the flow field has slipped against the direction of rotation, which is from pressure to suction side.

From Figure 36, large slip is found at the impeller exit because of unloading of the blade, which can cause large loss. However, the static pressure at the impeller exit increases as a result of mixing, as shown in Figure 37. Due to the thickness of the impeller blade and splitter, wakes are found at the end walls.
Figure 36 Velocity vector at the impeller exit

Figure 37 Mass-averaged static pressure in impeller
4.2.4 Tip leakage and secondary flow

Tip clearance is important for impeller performance. The leakage flow from the pressure side to the suction side of the blade, due to the pressure difference, interferes with the main flow. However, the effect of tip clearance on the overall performance is not simple.

In some early research, flow visualization studies showed that with a small clearance, the separation in the endwall-suction corner due to secondary flow was reduced. However, most studies, as discussed in the review, showed that tip leakage reduces the overall efficiency of the impeller.

The current numerical study of tip leakage considers a uniform tip leakage from the inlet to the outlet of the impeller. The clearance is about 0.00025 m, which is about 10% of the blade height at the exit of the impeller, and less than 3% of the blade height at the inlet of the impeller. The result is shown in Figure 38. Due to the geometrical complications, the original impeller geometry is simplified to 20 full length blades. The inlet condition is uniform and perpendicular to the boundary. The exit static pressure is 1.3 atm.

For comparison, simulation of the same geometry without tip leakage is carried out, and the results are presented in Figure 39.

Without tip leakage, the flow separates at the pressure side of the impeller blade, and a low momentum region is found. But the tip leakage flow eliminates the separation by pushing the low momentum flow from the pressure side to the suction side of the
blade. The tip leakage flow interacting with the secondary flow is clearly observed. The leakage flow, from pressure side to suction side of the blade, depresses the secondary flow to the bottom of the passage, and finally mixes together downstream. At meridional distance \( x/X = 0.6/1 \), the tip leakage flow is strengthened because the blade height reduces more. The tip leakage flow dominates the flow. There are more losses due to the friction and the mixing. It is found that the relative tangential velocity in the tip clearance is very high.
Figure 38 Relative total pressure (with tip leakage)
Figure 39 Relative total pressure (without tip leakage)
Without tip leakage, the separation flow caused by the incidence interacts with the secondary flow. The mixing process occurs faster than the mixing occurs in the flow with tip leakage. A low momentum region is found at the suction-hub side close to the end wall as a result of the motion of the separation flow.

Table 4 lists the isentropic efficiency of the impeller with and without tip leakage along the meridional direction, which is defined as

\[
\eta_{\text{isen}} = \frac{\Pr^{\gamma/(\gamma-1)} - 1}{\Tr - 1}
\]

Before \( x/X = 0.7/1 \), the impeller gives a better performance with tip leakage than without tip leakage. As discussed previously, it is because the separation is reduced by the tip leakage flow. The boundary layer is reduced, and the friction loss decreases. The ratio between tip clearance and blade height increases close to the end wall, which causes more tip leakage loss. Therefore, after \( x/X = 0.8/1 \), the impeller without tip leakage performs better than the one with tip leakage.

Table 4 Isentropic efficiency in meridional direction (with and without tip leakage)

<table>
<thead>
<tr>
<th>Distance ( x/X )</th>
<th>Without Tip Leakage</th>
<th>With Tip Leakage</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.5</td>
<td>44.48%</td>
<td>63.97%</td>
</tr>
<tr>
<td>0.7</td>
<td>74.40%</td>
<td>75.89%</td>
</tr>
<tr>
<td>0.8</td>
<td>81.47%</td>
<td>79.39%</td>
</tr>
<tr>
<td>0.85</td>
<td>83.49%</td>
<td>82.24%</td>
</tr>
<tr>
<td>0.9</td>
<td>84.01%</td>
<td>80.30%</td>
</tr>
</tbody>
</table>
The overall isentropic efficiency of the impeller with tip leakage is 90%, and without tip leakage is 99%. This result is high because the boundary condition at the exit of the impeller is static pressure, and the flow reverses at the exit. The turning bend is the main reason causing the separation, which is discussed in the following section. Regardless of the reverse flow at the exit, a difference of 10% impeller efficiency with and without tip leakage is found. This loss of isentropic efficiency is much higher than in conventional ones, which is around 4% for 10% of tip clearance. Since, the main reason of the tip leakage loss is due to the friction. Therefore, the tip leakage loss in small scale compressors is greater than that in large scale compressors.

In a multistage compressor, the precise effect of tip clearance becomes confusing because the performance of each blade row affects all those downstream. The tip leakage problem requires quite a large quantity of mesh to solve, which is beyond current available computational resources.
4.2.5 Loss analysis

The numerical results show that the turning bend loss dominates the impeller loss, as shown in Figure 40, along with friction and secondary flow loss. Due to the turning bend before the impeller, the secondary flow merges with low momentum flow to the shroud side of the impeller. The flow at the shroud side tends to separate. The fact that the impeller is a diffuser also enhances the separation. The low momentum flow at the shroud side encounters the positive static pressure gradient and separation is inevitable.

Figure 40 Total pressure (Friction study)

At the exit of the impeller, there is another 90 degree turning bend to the axial direction diffuser. The static pressure increases. The shroud side flow, which already
separates, turns to the outside of the bend, which makes the situation even worse. As one can see, the loss in the third bend of the compressor reaches the peak. Both inner and outer have a great deal of loss. The pressure loss due to the turning bend and the wall friction is 30.8% based on inlet energy level, which is defined as

\[ C_{P,\text{Loss,Impeller}} = \frac{P_{t,i} - P_{t,\infty}}{P_{t,i} - P_{s,i}}. \]

### 4.2.6 Summary

The impeller is a key component in the miniature centrifugal compressor. The current design under-predicts mass flow rate, which causes large incidence at the pressure side of the leading edge of the impeller. Through various numerical simulations, the following are found.

1. The impeller with 10 blades and 10 splitters does not perform as well as the 20 blades impeller.
2. The tip leakage loss for the miniature impeller has more impact on the impeller efficiency than that of a conventional large scale impeller. The tip leakage has a positive effect when separation and secondary flow exist. Therefore, the tip clearance needs to be carefully controlled.
3. The turning bend loss is significant in the impeller, which also results from the upstream and downstream turning bends. To improve the impeller efficiency, it is recommended to add a deswirler in front of the diffuser inlet.
4.3 Diffuser

4.3.1 Diffuser performance

The kinetic energy of the flow leaving the impeller of a centrifugal stage is equivalent to approximately 30% to 40% of the total work input under typical operating conditions. If an efficient stage is to be designed, this kinetic energy into static energy must be efficiently recovered. Compressor diffusers convert kinetic energy into a static pressure rise by one or both of two principle techniques:

1. An increase in flow passage area which brings about a reduction in the average velocity and, hence, an increase in static pressure.

2. A change in mean flow path radius which brings about a recovery in angular velocity according to the conservation of angular momentum principle,

\[ rC_\theta = \text{constant}. \]

Either of these techniques is responsible for the recovery of kinetic energy in common pump and compressor diffuser designs. There are various ways to affect this recovery, as described in the following sections. Most of these ways involve modes where separation or stall will be encountered, frequently including transitory stall or rotating stall, depending on the particular class of diffuser. Hence the designer is concerned not only with achieving high levels of pressure recovery, but also with maintaining stable operating conditions.
Cascade diffusers are one class of vaned diffusers. Compared with vaneless diffusers, vaned diffusers achieve a similar or better pressure recovery in a smaller diameter, and are therefore used where a vaneless diffuser would lead to a prohibitively large compressor or where the stage pressure ratio is too high to permit an efficient vaneless diffuser. In addition, a vaned diffuser permits control of the diffuser exit flow angle, which may be important for matching a downstream volute or return channel. On the other hand, the compressor operating range tends to be more restricted, and the complexity and cost of the stage is higher.

A cascade diffuser comprises one or more rows of airfoil-section vanes imposed on an annulus about the compressor axis. It takes its name from the fact that the vane design is based on airfoils tested in linear cascades. The final shape of the vanes is obtained by conformal transformation from the linear airfoil form.

The traditional perception of a vaned diffuser is that the cascade diffuser delivers a performance and range which is intermediate between the vaneless and the channel diffuser.

The designer selects a suitable rectilinear cascade profile and then transforms the airfoil into the axial plane. The profile selected depends on the stall margin required of the first row (or of the only row if it is a single-row cascade) as dictated by the operating conditions and the diffusion limits set by the cascade database as applied by the other rows. The loss across any given row depends on the camber angle $\phi$, the stagger angle $\gamma$, solidity $\sigma = c/s$ of the airfoil, the Mach number of the incoming flow field and the angle of attack.
Seleznev and Galerkin (1982) reported test results both for single-row cascade diffusers, with and without splitter vanes, and for a tandem cascade diffuser. The tandem cascade showed a higher efficiency than did a single-row cascade.

An axial direction diffuser is used due to the space limit in the radial direction. The first design of the diffuser failed to recover the static pressure from dynamic energy. It was found that the vane angle at the diffuser inlet did not match the flow angle. Based on CFD results, the new design has four rows of vanes, as shown in Figure 41.

![Figure 41 Comparison of two diffuser designs](image)

The diffuser efficiency increases from 8.6% to 74.8%. Performance of the IGV and the impeller with two diffusers is shown in Table 5. The efficiency of the IGV and the impeller does not change with the two diffusers. Therefore, the effect of the downstream component on performance of the upstream components is weak.

Figure 42 and Figure 43 present plots of tangential velocity at the impeller exit and diffuser inlet. The high tangential velocity at the diffuser inlet is coincident with the
high tangential velocity region in the impeller, but is lower. The loss due to the mixing at the end wall of the impeller and turning bend loss is high.

Figure 44 shows the mass-averaged static pressure in the diffuser. It can be seen that most static pressure recovery occurs in the first and second rows of vanes. Since the first and second rows are overlapped, it is hard to distinguish the effects of the two. The slight variation at 0.005 m indicates where the second row of vanes starts. The total pressure along the diffuser is shown in Figure 45. More than 75 percent of the total pressure loss is found between the first and second vane rows.

Table 5 Comparison of compressor performance with two diffusers

<table>
<thead>
<tr>
<th></th>
<th>Two-row diffuser</th>
<th>Four-row diffuser</th>
</tr>
</thead>
<tbody>
<tr>
<td>$P_{t,2}$ (atm)</td>
<td>1.564</td>
<td>1.590</td>
</tr>
<tr>
<td>$P_{s,2}$ (atm)</td>
<td>1.302</td>
<td>1.298</td>
</tr>
<tr>
<td>Mass flow rate (m/s)</td>
<td>15.97</td>
<td>15.40</td>
</tr>
<tr>
<td>IGV Efficiency</td>
<td>76.3%</td>
<td>77.9%</td>
</tr>
<tr>
<td>Impeller Efficiency</td>
<td>83.0%</td>
<td>81.0%</td>
</tr>
<tr>
<td>Diffuser Efficiency</td>
<td>8.6%</td>
<td>74.8%</td>
</tr>
</tbody>
</table>
Figure 42 Tangential velocity (impeller exit and diffuser inlet)

Figure 43 Tangential velocity (diffuser inlet)
Figure 44 Mass-averaged static pressure (diffuser)

Figure 45 Mass-averaged total pressure (diffuser)
Figure 46 Velocity magnitude with time (impeller exit to diffuser inlet)
Figure 46 shows time variation of the velocity at the impeller exit and the diffuser inlet, where $T$ represents a time period that two adjacent impeller blades pass a fix point. The load on the first row of vanes is transient. However, as shown in Figure 47, averaged static pressure recovered is steady. Therefore, transient interaction between the impeller and diffuser does not affect the steady performance of the diffuser. But the transient load on the diffuser vane may cause vibration problems.

The pressure loss coefficient of the diffuser is defined as

$$C_{P,loss,diff} = \frac{P_{i,\text{in}} - P_{i,\text{out}}}{P_{i,\text{in}} - P_{s,\text{in}}}. $$
From the friction study, the pressure loss coefficient of the diffuser is 19.6%, which includes turning bend loss and friction loss of the wall only. In this case, the loss due to the existing blade and non-uniform inlet are not considered. With the interaction between the impeller and diffuses, and the diffuser vanes, the loss coefficient is found to be 25.8%. Thus, about 75.8% of the diffuser loss is due to the turning bend loss and friction loss of the walls. To improve the diffuser performance, reducing the turning bend loss should be considered first.

The velocity magnitude at the diffuser inlet and outlet are 186 m/s and 14 m/s, respectively. About 95% of dynamics energy is recovered.

4.3.2 Summary

The old design of the diffuser does not match the flow from the impeller. A new design of the diffuser has a total pressure loss of 25.8%, and 75% of the total pressure loss is due to the turning bend loss and friction loss of the diffuser walls.

Most of the static pressure recovery and total pressure loss occurs in the first and second diffuser vane rows.

The vane loading of the first row of diffuser vanes is transient, but the averaged static pressure rise is steady.
4.4 Overall Performance of the compressor

4.4.1 Performance of the compressor

Figure 48 Performance chart (air)

The performance chart of the centrifugal compressor is shown in Figure 48, where 00 and 03 mean values at compressor inlet and exit respectively. Every data point is calculated with considerations of interaction. The efficiency of the compressor is defined as
\[ \eta_c = \frac{Pr^{\gamma/\gamma-1}}{Tr - 1}. \]

The CFD simulation varies the pressure ratio between inlet and outlet to produce the performance chart instead of controlling the mass flow rate, which experiments do. As it approaches the surge line, the performance line for the same speed becomes flat, and change of the pressure ratio becomes smaller. Therefore, it is difficult to find the exact surge line in the current numerical study. With this plot, one can find the best performance point for each rotating speed.

### 4.4.2 Dimensional analysis

Derived from Pi theory, the dimensional analysis can predict the performance of an existing compressor in different sizes or with different working media. There are geometrical similarities and dynamical similarities.

The following equation applies to compressible flow. For a different gas, to keep the parameters on the left hand side same, all parameters on the right hand side remain constant.

\[
\frac{P_{01}}{P_{00}}, \eta, \frac{\Delta T_{01}}{T_{00}} = f\left(\frac{m\sqrt{RT_{00}}}{D^2P_{00}\sqrt{\gamma}}, \frac{ND}{\sqrt{\gamma RT_{00}}}, Re, \gamma\right)
\]

where 1 and 2 indicate conditions at the compressor inlet and outlet, respectively.

However, it is only possible to keep the first and the second parameters on the right hand side of the equation the same. Reynolds number and specific heat vary with
different gases. Many researchers have found that the Reynolds number effect on dimensional analysis can be neglected.

Since the original design is for helium, the equation is applied from air to helium. The rotating speed at the design point for helium is found to be 313 KRPM. To make a comparison, neon is employed. Helium and neon have the same specific heat ratio.

Numerical results are plotted in Figure 49. The helium line and neon line are similar, and efficiencies of the compressor are the same for helium and neon at the same pressure ratios. However, the air line is a bit offset compared to the other two. The specific heat ratio for air is 1.4, and for helium and neon it is 1.67. Hence, the specific heat ratio effect needs to be considered when dimensional analysis applies.

![Figure 49 Dimensional analysis](image_url)
Comparing the velocity contour between air and helium, where the velocity contour is modified by a factor $f$, it is found that the two flow fields are similar, as shown in Figure 50 and Figure 51.

$$f = \frac{U_{\text{air}}}{U_{\text{helium}}}$$

Figure 52 plots the Reynolds number for air, helium and neon at different sections of the compressor. $Re/D$ is defined as $Re/D = \frac{DV}{\mu}$. D is same for all the gases. Since the Reynolds numbers for all three gases are different and yet the results shown in Figure 49 are very similar, it can be concluded that the Reynolds effect can be neglected in compressor performance study.
Figure 50 Velocity contour (helium 313K corrected)

Figure 51 Velocity contour (air 108K)
Figure 52 Comparison of Reynolds number
CHAPTER FIVE: CONCLUSION

The miniature centrifugal compressor is composed of three components, the radial IGV, radial impeller and axial diffuser. Fluid details are inaccessible by current available experimental methods. CFD simulation predicts performance of the miniature centrifugal compressor with consideration of the interaction between blade rows. In this study, loss mechanisms and dimensional analysis are studied numerically.

- It is found that the interaction between the IGV and impeller is due to potential interaction. The IGV loss coefficient increases 2% due to the interaction. Upstream components are rarely affected by the performance of downstream components. Wake interaction between the impeller and diffuser is time dependent. However, the performance of the diffuser is not a function of time. Therefore, the only concern of interaction is diffuser vane vibration due to the transient blade loading.

- Due to space limitations, the compressor has a unique geometry. It is found that for such geometry the turning bend loss is significant. It dominates most of the loss among the three components. The impeller, also functioning in part as a diffuser, enhances the turning bend loss. To improve the performance of a compressor with an axial diffuser, it is necessary to reduce the turning bend loss. How to reduce turning bend loss should be a future study topic.

- When the size goes down, precision loss and friction loss increase. A study of tip leakage shows there is 10% loss of efficiency with a 10% tip clearance, which is high compared to 4% loss in a conventional compressor. Well
controlled tip leakage can benefit compressor performance by reducing boundary layer thickness.

- Dimensional analysis is commonly used in turbomachinery design. Both Reynolds number and specific heat ratio effects are studied numerically. Reynolds number does not have an effect on similarity, and can be neglected. However, specific heat ratio has effects on both efficiency and shape of the performance line. It is recommended to apply the similarity law to gases with the same specific heat ratio.

There are a few design defects that have been found:

- The mass flow rate predicted by CFD simulation does not match the one predicted by one-dimensional analysis, which causes mismatch of flow angle at the leading edge of the impeller. The results need to be validated by experimental measurement.

- It is found that a 20-blade impeller has better performance than a 10-blade-splitter impeller.

- The newly designed four-row diffuser performs much better than the previous two row diffuser. But most of static pressure is recovered in the first two rows, and the 4th row is dominated by separation.

- To reduce impeller turning bend loss, it is recommended to deswirl the flow in front of the bend.
**APPENDIX: AIR, NEON, AND HELIUM PROPERTIES**

<table>
<thead>
<tr>
<th>Property</th>
<th>Air</th>
<th>Neon</th>
<th>Helium</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density ($kg/m^3$) (Pressure 1~1.7 atm, T=300K)</td>
<td>1.178~2.003</td>
<td>0.082~1.715</td>
<td>0.162~0.259</td>
</tr>
<tr>
<td>Dynamic viscosity ($Pa\cdot s$)</td>
<td>1.789X10^{-5}</td>
<td>3.123X10^{-5}</td>
<td>1.932X10^{-5}</td>
</tr>
<tr>
<td>Specific heat constant pressure ($J/gK$)</td>
<td>1.006</td>
<td>1.030</td>
<td>5.193</td>
</tr>
<tr>
<td>Thermal conductivity ($W/mK$)</td>
<td>0.024</td>
<td>0.048</td>
<td>0.156</td>
</tr>
<tr>
<td>Specific heat ratio</td>
<td>1.40</td>
<td>1.67</td>
<td>1.67</td>
</tr>
</tbody>
</table>
LIST OF REFERENCES


111


