A Computational Study of the 3D Flow and Performance of a Vaned Radial Diffuser

by

Dikran Akseraylian

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

Master of Science

in

Mechanical Engineering

APPROVED:

[Signatures]

Dr. John Moore, Chairman

Dr. C. L. Dancey

Dr. W. F. Ng

August, 1996
Blacksburg, Virginia

Key Words: Diffuser, Vaned, CFD
LD
5655
V255
1992
A374
c.2
A COMPUTATIONAL STUDY OF THE 3D FLOW AND PERFORMANCE OF A VANED RADIAL DIFFUSER

by

Dikran Akseraylian

Dr. John Moore, Chairman

Mechanical Engineering

(ABSTRACT)

A computational study was performed on a vaned radial diffuser using the MEFP (The Moore Elliptic Flow Program) flow code. The vaned diffuser studied by Dalbert et al. was chosen as a test case for this thesis. The geometry and inlet conditions were established from this study. The performance of the computational diffuser was compared to the test case diffuser. The CFD analysis was able to demonstrate the 3D flow within the diffuser.

An inlet conditions analysis was performed to establish the boundary conditions at the diffuser inlet. The given inlet flow angles were reduced in order to match the specified mass flow rate. The inlet static pressure was held constant over the height of the diffuser.

The diffuser was broken down into its subcomponents to study the effects of each component on the overall performance of the diffuser. The diffuser inlet region, which comprises the vaneless and semi-vaneless spaces, contains the greatest losses, 56%, but the highest static pressure rise, 54%. The performance at the throat was also evaluated and the blockage and pressure recovery were calculated.

The results show the static pressure comparison for the computational study and
the test case. The overall pressure rise of the computational study was in good agreement with the measured pressure rise. The static pressure and total pressure loss distributions in the inlet region, at the throat, and in the exit region of the diffuser were also analyzed. The flow development was presented for the entire diffuser. The 3D flow calculations were able to illustrate a leading edge recirculation at the hub, caused by an inlet skew and high losses at the hub, and the secondary flows in the diffuser convected the high losses.

The study presented in this thesis demonstrated the flow development in a vaned diffuser and its subcomponents. The performance was evaluated by calculating the static pressure rise, total pressure losses, and throat blockage. It also demonstrated current CFD capabilities for diffusers using steady 3D flow analysis.
ACKNOWLEDGEMENTS

I would like to thank Dr. Moore and Joan Moore for their guidance in carrying out my research and preparing this thesis. I would also like to thank them for giving me the opportunity to work under their supervision and on a project which interested me a great deal. This thesis would not have been possible without the wisdom and patience provided by Dr. Moore. Joan helped me in understanding CFD and how to use the workstation and numerous other computational tasks. I would also like to thank Dr. Dancey and Dr. Ng in taking time out of their busy schedules to be a part of my advisory committee. Finally, I would like to thank my family and friends for their support during my graduate studies.
# TABLE OF CONTENTS

1.0 INTRODUCTION .................................................. 1

2.0 LITERATURE REVIEW .............................................. 4

2.1 Centrifugal Compressors ........................................ 4

2.2 Types of Diffusers .............................................. 7

2.3 Impeller Effects .................................................. 12

2.4 Effects of Vaned Diffuser Geometry ............................. 19

2.5 Diffuser Subcomponents ........................................ 22

2.5.1 The Semi-Vaneless Space .................................... 27

2.6 The Moore Elliptic Flow Program ............................... 28

3.0 OBJECTIVES OF THESIS ......................................... 30

4.0 DIFFUSER VANE GEOMETRY ..................................... 31

4.1 Diffuser Test Case ............................................... 31

4.2 Grid Generation ................................................... 37

5.0 INLET FLOW CONDITIONS ....................................... 46

5.1 Inlet Conditions Analysis ....................................... 46

5.2 Flow Angle Correction .......................................... 51

5.3 Computational Boundary Conditions ............................ 59

6.0 RESULTS AND DISCUSSION .................................... 63

6.1 Static Pressure Comparison ..................................... 63

6.1.1 Pressure Rise Comparison .................................. 63

6.2 The Inlet Region ................................................ 68
6.3 The Diffuser Throat

6.3.1 Static Pressure Analysis
6.3.2 Total Pressure Losses
6.3.3 Throat Blockage

6.4 Ideal and Possible Pressure Recovery Coefficients

6.5 The Exit Region

6.5.1 Static Pressure Analysis
6.5.2 Total Pressure Losses

6.6 Secondary Flows

6.7 Flow Development

6.7.1 Total Pressure Loss Development
6.7.2 Mass Averaged Total Pressure Losses
6.7.3 Summary of the Flow Development

7.0 CONCLUSIONS

BIBLIOGRAPHY

APPENDIX A - Program which solves for cartesian coordinates for the computational diffuser vane (MathCAD Program)

APPENDIX B - Program which initializes the inlet boundary conditions for the computational diffuser (FORTRAN Program)

VITA
LIST OF ILLUSTRATIONS

Figure 1.1  Nomenclature for an impeller and diffuser

Figure 2.1  Pressure rise and stability characteristics of a compressor stage and its components

Figure 2.2  Photograph of types of diffusers

Figure 2.3  Nomenclature for a vaned diffuser

Figure 2.4  Pipe diffuser

Figure 2.5  Impeller exit flow distortion

Figure 2.6  Impeller exit velocity triangles

Figure 2.7  Performance characteristics of diffusers with different leading edge radius ratios

Figure 2.8(a)  Radial velocity profile at $r/r_2 = 1.05$

Figure 2.8(b)  Radial velocity profile at $r/r_2 = 1.16$

Figure 2.9  Effects of number of diffuser vanes on total pressure losses

Figure 2.10  Nomenclature for the diffuser and its subcomponents

Figure 2.11  Pressure rise characteristics for 15, 25, and 30 degree diffusers

Figure 2.12  Stability effects of the 25 degree diffuser

Figure 4.1  Photograph of the impeller and 25 degree vaned radial diffuser

Figure 4.2  Cross-section of the centrifugal compressor stage

Figure 4.3  25 degree diffuser with its geometric specifications

Figure 4.4  Performance map of the stage

Figure 4.5  Sketch of the diffuser vane in x-y coordinate space
Figure 4.6 Sketch which illustrates the method for determining the r-θ coordinates of the diffuser vane

Figure 4.7 Computational grid

Figure 4.8 Computational grid doubled in the θ direction

Figure 4.9 Computational grid viewed in the θ direction

Figure 5.1 Inlet boundary conditions

Figure 5.2 Effects of changes in flow angle on mass flow rate

Figure 5.3 Comparison of the given inlet flow angles and the corrected flow angles

Figure 5.4 Comparison of the radial velocity profiles with given flow angles and corrected flow angles

Figure 5.5 Comparison of the tangential velocity profiles with given flow angles and corrected flow angles

Figure 5.6 Inlet total temperature distribution

Figure 5.7 Imposed radial velocity profile for calculations

Figure 6.1(a) Measured static pressure distribution for the test case diffuser

Figure 6.1(b) Calculated static pressure distribution for the computational diffuser

Figure 6.2 Comparison of the calculated static pressure rise to the given data

Figure 6.3 Velocity vectors for inlet to vane leading edge, near suction surface

Figure 6.4 Velocity vectors for inlet to vane leading edge, near pressure surface

Figure 6.5 Velocity vectors for inlet to vane leading edge, mid-passage

Figure 6.6 Velocity vectors on hub, mid-height, and shroud surfaces

Figure 6.7 Static pressure coefficients on throat plane
Figure 6.8  Relationship between flow angle, incidence, and static pressure coefficient
Figure 6.9  Total pressure losses on throat plane
Figure 6.10(a)  Throat blockage for the given data
Figure 6.10(b)  Throat blockage for the calculations
Figure 6.10(c)  Vaned diffuser pressure recovery coefficients
Figure 6.11  Static pressure coefficients on exit plane
Figure 6.12  Total pressure losses on exit plane
Figure 6.13  Secondary flows at inlet and leading edge of diffuser vane
Figure 6.14  Sketch of secondary flow mechanisms
Figure 6.15(a)  Secondary velocity vectors on throat plane
Figure 6.15(b)  Sketch of secondary flow patterns at the throat
Figure 6.16  Secondary velocity vectors on exit plane
Figure 6.17  Total pressure losses on hub, mid-height, and shroud surfaces
Figure 6.18  Total pressure losses on planes at I = 1, 4
Figure 6.19  Total pressure losses on planes at I = 12, 18
Figure 6.20  Total pressure losses on planes at I = 22, 26
Figure 6.21  Total pressure losses on planes at I = 33, 41
Figure 6.22  Mass averaged total pressure losses
Figure 6.23  Velocity vectors from $r/r_2 = 1.05$ to $r/r_2 = 2.00$ for blade suction surface and pressure surface
Figure 6.24  Velocity vectors from $r/r_2 = 1.05$ to $r/r_2 = 2.00$ on the hub wall
Figure 6.25  Velocity vectors from $r/r_2 = 1.05$ to $r/r_2 = 2.00$ on the shroud wall
LIST OF TABLES

Table 4.1  Geometric data of diffusers

Table 5.1  Final calculations of the diffuser inlet data

Table 5.2  Comparison of the inlet velocity with given flow angles and corrected flow angles

Table 5.3  Computational inlet boundary conditions

Table 6.1  Static pressure rise for the diffuser and its subcomponents

Table 6.2  Pressure recovery and throat blockage for the given data and calculations
NOMENCLATURE

A = geometric area

A_{\text{eff}} = effective area

AR = area ratio

Bl = blockage

b_2 = diffuser width

C_{ps} = static pressure coefficient, Equation 6.3.1

C_{pt} = total pressure loss coefficient, Equation 6.3.2

C_p = pressure recovery coefficient, Equation 2.1.1

c_p = specific heat capacity

D_p = pressure rise coefficient, Equation 2.5.2

d = diameter

h = diffuser channel height

i = incidence

k = isentropic coefficient

l = diffuser channel length

m = mass flow rate

M_u = tip speed Mach number, Equation 2.5.1

P = static pressure

P_t = total pressure

P_{t_{\text{ref}}} = maximum stagnation pressure at diffuser inlet, Section 5.3

R = gas constant or radius of curvature
\( r_k \) = radius of curvature

\( r_2 \) = radius of impeller outlet

\( T \) = static temperature

\( T_t \) = total temperature

\( T_{t2ref} \) = stagnation temperature which corresponds to the maximum stagnation pressure, \( P_{t2ref} \), at the diffuser inlet, Section 5.3

\( U \) = circumferential speed

\( V \) = absolute velocity

\( V \) = volumetric flow rate

\( W \) = relative velocity

\( \alpha \) = flow angle

\( \phi \) = flow coefficient

\( 2\theta \) = divergence angle

\( \rho \) = density

\( \rho_{2ref} \) = density evaluated at \( P_{t2ref} \) and \( T_{t2ref} \) at the diffuser inlet, Section 5.3

**Subscripts**

1 = stage inlet

2 = diffuser inlet

3 = diffuser exit

C = diffuser throat plane

D = diffuser exit plane
\( r = \text{radial} \)

\( \theta = \text{tangential} \)
1.0 INTRODUCTION

Centrifugal compressors are widely used in applications such as power plants, automobiles, and jet engines. It is a common practice to combine the centrifugal impeller with a vaned diffuser to achieve the optimal performance at a particular flow rate. Figure 1.1 illustrates an impeller with a vaned radial diffuser.

There are numerous studies concerning vaned diffusers and their governing flow physics. A successful design procedure for vaned diffusers is very difficult to find and maybe has never been produced. Prior studies usually are limited to comparisons of performance characteristics and diffuser geometries. The performance of a diffuser is largely affected by the impeller exit flow and its levels of unsteadiness as seen by the diffuser. The shape of the diffuser vane along with its leading edge configuration, number of vanes, and size of the vaneless space play a role when studying diffusers. There is a need for a well documented study which gives a clear and fast procedure in developing diffusers.

Many diffuser studies are experimental and often “vortex rigs” are used to simulate the exit flow of an impeller, this was done by Baghdadi and McDonald (1). Experimental studies tend to concentrate on overall performance and do not give a detailed assessment of the diffuser flow field, therefore, the use of 3D flow calculations is needed. A few people have utilized Computational Fluid Dynamics (CFD) in vaned diffuser studies. Two examples are Casey et al. (2), Dalbert et al. (3), who carried out their studies using Dawes’ (4) 3D flow code, and Kirtley and Beach (5). CFD is becoming an important tool in the development of vaned diffusers and other turbomachinery components. It
Figure 1.1 Centrifugal compressor components and velocity triangles (Reference 23)
sufficiently reduces the time involved for testing and development. Since CFD is being used by many manufacturers, there is a need for more, well documented, computational studies.

The present work utilizes CFD in studying a vaned radial diffuser. The primary aim of the study was to provide a comprehensive investigation of the 3D flow and performance. Static pressure recovery, throat blockage, total pressure losses, and incidence effects were studied. Descriptions of the flow field were also developed. The diffuser was broken down into its subcomponents. The entry, diffuser passage, and exit zone were individually investigated.

In order to perform the study in this thesis, a diffuser test case had to be found. The study by Dalbert et al. (3) was chosen for comparison. The geometry and inlet conditions provided by Dalbert et al. were used to set up the computational study of the diffuser. Discrepancies in the documented test case were revealed and the study was performed with very reasonable results.
2.0 LITERATURE REVIEW

2.1 Centrifugal Compressor Diffusers

The primary purpose of a diffuser is to decelerate the high velocity fluid leaving the impeller and to convert this inlet kinetic energy into static pressure. The impeller, typically, has efficiencies on the order of 95%, but with the addition of the diffuser the stage efficiency is around 80%. Therefore, the limiting element to achieving higher efficiencies is the diffuser (6).

When discussing diffusers, it is common to speak in terms of pressure recovery, defined as

$$C_p = \frac{(P_3-P_2)}{(P_{c2}-P_2)}$$  \hspace{1cm} (2.1.1)

where station 2 is at the diffuser inlet and station 3 is at the diffuser exit. Typically, the tangential component of velocity is around three times greater than the radial component, therefore, it is this component which gives most of the pressure rise (7). Pressure recovery is the governing parameter because this defines how much diffusion actually occurs. A large pressure recovery within a small radial distance is desired for a small and lighter weight engine.

Figure 2.1 shows the pressure rise and stability characteristics of the centrifugal compressor studied in this thesis. This typical compressor data is provided by Hunziker and Gyarmathy (19) and Dalbert et al. (3). In this compressor, the diffuser has a lower pressure rise than the impeller, which reduces the overall pressure rise by the stage. This stage is therefore designed with a reaction greater than 50%, i.e. the static pressure rise is more than half the total pressure rise of the impeller, thus, the static pressure rise occurs
Figure 2.1 Pressure rise and stability characteristics of the components (Reference 19)
mostly in the impeller. In centrifugal machines, the performance of the impeller is better than that of the diffuser because the centrifugal effects occur isentropically. The actual diffusion of the relative kinetic energy in the impeller and the absolute kinetic energy in the diffuser are about the same in terms of pressure recovery coefficient or efficiency. Figure 2.1 also shows that, at high flow rates, the diffuser is the most stabilizing component of the compressor. Stability effects will be discussed in Section 2.5.

In most diffuser studies, the throat area is the region of highest interest. Since the throat has the smallest area, blockage is important in this region. Throat blockage is defined as:

\[ B_{\text{th}} = 1 - \frac{A_{\text{eff}}}{A_{\text{th}}}. \]  

As throat blockage increases, the losses increase downstream of the throat and separation occurs from the blade walls. As throat blockage increases the pressure rise will decrease. The leading edge configuration affects the flow as it travels within the diffuser thus affecting the boundary layer growth on the blade walls.
2.2 Types of Diffusers

Diffusers fall into two main categories, vaneless and vaned. The vaneless diffuser does not achieve as high a pressure recovery as the vaned diffuser but does operate over a larger flow range. The vaneless diffuser is a very simple component but its lack of vanes causes a nonuniform flow entering the diffuser, and the flow direction will not be constant across the width of the diffuser (7). The vaneless diffuser has been extensively studied and is a common turbomachinery component due to its simplicity in design.

There are many types of vaneless diffusers (Figure 2.2). The nomenclature generally used for this type of diffuser is shown in Figure 2.3. Vaned diffusers can give higher pressure recoveries but over a narrower range of flows. A common type of diffuser is a vane-island diffuser, which is referred to as a channel diffuser in Figure 2.2, consisting of wedge shaped vanes as opposed to thin blades. Jiang and Yang (8) studied vane-island diffusers at high swirl and found that vane-islands with straight channels produce higher pressure recovery than with curved diffuser passages. Another type of vaned diffuser is the pipe diffuser, shown in Figures 2.2 and 2.4. It has a ring of metal surrounding the impeller outlet consisting of cylindrical and conical passages which allow the flow to enter the diffuser more smoothly. The pipe diffuser was developed by Pratt and Whitney of Canada and has resulted in increased stage efficiencies because it is less sensitive to high Mach numbers (7). The tandem diffuser, shown in Figure 2.2, consisting of two rows of vanes has demonstrated higher efficiencies and pressure recoveries in an operating range larger than that of a traditional vaned diffuser. Senoo et.al. (9) studied low-solidity
Figure 2.3 Nomenclature and configuration of a vaned diffuser (Reference 11)
Figure 2.4 The pipe diffuser (Reference 23)
tandem cascade diffusers and found that this type of diffuser is capable of a high pressure recovery over a wider flow range than a conventional vaned diffuser. The problem that may occur with tandem diffusers is that they are relatively complex in design.
2.3 Impeller Effects

The performance of the individual components of the compressor is important, but even more important is the interaction of the impeller and the diffuser. The flow leaving the impeller is often referred to as a jet-wake phenomenon (Figure 2.5). The two flows can have different angles, radial and tangential velocities entering the diffuser. Many people have studied the interaction between the impeller and the diffuser, and there are many theories concerning the mechanics of the flow leaving the impeller.

The jet and wake flows mix out quite rapidly in the circumferential direction, but mixing is not as apparent in the axial direction. Since the non-uniform flow remains in the axial direction, it is the main contributor to the diffuser inlet distortion. Yoshinaga et al. (10) concluded that the circumferential velocity was fairly uniform across the width of the diffuser, but the radial component of velocity was the main contributor to the inlet distortion (Figure 2.6). They found that diffusers with half-guide vanes gave a more uniform flow distribution in the axial direction with an increased pressure recovery.

The vaneless space, the radial distance between the impeller and diffuser vanes, is an area of concern. This region has similar tendencies to a vaneless diffuser and has a significant effect on diffuser performance. Arndt et al. (11) found that by increasing the radial gap the diffuser inlet distortions decreased by 50%. The lengthened radial distance reduces vibration that is seen by the diffuser. Rodgers (12) also studied the effects of the vaneless space, with leading edge radius ratios ranging from 1.035 to 1.215. He found that a radius of 1.125 gave the highest efficiency (Figure 2.7). Gyarmathy et al. (13) studied the effects of leading edge radius ratios. They studied the time dependent radial
Figure 2.5 Impeller exit flow distortion (Reference 7)
Figure 2.6 Impeller exit velocity triangles (Reference 10)
Figure 2.7 Effects of vaneless space, leading edge radius ratios (Reference 12)
velocity at ratios of 1.05 and 1.16. It is evident by Figure 2.8 that a ratio of 1.05 has a much more circumferentially distorted profile than at 1.16. Inoue and Cumpsty (14) determined that at a ratio of 1.04 a reversal of flow back into the impeller occurred, since the diffuser vanes were close to the impeller. Jiang and Yang (8) studied diffuser vanes with leading edge radius ratios from 1.05 to 1.20. They determined that a ratio between 1.15 and 1.20 produced the lowest losses, and at 1.15 good mixing was achieved.

It is still uncertain whether or not the flow leaving the impeller is seen as an unsteady or steady effect by the diffuser. There is no doubt that the flow leaving the impeller is unsteady. There is doubt, though, as to where the flow becomes steady and at what leading edge radius ratio is the flow steady enough to support the use of a steady flow calculation to produce reasonable results. Inoue and Cumpsty (14) did an experimental study of centrifugal impeller discharge flow in vaneless and vaned diffusers, and found that the nonuniform flow from the impeller is perceived as an unsteady flow by the diffuser vanes and, in turn, the diffuser vanes are seen as an unsteady disturbance by the impeller. Dawes (15) performed a simulation of the unsteady interaction of a centrifugal impeller with a vaned diffuser and found that little loss could be attributed to unsteady effects. He concluded that the high losses were due to the axial distortion in swirl angle resulting in a hub - corner stall. He also concluded that the hub to shroud nonuniformity was more important than the blade to blade nonuniformity. Arndt et al. (11) performed an experimental investigation of the rotor - stator interaction in a centrifugal pump with several vaned diffusers. They found that the flow from the impeller exit to the diffuser entrance region becomes steady if the vaneless space between the
Figure 2.8(a) Radial velocity profile at \( r/r_2 = 1.05 \) (Reference 13)

Figure 2.8(b) Radial velocity profile at \( r/r_2 = 1.16 \) (Reference 13)
impeller and diffuser is large enough because the flow mixes out rapidly. In most cases there is likely to be some irregularity in the circumferential direction, but the levels of unsteadiness are decreased when the leading edge of the diffuser vane is far enough away from the impeller.
2.4 Effects of Vaned Diffuser Geometry

The shape, vane angle, and number of vanes are important factors in the diffuser performance. Numerous studies have been performed on the effects of the vane shape, and particularly, the leading edge shape. The leading edge has a significant influence on the flow within the diffuser passage. It is desired to design the diffuser vanes so that incidence does not hinder the flow into the diffuser passage and cause blockage at the throat. When the incidence angle is negative the flow is accelerated, and when the incidence angle is positive the flow is decelerated. A sketch of the incidence angle and flow angle are shown on Figure 2.3. The flow angle is defined as:

\[ \alpha = \tan^{-1}(V_r/V_0) \]  

(2.4.1)

The incidence is defined as:

\[ i = \alpha_{\text{blade}} - \alpha_{\text{flow}} \]  

(2.4.2)

Many people have studied the effects of vane geometry on the diffuser performance. In 1971, Dean (16) had observed that shaping the suction surface did not have much effect, and that the vane setting angle could be altered by several degrees with little change in performance. Later, in 1989, Clements and Artt (17) studied the influence of vane leading edge geometries on the performance of a centrifugal compressor and found an opposite conclusion to that of Dean. They determined that over most of the useful flow range the diffuser flow angle will produce negative incidence on the pressure surface and positive incidence on the suction surface, therefore, the diffuser performance was dominated by the suction surface profile. Conrad et al. (18) concluded that only positive incidence angles affect the blockage factor at the diffuser throat.
Inoue and Cumpsty (14) studied a vaned diffuser with 10, 20, and 30 vanes and found that increasing the number of vanes reduced the strength of the reversed flow at the vane leading edge. Jiang and Yang (8) also found a beneficial effect by increasing the vane number. They studied diffusers with 8, 14, and 22 vanes at different flow rates and found that increasing the number of vanes reduced the amount of pressure loss in the diffuser (Figure 2.9).
Figure 2.9 A comparison of the total pressure loss coefficient with 8, 14, and 22 diffuser vanes (Reference 8)
2.5 Diffuser Subcomponents

The vaned diffuser is divided into three subcomponents. An illustration and the nomenclature used can be seen in Figure 2.10. The entry (2 - C), which includes both the vaneless and semi-vaneless space, extends near the impeller exit (2) to the throat (C). The second subcomponent, the diffuser channel (C - D), extends from the throat (C) to a perpendicular plane at the end of the vaned passage or the exit plane (D). The third subcomponent, the exit area (D - 3), is the vaneless space downstream of the channel from the exit plane (D) to a surface of constant radius ratio (3).

Hunziker and Gyarmathy (19) studied the stability of a centrifugal compressor and its dependence on the compressor subcomponents and, more specifically, the diffuser subcomponents. Three diffusers were tested with inlet vane angles of 15, 25, and 30 degrees. They were also analyzed at three Mach numbers 0.6, 0.75, and 0.9, defined as:

\[
Mu_2 = \frac{U_2}{(kRT_1)^{1/2}}
\]  

(2.5.1)

The subcomponents have different effects on the diffuser performance. When studied individually, it is possible to see the magnitude of pressure rise produced by each area (Figure 2.11), and if they have a stabilizing or destabilizing effect (Figure 2.12). The region which receives the most attention and discussion is the semi-vaneless space and will be discussed in further detail in the following section.

The subcomponents are related in terms of static pressure rise, defined as:

\[
Dp = \Delta p/0.5\rho_1U_2^2
\]  

(2.5.2)

Figure 2.11 shows that the entry produces the highest amount of pressure rise but it decreases with increasing flowrate. The diffuser channel has an opposite trend and its
Figure 2.10 Illustration and nomenclature of vaned diffuser subcomponents

2 - C: entry
C - D: diffuser passage
D - 3: exit area
2 - 3: diffuser
Figure 2.11 Pressure rise characteristics of 15, 25, and 30 degree diffusers and its subcomponents at different flow coefficients (Reference 19)
Figure 2.12 Stability effects of the diffuser subcomponents of the 25 degree diffuser (Reference 19)
pressure rise increases with an increasing flowrate. There is relatively little static pressure rise in the exit region. Similar trends exist in all three figures. It is interesting to note that the 15 degree diffuser has a much narrower flow range, and the pressure rise falls drastically between flowrates of 0.065 to 0.07.

Greitzer (20) discusses different types of flow instabilities for pumps and compressors. He provides criteria for static and dynamic stability and describes the dynamic stability limit as occurring at the peak of the static pressure variation with flow rate. Hunziker and Gyarmathy similarly express the stabilizing effects in terms of slope of the curve of pressure rise versus flow rate. A component which has a positively sloped characteristic is destabilizing and a negatively sloped characteristic is stabilizing. The flow coefficient is defined as

\[ \phi = \frac{V}{d_2^2 U_2} \]  

(2.5.3)

where \( V \) is the volumetric flow rate. Figure 2.12 shows that the entry has a greater stabilizing effect than the other two subcomponents. The diffuser channel is the destabilizing subcomponent but becomes more stabilizing with increasing flowrates. The exit region is slightly destabilizing. If pressure rise increases with a decreasing flow rate the system is considered to be stable. The entry has a higher pressure rise at lower flow rates, therefore, a stabilizing component. The diffuser channel has a lower pressure rise at lower flow rates, therefore, a destabilizing component. The concept of stability provides a way to study the diffuser subcomponents and shows which areas require greater attention.
2.5.1 The Semi-Vaneless Space

The semi-vaneless space is the triangular region within the diffuser extending from the leading edge to the throat and from the pressure side to the suction side (Figure 2.10). The interaction of the impeller exit flow and the leading edge of the diffuser vane causes a disturbance in the flow which affects this region. The semi-vaneless space is often studied and is an area of great interest.

Numerous people have researched this area and have similar results. Hunziker and Gyarmathy found that the semi-vaneless space contains the highest pressure rise of any component; it is also the most stabilizing region of the diffuser. According to Agrawal et al. (21), as much as 65% of the pressure recovery occurs in this region. Baghdadi and McDonald (1) also studied stability effects in a diffuser and found that surge is an instability caused by flow separation in the semi-vaneless region. This results in high losses in the diffuser causing an unsteady interaction between the diffuser and its exhaust receiver. Dawes (15) found that strong pressure gradients and backflow are also evident due to a circumferential variation of swirl angle from 10 to 15 degrees and an axial variation from hub to shroud of 20 to 25 degrees. Dalbert et al. (3) studied the flow phenomena in a vaned diffuser and their results were comparable to those of Dawes. With the aid of flow visualization by using moving pictures, Dalbert et al. concluded that the strongest pressure gradients and backflow occur in this region and that the semi-vaneless space is the most critical area in the vaned diffuser.
2.6 The Moore Elliptic Flow Program

In this thesis, the Moore Elliptic Flow Program or MEFP was used to carry out the flow calculations in a vaned diffuser. This was the first application of MEFP in a vaned diffuser of a centrifugal machine. MEFP uses a three dimensional pressure correction solution procedure which solves the Navier Stokes equations. It uses a Prandtl mixing length turbulence and a vorticity based search procedure for local boundary layer widths. The program was developed for steady flows in rotating and stationary turbomachinery blade rows and is used in other applications such as inducers, turbines, and compressors. The solution procedure and equations involved are described by J.G. Moore (22).

The continuity and momentum equations are defined as:

Mass:

\[ \nabla \cdot \rho \mathbf{W} = 0 \quad (2.7.1) \]

Momentum:

\[ \rho \mathbf{W} \cdot \nabla \mathbf{W} - (\nabla \cdot \mu \mathbf{W}) \mathbf{W} = \nabla \cdot \mu \mathbf{W}^T - \nabla p - 2\rho \mathbf{\Omega} \times \mathbf{W} - \rho \mathbf{\Omega} \times (\mathbf{\Omega} \times \mathbf{r}) \quad (2.7.2) \]

Turbulence Model:

\[ \mu = \mu_{\text{laminar}} + \mu_{\text{turbulent}} \quad (2.7.3) \]

\[ \mu_{\text{turbulent}} = \rho L^2 \left( \frac{du}{dy} \right) \quad (2.7.4) \]

where \( L \) is the smaller of:

- 0.08 times the width of the shear or boundary layer
- 0.41y where \( y \) is the distance to the nearest wall

The van Driest equation is described as:
\[ L = 0.41y[1 - \exp(-y(\rho \tau)^{1/2}/26\mu_{\text{laminar}})] \quad (2.7.5) \]

The near wall correction is described as:

\[ \mu = (\mu_{\text{laminar}})^{1/2}(\mu_{\text{laminar}} + \mu_{\text{turbulent}})^{1/2} \quad (2.7.6) \]

Other equations which define the flow are described as:

Equation of State:

\[ p = \rho RT \quad (2.7.7) \]

Rothalpy:

\[ \rho \mathbf{W} \cdot \nabla \mathbf{H} - (\nabla \cdot \mu \nabla) \mathbf{H} = 0 \quad (2.7.8) \]

\[ \mathbf{H} = c_p T + \mathbf{W}^2/2 - \omega^2 r^2/2 \quad (2.7.9) \]

Second Law of Thermodynamics:

\[ s - s_0 = c_p \ln(T/T_0) - R \ln(P/P_0) \quad (2.7.10) \]
3.0 OBJECTIVES OF THESIS

Centrifugal machines are widely used and with the addition of a vaned diffuser the overall efficiency is improved over a narrower flow range relative to the efficiency with a vaneless diffuser. It is, therefore, understandable to study vaned diffusers in greater detail. A comprehensive literature review was performed to understand the trends and theories involved in vaned diffusers. It was also evident that computational studies of vaned diffusers are scarce.

The objectives of this thesis were to perform a computational study of a vaned diffuser using a 3D steady flow calculation procedure (MEFP), and to thoroughly investigate the flow development in the diffuser. The vaned diffuser studied by Dalbert et al. (3) was chosen as a test case. The computational vaned diffuser studied in this thesis was generated from the specifications given by Dalbert et al.
4.0 DIFFUSER VANE GEOMETRY

4.1 Diffuser Test Case

In order to do a computational study of a vaned diffuser, a suitable test case had be
found for a proper comparison. After performing an extensive literature review, the
studies completed by Dalbert et al. (3), Hunziker and Gyarmathy (19), and Gyarmathy et
al. (13) were chosen. The combination of these papers gave an adequate geometric
description for a vaned diffuser with inlet conditions and performance characteristics.
Essentially, the authors provided the necessary information to set up the diffuser and
provided data for comparison.

The diffuser that was chosen was a vaned radial diffuser with 24 circular arc
vanes, an inlet blade angle of 25 degrees and operating at design conditions with a tip
speed Mach number of 0.75 (Figure 4.1). A cross-sectional view of the centrifugal
compressor stage is shown in Figure 4.2, and a sketch of the vaned diffuser with its
specifications is shown in Figure 4.3. The leading edge radius ratio is at \( r_{1e}/r_2 = 1.15 \) and
the trailing edge radius ratio is at \( r_{te}/r_2 = 1.54 \). Although Hunziker and Gyarmathy
studied 15, 25, and 30 degree diffusers, the 25 degree diffuser was chosen because the
data given in Dalbert et al. was for the 25 degree diffuser at design conditions.

It is common to describe diffusers by their area ratio, AR, and by the ratio of
length, \( l \), to height, \( h \), or by their divergence angle, \( 2\theta \). The relationship between \( 2\theta \) and
\( l/h \) is described as:

\[
2\theta = 2\arctan[(AR - 1)/(2l/h)]
\]  \hspace{1cm} (4.1.1)

Table 4.1 gives the geometric data for all three diffusers in terms of AR, \( l/h \), and \( 2\theta \). It
Figure 4.1 Centrifugal impeller and 25 degree vaned radial diffuser (Reference 19)
Figure 4.2 Cross-section of the centrifugal compressor stage (Reference 19)
Figure 4.3 Sketch of the diffuser with its geometric specifications (Reference 19)
also describes the diffuser from the throat to an exit plane (C - D) and from an entry plane to an exit plane (B - D).

Table 4.1 Geometric data of diffusers

<table>
<thead>
<tr>
<th>α_{1vane}</th>
<th>AR_{BD}</th>
<th>l_{BD} h_{B}</th>
<th>2θ_{BD}</th>
<th>AR_{CD}</th>
<th>l_{CD} h_{C}</th>
<th>2θ_{CD}</th>
</tr>
</thead>
<tbody>
<tr>
<td>15°</td>
<td>2.95</td>
<td>10.2</td>
<td>10.9°</td>
<td>2.37</td>
<td>6.5</td>
<td>12.0°</td>
</tr>
<tr>
<td>25°</td>
<td>2.14</td>
<td>5.9</td>
<td>11.1°</td>
<td>1.90</td>
<td>4.2</td>
<td>12.1°</td>
</tr>
<tr>
<td>30°</td>
<td>1.96</td>
<td>4.9</td>
<td>11.3°</td>
<td>1.79</td>
<td>3.7</td>
<td>12.2°</td>
</tr>
</tbody>
</table>

The diffuser studies were carried out with the use of an unshrouded impeller. The impeller has a tip diameter, d_{2}, of 280 mm (r_{2} = 140 mm). The exit blade angle is 30 degrees back leaned and the exit width is 17 mm. The impeller has 22 total blades, 11 full blades and 11 splitter blades.

The performance map of the stage is given in Figure 4.4. The performance map shows three operating points, mild surge (A), design conditions (B), and near choke (C). The design condition operating point was chosen for comparison due to the availability of information provided by the author. This case had a volume flow rate of \( V = 1.3 \, \text{m}^3/\text{sec} \) taken from Figure 4.4 with a corresponding total pressure ratio of \( \pi = 1.57 \). The flow coefficient (Equation 2.5.3) for the diffuser was \( \varphi = 0.064 \).
Figure 4.4 Performance map of the stage at operating points (A), (B), and (C) (Reference 3)
4.2 Grid Generation

In order to generate a computational grid, the diffuser vane had to be plotted in $x$-$y$ coordinate space. Using the information provided by Dalbert et al. and Hunziker and Gyarmathy such as leading edge and trailing edge radius ratios, inlet blade angle, exit blade angle, impeller exit radius, and that the vane is a circular arc, the $x$ and $y$ coordinates of the vane could be determined. The solution required five equations and five unknowns and, with the use of a MathCAD routine (Appendix A), the $x$ and $y$ coordinates of the diffuser vane trailing edge ($x_2, y_2$), center of curvature ($x_c, y_c$), and radius ($R$) could be determined (Figure 4.5).

After the coordinates of the blade centerline were determined in $x$-$y$ coordinate space, the $r$-$\theta$ coordinates had to be determined for the inside arc (suction side), center arc, and outside arc (pressure side) from the grid origin. The vane thickness, $d$, is 2.94 mm, therefore, the inside arc was taken at a radius of $R - d/2$, the center arc was taken at a radius of $R$, and the outside arc was taken at a radius of $R + d/2$, which was referenced from the center of curvature of the diffuser vane. The center arc equation was solved for $x$, by incrementing $y$ in steps of $d/2$. After a point $(x, y)$ on the center arc was determined, the corresponding inner point $(x_i, y_i)$ and outer point $(x_o, y_o)$ were determined (Figure 4.6). The following equations were used to solve for the $x$-$y$ coordinates for the three arcs.

**Center arc:**

$$\begin{align*}
(x - x_c)^2 + (y - y_c)^2 &= R^2 \\
(4.2.1)
\end{align*}$$

**Inside arc:**
Figure 4.5 Diffuser vane plotted in x - y coordinate space
Figure 4.6 Method for determining the $r - \theta$ coordinates for the diffuser vane
\[ \frac{y_i - y_e}{y - y_e} = \frac{x_i - x_e}{x - x_e} = \frac{R - d/2}{R} \]  
\[ (4.2.2) \]

Outside arc:

\[ \frac{y_o - y_e}{y - y_e} = \frac{x_o - x_e}{x - x_e} = \frac{R + d/2}{R} \]  
\[ (4.2.3) \]

The first x-y point for the blade centerline was taken at a distance of \( d/2 \) from the x-axis then the corresponding inner and outer points were determined. The leading edge circular arc was then constructed from these points, tangent to the x-axis. Therefore, where the blade centerline crosses the x-axis is where the leading edge radius ratio equals 1.15.

The computational grid was specified in cylindrical coordinates, \( r, \theta, z \), with a repeating boundary midway between the vanes. The following equations were used to convert the x - y coordinates to r - \( \theta \) coordinates:

\[ r = (x^2 + y^2)^{1/2} \]  
\[ (4.2.4) \]

\[ \theta = \cos^{-1}(x/r) \]  
\[ (4.2.5) \]

The \( r - \theta \) values were then converted to Cartesian coordinates with a FORTRAN program, which produced the computational grid. The size of the grid is 52 x 21 x 15 (Figure 4.7). The i coordinate is in the flow direction, the j coordinate is in the tangential direction, and the k coordinate is in the hub to shroud direction. The near wall spacing at all the walls is 0.14 mm, which is 0.83% of the blade height.

There were two uncertainties when generating the grid. The first uncertainty was the trailing edge configuration. The leading edge of the vane was said to be a circular arc but the trailing edge configuration was not specified. Figure 6, in Dalbert et al., showed a picture of the diffuser vane used for measurements. It was decided from this figure that
Figure 4.7 Computational grid: I = 1 - 52, J = 1 - 21, K = 1 - 15
the computational grid would have a blunt trailing edge, similar to the one used by Dalbert et al. The second uncertainty was the magnitude of the exit blade angle. Dalbert et al. stated that the exit blade angle was 39 degrees. Figure 4, in Hunziker and Gyarmathy, shows the radius of curvature of the vane to impeller exit radius as \( r_2/r_2 = 4.28 \). With the given geometry, the radius of curvature was determined to be \( r_2/r_2 = 3.39 \) to obtain an exit angle of 39 degrees. The radius of curvature given by Hunziker and Gyarmathy corresponds to an exit blade angle of 40.85 degrees instead of the specified 39 degrees. It is also unclear if a consistent leading edge radius ratio was used for the two studies. Dalbert et al. specifies a ratio of 1.15 but Hunziker and Gyarmathy specify a ratio of 1.16, which may be the reason for the discrepancy of exit blade angle. The given leading edge radius ratios do not specifically define the location of the leading edge. It is subject to interpretation by the reader, whether the value given refers to the closest point to the impeller, center of the circular arc leading edge, or some other location. The data given by Dalbert et al. was used in this study. A leading edge radius ratio of 1.15 and an exit blade angle of 39 degrees. These uncertainties are noted for the reader to recognize what values were used in this thesis, since some conflict exists in the given data. It is doubtful that these uncertainties affected the outcome of the results a great deal.

The grid in Figure 4.7 was used to carry out the flow calculations. For the purpose of viewing the results, one diffuser vane did not allow for a good understanding of the flow development. The grid was doubled in the \( \theta \)-direction to produce a full blade-to-blade passage (Figure 4.8). It was also desired to find the throat plane and an exit plane to obtain certain diffuser performance characteristics and to study the diffuser
Figure 4.8 Doubled grid: \( I = 1 - 52, J = 1 - 42, K = 1 - 15 \)
Exit Line: \( I = 25 - 39, J = 12 - 26 \)
Throat Line: \( I = 14 - 19.6, J = 12 - 30 \)
subcomponents.

To prevent backflow at the computational exit the grid extends through the throttle as shown in Figure 4.9. The contraction was estimated at 25% of the diffuser height, which was taken from Figure 2 in Hunziker and Gyarmathy. The throttle ring is located at a radius of 280 mm (Figure 4.2).
Figure 4.9 Computational diffuser viewed in θ - direction
5.0 DIFFUSER INLET FLOW CONDITIONS

In order to carry out the computational study, the inlet flow conditions had to be established to produce the proper flow pattern within the diffuser. Data was taken from Dalbert et al. (3) and Hunziker and Gyarmathy (19) which was used to fix the inlet conditions for the diffuser study. After the analysis was performed and the flow calculations were run, the shroud static pressure distribution and mass flow rate were checked for agreement with the diffuser test results from Dalbert et al.

5.1 Inlet Conditions Analysis

The inlet boundary conditions were given in Dalbert et al., Figure 5, for a radius ratio of 1.05. The stagnation pressure distribution and flow angles were given for 20 data points from hub to shroud (Figure 5.1). The density was determined from the compressor inlet total pressure, $P_{t1} = 96.0$ kPa, and the compressor inlet total temperature, $T_{t1} = 297.15$ K, from Hunziker and Gyarmathy. The density was determined from the following equation:

$$\rho_{t1} = P_{t1}/(RT_{t1}) \quad (5.1.1)$$

The mass flow rate was then determined from the density and volume flow rate, $V = 1.3$ m$^3$/sec, from Dalbert et al., and was calculated to be $m = 1.4634$ kg/sec. The mass flow rate was determined from the following equation:

$$m = V\rho_{t1} \quad (5.1.2)$$

The tip speed Mach number was given to be $M_{u2} = 0.75$, and from the equation

$$M_{u2} = U/(kRT_{t1})^{1/2} \quad (5.1.3)$$
Figure 5.1 Inlet boundary conditions (Reference 3)
the tip speed was determined to be \( U = 259.15 \text{ m/sec} \). Since the radius ratio from impeller exit to diffuser inlet was \( r/r_2 = 1.05 \), the equivalent blade speed at the diffuser inlet was determined to be \( U_2 = 272.11 \text{ m/sec} \).

To determine the inlet boundary conditions, a total temperature distribution and a static temperature were required. Since the inlet stagnation pressure was given, the following relationship was used:

\[
P_2/P_{t2} = (T_2/T_{t2})^{k-1}
\]  

For the total temperature, a uniform total energy or rotary stagnation temperature, \( T_2^* \), was assumed, described as

\[
T_2^* = T_{t1} = T_2 + \frac{V_2^2}{2c_p} - \left[\frac{(2U_2V_{\theta2})}{2c_p}\right]
\]  

where:

\[
2U_2V_{\theta2} = V_2^2 - W_2^2 + U_2^2
\]

The following equations were then substituted into Equation 5.1.4

\[
T_2 = T_{t1} - \frac{V_2^2}{2c_p} + \frac{(U_2V_{\theta2})}{2c_p}
\]

\[
T_{t2} = T_2 + \frac{V_2^2}{2c_p} = T_{t1} + \frac{U_2V_{\theta2}}{2c_p}
\]

to obtain the following equation:

\[
\frac{P_2}{P_{t2}} = \left[ \frac{T_{t1} - \frac{(V_{rad2}/\sin \alpha_2)^2}{2C_p} + \frac{U_2}{C_p} \frac{V_{rad2}}{\tan \alpha_2}}{T_{t1} + \frac{U_2}{C_p} \frac{V_{rad2}}{\tan \alpha_2}} \right]^{k-1}
\]  

(5.1.9)
Equation 5.1.9 was manipulated into the form of a quadratic formula from which one unknown, the radial velocity, could be found. The following list of equations were then used to establish the inlet conditions:

\[ V_2 = \frac{V_{rad2}}{\sin \alpha_2} \]  
\[ V_{02} = \frac{V_{rad2}}{\tan \alpha_2} \]  
\[ T_{t2} = T_{t1} + \frac{(U_2 V_{02})}{c_p} \]  
\[ T_2 = T_{t2} - \frac{V_2^2}{2c_p} \]

The equations were solved for each data point given in Figure 5.1. The stagnation pressure, \( P_{t2} \), and flow angle, \( \alpha_2 \), were taken from the plots and used in Equation 5.1.9. The static pressure plots given in Figure 6(b), Dalbert et al., show the measured inlet static pressure, \( P_2 \), to be 1.260 bar. This value was used for the calculations and held constant over the diffuser height. The given data and the equations were solved using a FORTRAN program (Appendix B). The density was determined at each data point from hub to shroud using the static temperature and pressure. The mass flow rate was then determined by summing over the diffuser height, using the following equation, where \( r \) is the radius at the diffuser inlet (\( r = 147 \) mm):

\[ m = 2 \pi r \int_0^{b_2} \rho_2 V_{rad2} \, dz \]

It was desired to match the mass flow rate in the calculations with the flow rate calculated by the given inlet conditions. With the original data, a mass flow rate of 1.76 kg/s was found, which did not match the specified mass flow rate of 1.46 kg/s. It was
then decided that the given flow angles had to be altered to obtain the correct mass flow rate. The flow angles were chosen to be examined more closely rather than the given inlet stagnation pressure because there is a greater chance to measure angles incorrectly than the stagnation pressure. This was confirmed by a letter from Peter Dalbert stating that their measured flow angles could be off by two degrees. Since the original calculation produced a mass flow rate higher than the desired flow rate, the flow angles needed to be reduced. Another parameter that was chosen to be examined more closely was the inlet static pressure. Adjustments of the inlet static pressure were considered to modify the mass flow rate. From Dalbert et al., Figure 6(b), it was estimated that the mean static pressure at \( r/r_2 = 1.05 \) was 1.280 bars. However, with most of the velocity head in the tangential direction, small changes in pressure have little effect on the flow rate.
5.2 Flow Angle Correction

The initial calculations were run using the data given in Figure 5.1. The correct mass flow rate, \( m = 1.46 \text{ kg/sec} \), could not be obtained using the given data. The flow angles were adjusted until the proper mass flow rate was obtained. Uniform changes in flow angle were examined to investigate the effects on the mass flow rate (Figure 5.2). It was concluded that the flow angles were off by 4.1 degrees. The flow angles were reduced by this amount and the calculations were run. Table 5.1 shows the final results of the calculations, and Table 5.2 shows the comparison of the data with the given flow angles and the corrected flow angles. The flow angles given by Dalbert et al. and the corrected flow angles were plotted versus the relative span and shown in Figure 5.3.

The corrected flow angles gave a higher incidence on the leading edge of the diffuser vane. The radial velocity was correspondingly decreased by a significant amount over the entire diffuser height (Figure 5.4), and the tangential velocity was increased but only for the last 50% of the diffuser height from hub to shroud (Figure 5.5). This shows that the radial component of velocity is largely affected by a variation of flow angle and that the tangential component is only slightly affected. The slight decrease in the tangential velocity is due to the change in the stagnation temperature. The calculations were based on a uniform rotary stagnation temperature in the relative frame, but, the stagnation temperature in the absolute frame was not uniform (Figure 5.6).
Figure 5.2: Effects of reduction in flow angle on the mass flow rate,
Table 5.1 Final calculations of the diffuser inlet data

<table>
<thead>
<tr>
<th>b/b2 (relative span)</th>
<th>P2 (mbar)</th>
<th>Pt2 (mbar)</th>
<th>alpha2 (deg.)</th>
<th>V2 (m/sec)</th>
<th>Vrad2 (m/sec)</th>
<th>Vth2 (m/sec)</th>
<th>Tt2 (K)</th>
<th>T2 (K)</th>
<th>rho2 (kg/m3)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>1280.0</td>
<td>1383.0</td>
<td>0.2</td>
<td>120.31</td>
<td>0.42</td>
<td>120.31</td>
<td>329.77</td>
<td>322.56</td>
<td>1.3827</td>
</tr>
<tr>
<td>0.025</td>
<td>1280.0</td>
<td>1409.0</td>
<td>0.6</td>
<td>134.62</td>
<td>1.41</td>
<td>134.61</td>
<td>333.65</td>
<td>324.62</td>
<td>1.3739</td>
</tr>
<tr>
<td>0.050</td>
<td>1280.0</td>
<td>1428.0</td>
<td>1.5</td>
<td>144.11</td>
<td>3.77</td>
<td>144.06</td>
<td>336.21</td>
<td>325.87</td>
<td>1.3686</td>
</tr>
<tr>
<td>0.120</td>
<td>1280.0</td>
<td>1456.0</td>
<td>5.4</td>
<td>156.23</td>
<td>14.77</td>
<td>156.23</td>
<td>339.51</td>
<td>327.24</td>
<td>1.3629</td>
</tr>
<tr>
<td>0.175</td>
<td>1280.0</td>
<td>1483.0</td>
<td>10.1</td>
<td>166.15</td>
<td>29.49</td>
<td>165.54</td>
<td>342.04</td>
<td>327.95</td>
<td>1.3599</td>
</tr>
<tr>
<td>0.235</td>
<td>1280.0</td>
<td>1520.0</td>
<td>14.8</td>
<td>182.12</td>
<td>48.52</td>
<td>176.07</td>
<td>344.89</td>
<td>328.37</td>
<td>1.3582</td>
</tr>
<tr>
<td>0.295</td>
<td>1280.0</td>
<td>1553.0</td>
<td>18.4</td>
<td>193.43</td>
<td>61.06</td>
<td>183.54</td>
<td>346.92</td>
<td>328.28</td>
<td>1.3586</td>
</tr>
<tr>
<td>0.355</td>
<td>1280.0</td>
<td>1583.0</td>
<td>20.0</td>
<td>203.08</td>
<td>69.46</td>
<td>190.83</td>
<td>348.90</td>
<td>328.35</td>
<td>1.3583</td>
</tr>
<tr>
<td>0.418</td>
<td>1280.0</td>
<td>1608.0</td>
<td>21.9</td>
<td>210.58</td>
<td>78.54</td>
<td>195.38</td>
<td>350.13</td>
<td>328.04</td>
<td>1.3596</td>
</tr>
<tr>
<td>0.475</td>
<td>1280.0</td>
<td>1623.0</td>
<td>22.7</td>
<td>214.92</td>
<td>82.94</td>
<td>198.27</td>
<td>350.91</td>
<td>327.90</td>
<td>1.3602</td>
</tr>
<tr>
<td>0.538</td>
<td>1280.0</td>
<td>1623.0</td>
<td>23.8</td>
<td>214.77</td>
<td>86.67</td>
<td>196.51</td>
<td>350.43</td>
<td>327.45</td>
<td>1.3620</td>
</tr>
<tr>
<td>0.593</td>
<td>1280.0</td>
<td>1617.0</td>
<td>25.4</td>
<td>212.79</td>
<td>91.27</td>
<td>192.22</td>
<td>349.27</td>
<td>326.71</td>
<td>1.3651</td>
</tr>
<tr>
<td>0.655</td>
<td>1280.0</td>
<td>1607.0</td>
<td>25.8</td>
<td>209.77</td>
<td>91.30</td>
<td>188.86</td>
<td>348.38</td>
<td>326.44</td>
<td>1.3682</td>
</tr>
<tr>
<td>0.713</td>
<td>1280.0</td>
<td>1601.0</td>
<td>26.7</td>
<td>207.83</td>
<td>93.38</td>
<td>185.67</td>
<td>347.50</td>
<td>325.97</td>
<td>1.3682</td>
</tr>
<tr>
<td>0.773</td>
<td>1280.0</td>
<td>1600.0</td>
<td>28.0</td>
<td>207.34</td>
<td>97.34</td>
<td>183.07</td>
<td>346.79</td>
<td>325.37</td>
<td>1.3707</td>
</tr>
<tr>
<td>0.835</td>
<td>1280.0</td>
<td>1599.0</td>
<td>28.6</td>
<td>206.95</td>
<td>99.06</td>
<td>181.70</td>
<td>346.42</td>
<td>325.08</td>
<td>1.3719</td>
</tr>
<tr>
<td>0.895</td>
<td>1280.0</td>
<td>1597.0</td>
<td>28.2</td>
<td>206.40</td>
<td>97.53</td>
<td>181.90</td>
<td>346.47</td>
<td>325.25</td>
<td>1.3712</td>
</tr>
<tr>
<td>0.948</td>
<td>1280.0</td>
<td>1564.0</td>
<td>28.0</td>
<td>196.03</td>
<td>92.03</td>
<td>173.08</td>
<td>344.08</td>
<td>324.94</td>
<td>1.3726</td>
</tr>
<tr>
<td>0.973</td>
<td>1280.0</td>
<td>1536.0</td>
<td>27.0</td>
<td>186.73</td>
<td>84.77</td>
<td>166.38</td>
<td>342.28</td>
<td>324.69</td>
<td>1.3727</td>
</tr>
<tr>
<td>1.000</td>
<td>1280.0</td>
<td>1493.0</td>
<td>25.4</td>
<td>171.11</td>
<td>73.40</td>
<td>154.57</td>
<td>339.06</td>
<td>324.48</td>
<td>1.3745</td>
</tr>
</tbody>
</table>

Mass flow rate = 1.4601 kg/sec
Table 5.2 Comparison of the inlet velocity with given and corrected flow angles

<table>
<thead>
<tr>
<th>b/b2</th>
<th>Pt2</th>
<th>alpha2</th>
<th>alpha2</th>
<th>Vrad2</th>
<th>Vrad2</th>
<th>Vth2</th>
<th>Vth2</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>a2corr.</td>
<td></td>
<td>a2corr.</td>
<td></td>
<td>a2corr.</td>
<td></td>
</tr>
<tr>
<td>0.000</td>
<td>1383.0</td>
<td>0.2</td>
<td>0.4</td>
<td>0.42</td>
<td>0.92</td>
<td>120.31</td>
<td>132.49</td>
</tr>
<tr>
<td>0.025</td>
<td>1409.0</td>
<td>0.6</td>
<td>2.0</td>
<td>1.41</td>
<td>5.09</td>
<td>134.61</td>
<td>145.63</td>
</tr>
<tr>
<td>0.060</td>
<td>1428.0</td>
<td>1.5</td>
<td>4.0</td>
<td>3.77</td>
<td>10.78</td>
<td>144.06</td>
<td>154.21</td>
</tr>
<tr>
<td>0.120</td>
<td>1456.0</td>
<td>5.4</td>
<td>9.5</td>
<td>14.77</td>
<td>27.50</td>
<td>156.23</td>
<td>164.31</td>
</tr>
<tr>
<td>0.175</td>
<td>1483.0</td>
<td>10.1</td>
<td>14.2</td>
<td>29.49</td>
<td>43.46</td>
<td>165.54</td>
<td>171.74</td>
</tr>
<tr>
<td>0.235</td>
<td>1520.0</td>
<td>14.8</td>
<td>18.9</td>
<td>46.52</td>
<td>61.66</td>
<td>176.07</td>
<td>180.10</td>
</tr>
<tr>
<td>0.295</td>
<td>1553.0</td>
<td>18.4</td>
<td>22.5</td>
<td>61.06</td>
<td>76.96</td>
<td>183.54</td>
<td>185.80</td>
</tr>
<tr>
<td>0.355</td>
<td>1583.0</td>
<td>20.0</td>
<td>24.1</td>
<td>69.46</td>
<td>85.89</td>
<td>190.83</td>
<td>192.00</td>
</tr>
<tr>
<td>0.418</td>
<td>1608.0</td>
<td>21.9</td>
<td>26.0</td>
<td>78.54</td>
<td>95.34</td>
<td>195.36</td>
<td>195.48</td>
</tr>
<tr>
<td>0.475</td>
<td>1623.0</td>
<td>22.7</td>
<td>26.8</td>
<td>82.94</td>
<td>99.94</td>
<td>198.27</td>
<td>197.84</td>
</tr>
<tr>
<td>0.538</td>
<td>1623.0</td>
<td>23.8</td>
<td>27.9</td>
<td>86.67</td>
<td>103.63</td>
<td>196.51</td>
<td>195.72</td>
</tr>
<tr>
<td>0.593</td>
<td>1617.0</td>
<td>25.4</td>
<td>29.5</td>
<td>91.27</td>
<td>108.09</td>
<td>192.22</td>
<td>191.04</td>
</tr>
<tr>
<td>0.655</td>
<td>1607.0</td>
<td>25.8</td>
<td>29.9</td>
<td>91.30</td>
<td>107.96</td>
<td>188.86</td>
<td>187.74</td>
</tr>
<tr>
<td>0.713</td>
<td>1601.0</td>
<td>26.7</td>
<td>30.8</td>
<td>93.38</td>
<td>109.92</td>
<td>185.67</td>
<td>184.40</td>
</tr>
<tr>
<td>0.773</td>
<td>1600.0</td>
<td>28.0</td>
<td>32.1</td>
<td>97.34</td>
<td>113.80</td>
<td>183.37</td>
<td>181.41</td>
</tr>
<tr>
<td>0.835</td>
<td>1599.0</td>
<td>28.6</td>
<td>32.7</td>
<td>99.06</td>
<td>115.48</td>
<td>181.70</td>
<td>179.88</td>
</tr>
<tr>
<td>0.895</td>
<td>1597.0</td>
<td>28.2</td>
<td>32.3</td>
<td>97.53</td>
<td>113.95</td>
<td>181.90</td>
<td>180.24</td>
</tr>
<tr>
<td>0.948</td>
<td>1564.0</td>
<td>28.0</td>
<td>32.1</td>
<td>92.03</td>
<td>108.03</td>
<td>173.08</td>
<td>172.21</td>
</tr>
<tr>
<td>0.973</td>
<td>1536.0</td>
<td>27.0</td>
<td>31.1</td>
<td>84.77</td>
<td>100.42</td>
<td>166.38</td>
<td>166.47</td>
</tr>
<tr>
<td>1.000</td>
<td>1493.0</td>
<td>25.4</td>
<td>29.5</td>
<td>73.40</td>
<td>88.43</td>
<td>154.57</td>
<td>156.29</td>
</tr>
</tbody>
</table>
Figure 5.3 Comparison of the inlet flow angles
Figure 5.4 Comparison of the radial velocity profiles
Figure 5.5 Comparison of the tangential velocity profile
Figure 5.8 Inlet total temperature distribution
5.3 Computational Boundary Conditions

In order to run the calculations, certain boundary conditions and inlet data had to be specified. At the inlet \((r/r_2 = 1.05)\), the radial and tangential components of velocity define the flow angles. The stagnation temperature was input as \(T_{t_2} - T_{t2\text{ref}}\) where \(T_{t2\text{ref}}\) is the stagnation temperature which corresponds to the maximum stagnation pressure, \(P_{t2\text{ref}}\). The static pressure was input as \(P_2 - P_{t2\text{ref}}\), which was one value because the static pressure was kept constant over the diffuser height. The density, \(\rho_{t2\text{ref}}\), was also input, evaluated at \(P_{t2\text{ref}}\) and \(T_{t2\text{ref}}\). The inlet profiles specified for the calculations are given in Table 5.3. The inlet analysis had 20 data points from hub to shroud, but there are only 15 z-grid lines for the computational diffuser. Therefore, the relative span, radial velocity, tangential velocity, and stagnation temperature were linearly interpolated from the inlet analysis to the grid lines. The radial velocity and tangential velocity were set to zero at the viscous hub and shroud walls (no-slip condition). The imposed radial velocity profile is shown in Figure 5.7.

The modified measured data at the inlet was consistent with a mass flow rate, \(m\), of 1.46 kg/sec, which, for 24 diffuser vanes, gives a mass flow rate of 0.0608 kg/sec for each passage. The calculations were checked for consistency of the mass flow rate. The calculations produced a flow rate of 0.0604 kg/sec for each passage, which gives an overall calculated flow rate, \(m_{\text{calculated}}\), of 1.45 kg/sec. This reduction in flow rate of approximately 1% is due to the boundary layer truncation due to the no-slip condition at the shroud wall. This truncation can be seen in the imposed radial velocity profile in Figure 5.7.
Table 5.3  Computational inlet boundary conditions

<table>
<thead>
<tr>
<th>b/b2 (m/sec)</th>
<th>Vrad2 (m/sec)</th>
<th>Vth2 (m/sec)</th>
<th>Tt2-Ttref (K)</th>
<th>Pt2ref = 1623 mbars (162300 Pa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.000</td>
<td>0.00</td>
<td>0.00</td>
<td>-21.14</td>
<td>Ttref = 350.91 K</td>
</tr>
<tr>
<td>0.008</td>
<td>0.75</td>
<td>125.06</td>
<td>-19.85</td>
<td>P2 - Pt2ref = -343</td>
</tr>
<tr>
<td>0.025</td>
<td>1.41</td>
<td>134.61</td>
<td>-17.26</td>
<td></td>
</tr>
<tr>
<td>0.058</td>
<td>3.64</td>
<td>143.55</td>
<td>-14.84</td>
<td>rho2ref (Pt2ref, Ttref) = 1.6115 kg/m3</td>
</tr>
<tr>
<td>0.125</td>
<td>15.98</td>
<td>157.08</td>
<td>-11.19</td>
<td></td>
</tr>
<tr>
<td>0.250</td>
<td>50.16</td>
<td>177.94</td>
<td>-5.51</td>
<td></td>
</tr>
<tr>
<td>0.375</td>
<td>72.35</td>
<td>192.27</td>
<td>-1.62</td>
<td></td>
</tr>
<tr>
<td>0.500</td>
<td>84.42</td>
<td>197.57</td>
<td>-0.19</td>
<td></td>
</tr>
<tr>
<td>0.625</td>
<td>91.29</td>
<td>190.49</td>
<td>-2.11</td>
<td></td>
</tr>
<tr>
<td>0.750</td>
<td>95.83</td>
<td>184.07</td>
<td>-3.85</td>
<td></td>
</tr>
<tr>
<td>0.876</td>
<td>98.03</td>
<td>181.84</td>
<td>-4.46</td>
<td></td>
</tr>
<tr>
<td>0.942</td>
<td>92.67</td>
<td>174.13</td>
<td>-6.55</td>
<td></td>
</tr>
<tr>
<td>0.975</td>
<td>83.89</td>
<td>165.51</td>
<td>-8.89</td>
<td></td>
</tr>
<tr>
<td>0.992</td>
<td>76.90</td>
<td>158.20</td>
<td>-10.85</td>
<td></td>
</tr>
<tr>
<td>1.000</td>
<td>0.00</td>
<td>0.00</td>
<td>-11.85</td>
<td></td>
</tr>
</tbody>
</table>
Figure 5.7 Imposed radial velocity profile for the calculations
The exit boundary condition was that $\partial P/\partial n = \text{constant}$. The pressure level varies over the exit plane, but the pressure changes a uniform amount from the next to last plane to the last plane. The mass flow rate, $m_{\text{calculated}}$, of 1.45 kg/sec was used as a boundary condition at the exit of the diffuser.

Some turbulence properties also had to be specified for the calculations. The calculation uses a mixing length model approximation. The free-stream mixing length for the calculation was 0.13 mm.
6.0 RESULTS AND DISCUSSION

6.1 Static Pressure Comparison

The results obtained by the computations were verified with the static pressure plot in Figure 6b from Dalbert et al. (3) Dalbert et al. used 81 static pressure taps on the shroud wall to determine the static pressure distribution, therefore, the calculated plot was also evaluated at the shroud wall. The calculated static pressure plot was very close to the given plot in terms of overall pressure rise. The plot given in Dalbert et al. shows an overall pressure rise to 1.5 bars, and the calculated plot had an overall pressure rise to 1.488 bars (Figure 6.1). The measurements had a static pressure at the throat of 1.41 bars, which gave a static pressure rise of 0.09 bars to the diffuser exit. The calculations had a static pressure of 1.391 bars at the throat, which gave a static pressure rise of 0.097 bars.

6.1.1 Pressure Rise Comparison

The diffuser was broken down into its subcomponents (Figure 4.8) and the static pressure rise was examined and compared to Figure 11 in Hunziker and Gyarmathy (19). The pressure rise coefficient was defined as:

\[ D_p = \frac{\Delta p}{1/2 \rho U^2} \]  \hspace{1cm} (6.1.1)

The static pressure at the inlet, \( P_2 \), was taken as 1.280 bars, and at the exit, \( P_3 \), was determined to be 1.488 bars. The static pressures at the throat plane, \( C \), and at the exit plane, \( D \), were determined through a FORTRAN program. Once the computational grid
Figure 6.1(a) Measured static pressure distribution (Reference 3)

Figure 6.1(b) Calculated static pressure distribution
was established, a grid line was generated for each of these planes. The static pressures were determined along these grid lines on the shroud side of the diffuser and then the average static pressures were calculated.

Table 6.1 gives the static pressure rise coefficient for each subcomponent as well as the overall static pressure rise for the diffuser.

<table>
<thead>
<tr>
<th>Component</th>
<th>Calculated</th>
<th>From Figure 11</th>
</tr>
</thead>
<tbody>
<tr>
<td>entry 2-C</td>
<td>0.30</td>
<td>0.35</td>
</tr>
<tr>
<td>channel C-D</td>
<td>0.22</td>
<td>0.21</td>
</tr>
<tr>
<td>exit D-3</td>
<td>0.04</td>
<td>0.03</td>
</tr>
<tr>
<td>diffuser 2-3</td>
<td>0.56</td>
<td>0.60</td>
</tr>
</tbody>
</table>

Figure 6.2 shows the data from Figure 11 for a Mach number of 0.75 and the calculated data for a flow coefficient of 0.064. The figure shows that the calculations were very close to the given information and the calculated pressure rise coefficient was only in error by 0.04 for the diffuser.

The purpose of examining the subcomponents is to understand how they affect the overall diffuser performance. Since static pressure rise is the main goal of a diffuser, it is beneficial to see which component yields the most pressure rise. The entrance of the diffuser, inlet to the throat, produces the most pressure rise in the diffuser. In this case,
Figure 6.2 Comparison of the calculated static pressure rise to the given data (Reference 19)
the entry region produces 54% of the diffuser's pressure recovery, the passage produces 39%, and the exit region only produces 7%. The entry region is where the calculation had the largest discrepancy with the measurements, with a lower calculated pressure rise coefficient of 0.30 compared with 0.35 in the data. Overall, however, the calculated static pressure rise coefficient of 0.56 was in very good agreement with the measured value of 0.60.
6.2 The Inlet Region

The inlet of the diffuser is from plane 2 \((r/r_2 = 1.05)\) to the throat plane, seen in Figure 4.8. This area comprises the vaneless space and the semi-vaneless space. The radial velocity vectors were generated for the inlet of the diffuser to obtain an understanding of the flow before it enters the blade passage. This was done with a post-processing program which uses the results produced by the 3D flow calculation. The vectors were analyzed for the inlet of the diffuser from \(r/r_2 = 1.05\) to \(r_{1c}/r_2 = 1.15\), which is the vane leading edge. They were plotted from hub to shroud for three grid surfaces, the upstream extension of the suction surface (Figure 6.3) and the pressure surface (Figure 6.4) and midway between the blades (Figure 6.5). All three plots show a hub surface/leading edge flow separation, but it is more evident upstream of the pressure and suction surfaces, i.e. upstream of the vane leading edge. It is apparent that there is a horseshoe vortex/leading edge recirculation which extends radially inwards, but it is unclear if the physical flow goes back to the impeller exit. The calculations were started at a radius ratio of 1.05 with a circumferentially uniform flow. It was not computationally possible to determine how far the flow extends back towards the impeller exit, but, according to the measurements, the circumferentially mean flow was forward. Dawes (15) also found a hub/corner stall during his computational study. This was attributed to a large variation of inlet swirl angle on the order of 20 to 25 degrees from hub to shroud. In this computational study, the flow angles also have a large variation from hub to shroud on the order of 25 degrees. This could be the cause of the hub surface flow separation at the inlet, similar to what Dawes had found. There is also evidence of a much smaller
Figure 6.3 Inlet velocity vectors upstream of the suction surface
Figure 6.4  Inlet velocity vectors upstream of the pressure surface
Figure 6.5 Inlet velocity vectors midway between SS and PS
horseshoe vortex near the shroud, which can be seen in Figure 6.3.

Velocity vectors were also plotted on the hub, mid-height, and shroud surfaces (Figure 6.6). The plots show the flow for one blade-to-blade passage from the inlet to just downstream of the throat. The plots give a better illustration of the inlet skew at the leading edge. This inlet skew is the variation in flow angle which increases from hub to shroud.

The entry region is an important component of the diffuser. It has already been established that this area contains the highest static pressure rise and that flow separation occurs. The characteristics of the flow in this region strongly affects the flow at the throat of the diffuser. The throat is a critical area and has a significant role of the overall diffuser performance.
Figure 6.6 Calculated velocity vectors on hub, mid-height, and shroud surfaces for $I = 1 - 21$
6.3 The Diffuser Throat

As discussed earlier, a throat plane (Figure 4.8) was established to examine this area of the diffuser. Static pressures, total pressure losses, and blockage were studied to gain insight of the flow at the throat. The static pressure and total pressure losses were normalized with $1/2\rho_1U_2^2$ to obtain a consistent analysis with the given data.

6.3.1 Static Pressure Analysis

The area averaged static pressure at the throat was determined to be 1.397 bars, the line averaged pressure along the shroud, which was used in the pressure rise analysis, was determined to be 1.391 bars, and, for comparison, the line averaged static pressure along the hub was determined to be 1.406 bars. A static pressure coefficient, $C_{ps}$, was used to analyze the static pressures. The coefficients were plotted at the throat (Figure 6.7) and were defined as:

$$C_{ps} = \frac{P - P_{ref}}{1/2\rho_1U_2^2} \quad (6.3.1)$$

The plot shows a high pressure region at the hub/pressure side corner. There is also a band of low pressures on the pressure surface, which begins about 25% above the hub ranging to the shroud wall. At the hub/pressure side corner the static pressure coefficient is around -0.5 and falls to about -0.9 further up the pressure side wall. There is also a low pressure region at the hub/suction side corner, but it is not as apparent as the low pressure on the pressure side corner. The static pressure coefficient at the suction surface corner is about -0.58.
Figure 6.7 Static Pressure Coefficients, $C_p$, on throat plane
The low and high pressure regions are due to the incidence, defined in Equation 2.4.2, imposed by the angle of the flow and the leading edge angle of the blade. Figure 6.8 shows the relationship of the flow angle, incidence, and static pressure coefficients on the pressure side of the throat plane at the leading edge. As the flow angle increases with height, the incidence decreases causing the static pressure coefficient to also decrease. The low pressure region is due to a negative incidence. A negative incidence causes a stagnation point to occur at the nose of the blade and a local suction peak around the circular arc leading edge. In this study, for an inlet blade angle of 25 degrees, the incidence did not become negative until 58% of the relative span, which corresponds to a diffuser height of 9.7 mm. It is around this height that the lowest static pressure coefficient of −0.9 begins. The high pressure region at the hub/pressure side corner is due to a highly loaded blade caused by a positive incidence. The incidence is approximately zero at mid-height and near the shroud wall. At the hub, the incidence is approximately 25 degrees, therefore, the absolute velocity is approximately equal to the tangential velocity, which can be seen in Figure 6.6.

6.3.2 Total Pressure Losses

The total pressure loss coefficient was defined as:

\[ C_{pt} = \frac{P_{ref} - P_t}{1/2 \rho U_2^2} \]  \hspace{1cm} (6.3.2)

A total pressure loss plot is shown in Figure 6.9 to determine where the losses occur at the throat. A region of high loss is evident at the hub/pressure surface corner. This is
Figure 6.8 Relationship between flow angle, incidence, and static pressure coefficient at the pressure side of the leading edge
Figure 6.9 Total Pressure Losses, Cpt, on throat plane
due to the inlet losses at the hub and the hub-corner stall at the leading edge of the diffuser vane. An area of high loss also occurs on the shroud side near the suction surface of the vane. This may be due a horseshoe vortex at the shroud wall. The center of the throat plane sees very little to no losses, in fact, a large majority of the throat sees little pressure losses. The walls cause some pressure loss but it most evident at the hub/pressure surface corner. The flow is moving in a counter-clockwise direction which causes the high losses at the hub/pressure side corner. Conversely, the flow is moving in clockwise direction at the shroud causing high losses at the shroud/suction side corner. It is also apparent that higher losses occur on the suction surface than the pressure surface at the diffuser throat due to the longer blade length for the development of the suction surface boundary layer.

6.3.3 Throat Blockage

Blockage at the throat is a very important parameter which effects the overall diffuser performance. Since the flow leaving the impeller is highly distorted, some blockage will occur at the throat. The least amount of throat blockage is favorable in order to achieve the maximum amount of pressure rise in the diffuser. Throat blockage can be defined as

\[ Blc = 1 \frac{A_{effc}}{A_c} \quad (6.3.3) \]

where \( A_c \) is the geometric area at the throat and \( A_{effc} \) is the effective area at the throat. In order to determine the throat blockage, the effective area at the throat had to be computed. The throat static pressure, \( P_c \), the stagnation pressure, \( P_t \), and temperature, \( T_t \),
in the flow core, and the mass flow rate were needed. The following isentropic relationship was used to determine the effective area:

\[
\frac{m\sqrt{c_p \cdot T_t}}{A_{eff} \cdot P_t} = \frac{P}{P_t} = \sqrt{2} \cdot \frac{c_p}{R} \cdot \sqrt{\frac{P}{P_t}} \cdot \left[ \frac{P}{P_t} \right]^{\frac{1-k}{k}} \cdot \left[ \frac{P}{P_t} \right]^{\frac{1-k}{k}} - 1
\]  

(6.3.4)

Hunziker et al. plotted channel pressure recovery versus the throat blockage. The channel pressure recovery is defined as:

\[
C_{pc - D} = \frac{\Delta P_{c - D}}{P_t - P_c}
\]

(6.3.5)

The blockage was first determined for the given data in Hunziker and Gyarmathy. The throat static pressure and exit static pressure were averaged from the static pressure plot shown in Figure 6.1, but it was unclear what stagnation pressure and temperature was used in determining the throat blockage. Hunziker and Gyarmathy said that a pitot probe was used to measure the stagnation pressure in the flow core, therefore, the properties given at mid-height were initially used for determining the throat blockage. The stagnation pressure at the inlet of the diffuser for mid-height was given as 1.623 bars, and the corresponding stagnation temperature was 350.72 K. The throat blockage was determined for these properties and compared to Figure 15 in Hunziker and Gyarmathy. It was clear that the mid-height properties were not consistent with Figure 15, therefore, the mass averaged properties at the inlet were used to carry out the calculations. The mass averaged stagnation pressure was determined to be 1.586 bars, with a corresponding stagnation temperature of 347.20 K. The throat blockage was again determined and
favorable results were produced. Figure 6.10(a) shows the data plotted on Figure 15 from Hunziker and Gyarmathy. The blockage evaluated with measured mid-height properties was determined to be 0.19 and was not near the given data series, but, using the measured mass averaged properties, the blockage was determined to be 0.11. The blockage evaluated with mass averaged properties was very close to one of the given data points.

The calculations were evaluated in the same manner and similar results were obtained. Figure 6.10(b) shows the calculated blockages plotted on Figure 15 from Hunziker and Gyarmathy. The calculations were evaluated using an area averaged static pressure (denoted with a subscript 1 on Figure 6.10(b)) and a shroud averaged static pressure (denoted with a subscript 2 on Figure 6.10(b)) for comparison. Blockages ranged from 0.15 to 0.23 depending on the properties used. Using mass averaged properties, the blockage was determined to be 0.17 with a shroud static pressure and 0.15 with an area averaged static pressure. Using mid-height properties, the blockage was determined to be 0.23 with a shroud static pressure and 0.22 with an area averaged static pressure.

Table 6.2 presents the values used for the given data and the calculations and the results obtained. It is evident from the figures and table that the evaluation of throat blockage is very sensitive to the stagnation pressure. It was clear that blockages evaluated with mass averaged properties were closer to the given data series for both cases.
Figure 6.10(a) Throat blockage for the given data (Reference 19)

Figure 6.10(b) Throat blockage for the calculations (Reference 19)
Table 6.2 Pressure recovery and throat blockage for the given data and the calculations

<table>
<thead>
<tr>
<th></th>
<th>(P_e)</th>
<th>(P_d)</th>
<th>(P_t)</th>
<th>(T_t)</th>
<th>(C_{p_{sat}})</th>
<th>(B_{le})</th>
</tr>
</thead>
<tbody>
<tr>
<td>Given Data</td>
<td>1.414</td>
<td>1.489</td>
<td>1.623</td>
<td>350.72</td>
<td>0.36</td>
<td>0.19</td>
</tr>
<tr>
<td>(shroud)</td>
<td>(shroud)</td>
<td>(mid-height)</td>
<td>(mid-height)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Calculations</td>
<td>1.414</td>
<td>1.489</td>
<td>1.586</td>
<td>347.20</td>
<td>0.44</td>
<td>0.11</td>
</tr>
<tr>
<td>(shroud)</td>
<td>(shroud)</td>
<td>(mass ave.)</td>
<td>(mass ave.)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>1.391</td>
<td>1.473</td>
<td>1.623</td>
<td>350.72</td>
<td>0.35</td>
<td>0.23</td>
</tr>
<tr>
<td>(shroud)</td>
<td>(shroud)</td>
<td>(mid-height)</td>
<td>(mid-height)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>1.397</td>
<td>1.473</td>
<td>1.623</td>
<td>350.72</td>
<td>0.34</td>
<td>0.22</td>
</tr>
<tr>
<td>(area ave.)</td>
<td>(shroud)</td>
<td>(mid-height)</td>
<td>(mid-height)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>1.391</td>
<td>1.473</td>
<td>1.586</td>
<td>347.20</td>
<td>0.42</td>
<td>0.17</td>
</tr>
<tr>
<td>(shroud)</td>
<td>(shroud)</td>
<td>(mass ave.)</td>
<td>(mass ave.)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>1.397</td>
<td>1.473</td>
<td>1.586</td>
<td>347.20</td>
<td>0.40</td>
<td>0.15</td>
</tr>
<tr>
<td>(area ave.)</td>
<td>(shroud)</td>
<td>(mass ave.)</td>
<td>(mass ave.)</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Based on the shroud static pressure and mass averaged total pressure and temperature, the calculation underpredicts the pressure recovery by 5% and overpredicts the blockage by about 50%. Based on the core flow total pressure and temperature, the calculation underpredicts the pressure recovery by 3% and overpredicts the blockage by about 20%.

The amount of blockage at the throat is due to the inlet distortion and boundary layer growth on the suction side and on the hub and shroud walls. This can be seen from the total pressure loss plot (Figure 6.9). The losses accumulating on these walls restrict
the flow as it travels into the diffuser passage. These "boundary layers" reduce the area effectively used by the flow which increases the amount of blockage.
6.4 Ideal and Possible Pressure Recovery Coefficients

The pressure recovery coefficients in Table 6.2 may be compared with the ideal pressure recovery coefficient for the channel, \( C - D \), and with the pressure recovery expected from channel diffuser data.

Based on the shroud static and the mass averaged total pressure, Table 6.2 shows a measured pressure recovery coefficient of 0.44 with a blockage of 11% and a calculated value of 0.42 with a blockage of 17%.

The ideal pressure recovery coefficient is given by:

\[
C_{P_{C,D}} = 1 - \frac{1}{AR_{C,D}^2}
\]  

From Table 4.1, the 25° diffuser has an \( AR_{C,D} \) of 1.90 giving an ideal pressure recovery coefficient of 0.723. This would be the recovery for steady, incompressible, and inviscid flow with no boundary layer blockage.

Hunziker and Gyarmathy also show the expected pressure recovery based on the channel diffuser data of Reneau et al. (24) for a blockage of 5%. This is shown in Figure 6.10(c). For the 25° diffuser, the expected pressure recovery coefficient is seen to be approximately 0.55.

By comparing these results, it is clear that improved performance could be achieved if the throat blockage could be reduced. It is unlikely that a pressure recovery coefficient much above 0.55 could be obtained.
Figure 6.10(c) Expected vaned diffuser pressure recovery coefficients based on channel diffuser performance (Reference 3)
6.5 The Exit Region

An exit plane was established to analyze the flow once it left the vaned passage. The exit region is the area from the exit plane to plane 3 (r_3/r_2 = 1.79), which can be seen in Figure 4.8. The static pressure coefficient, C_{ps}, and total pressure losses, C_{pt}, were analyzed at the exit plane. The averaged shroud static pressure at the exit plane was determined to be 1.473 bars.

6.5.1 Static Pressure Analysis

Figure 6.11 shows the plot for static pressure coefficients, defined in Equation 6.3.1, at the exit plane. Higher pressure coefficients are a result of higher static pressures at the exit. Since the inlet distortion has mixed out, smaller pressure variations exist at the exit of the diffuser. The pressure side of the exit plane on the blade surface has a static pressure coefficient around -0.42, while the suction side has a static pressure coefficient around -0.375. The suction side, denoted as “SS”, does not lie within the diffuser passage, but extends to a line tangent to the suction side of the diffuser vane. The high pressure region seen on the hub/suction side, which was evident at the throat plane, has continued to the exit plane. The hub and shroud walls have a pressure variation from -0.36 to -0.42.

6.5.2 Total Pressure Losses

Total pressure losses, defined in Equation 6.3.2, are shown in Figure 6.12. The flow in the diffuser exit region is not as distorted as in the entrance region, resulting in
Figure 6.11 Static Pressure Coefficients, $C_{ps}$, on exit plane
Figure 6.12 Total Pressure Losses, Cpt, on exit plane
lower peak values of pressure losses. The low loss region seen at the center of the throat plane (Figure 6.9) has moved towards the hub/suction side corner of the exit plane. The high loss region, which was evident at the hub/pressure side corner of the throat plane, has moved up the pressure side at the exit. The peak loss coefficient has decreased from 0.55 at the throat to 0.45 at the exit. The high losses at the hub and shroud walls on the throat plane have also decreased on the exit plane as the wall static pressure has risen.

The movement of the low loss region and high loss region from the throat plane to the exit plane is evident from comparing the two plots. The secondary flows, which exist in the diffuser passage, explain the shifting of these regions from plane to plane.
6.6 Secondary Flows

The secondary flows that exist in the diffuser may be attributed to several mechanisms. Secondary flows are present at the inlet which are caused by inlet skew. The flow at the inlet can be seen in Figure 6.13. The flow moves towards the pressure surface from the hub to mid-height and towards the suction surface from mid-height to the shroud wall. There are also high losses at the inlet near the hub wall, which may be confined to the recirculating flow and may not convect into the passage. This recirculating flow that occurs at the hub can be seen in Figure 6.3. A horseshoe vortex may also occur at the hub of the leading edge of the diffuser vane, but this vortex flow at the hub is dominated by the recirculating flow. The secondary flows at the leading edge can also be seen in Figure 6.13. There is a horseshoe vortex at the shroud wall as well (Figure 6.3) and this convects into the blade passage along the suction side wall as seen in Figure 6.13. The secondary flow associated with this vortex carries high loss fluid from the suction side towards the pressure side, near the shroud wall. An illustration of the inlet skew, leading edge vortices, the hub recirculation, and total pressure effects are shown in Figure 6.14.

The secondary flow vectors at the throat plane are shown in Figure 6.15(a) and the secondary flow patterns are shown schematically in Figure 6.15(b). The flow generally travels in a counter-clockwise direction for about 70% of the diffuser height from hub to shroud and in an opposing direction near the shroud wall. This is a superpositioning of the main passage vortex due to the inlet skew and the horseshoe vortex near the shroud wall. A much weaker vortex is apparent at the hub/pressure side corner region of the throat.
Figure 6.13 Secondary flows at inlet and leading edge viewed at 25 degrees
Figure 6.14 Sketch of secondary flow mechanisms in a diffuser
Figure 6.15(a) Secondary flows at the throat plane
Figure 6.15(b) Sketch of secondary flow patterns at the throat
plane.

The secondary flows at the exit plane are shown in Figure 6.16. Little secondary flows exist at the exit due to the viscous action or due to the slowing down of the vortex. The flow on the pressure side has moved up towards the shroud but gets taken over by the majority of the flow moving from the suction side to the pressure side.

The flow development in the diffuser is affected by the secondary flows. The total pressure loss patterns are caused by the pattern of secondary flow within the diffuser.
Figure 6.16 Secondary flows at the exit plane
6.7 Flow Development

6.7.1 Total Pressure Loss Development

The flow development in the diffuser was analyzed from inlet to exit. Total pressure losses were studied on the near hub, mid-height, and near shroud surfaces and at certain circumferential planes throughout the diffuser. Studying the pressure losses allowed for a good understanding of the flow as it traveled through the diffuser.

Figure 6.17 shows total pressure losses calculated for the near hub, mid-height, and near shroud surfaces. They were analyzed from the grid line at I = 1, which corresponds to a radius ratio of 1.05, to the grid line at I = 52, which corresponds to the throttle of the diffuser. The hub surface plot shows the high losses in the vaneless and semi-vaneless areas. This corresponds to the highly distorted flow exiting the impeller and entering the diffuser. The high losses are evident up to the throat and are mostly confined to the recirculating flow seen in Figure 6.3. Downstream of the diffuser throat the losses are much lower. The mid-height plot has the lowest losses of the three planes. A low loss region seems to accumulate on the suction side of the vane and continues into the exit area of the diffuser. The shroud surface plot has an area of high losses at the suction side of the diffuser vane beginning at the leading edge and extending to the throat. These are associated with the horseshoe vortex flow seen in Figure 6.13. The losses decrease downstream of the throat. The losses are greater on the hub and shroud due to the boundary layer developments on these walls. The mid-height plane has no walls, except for the diffuser vanes, to alter the passing of the flow, therefore, lower losses exist. As the flow mixes out and decelerates in the diffuser passage, total pressure losses appear to be
Figure 6.17 Total Pressure Losses, Cpt, hub, mid-height, and shroud surfaces
Eight circumferential planes were chosen to examine the flow as it travels through the diffuser. The plots were created to give the impression of looking downstream in the passage. The circumferential planes at grid lines I =1, 4, 12, 18, 22, 26, 33, and 41 were chosen to be examined from hub to shroud for a vane-to-vane passage. Figures 6.18 through 6.21 show the total pressure loss plot for the eight circumferential planes.

At the inlet, high losses occur at the hub which increase to the leading edge of the diffuser due to the inlet recirculation. The high loss region still exists at the leading edge of the diffuser across the hub. The main passage vortex carries the high losses to the hub/pressure side corner for 25% of the diffuser height. Correspondingly, the main passage vortex moves the low loss fluid into the suction side half of the passage. As the flow further develops along the diffuser passage, high loss fluid is convected up the pressure surface by the passage vortex, and the high losses at the shroud/suction side corner at the leading edge are moved to the middle of the passage by the suction side leg of the shroud horseshoe vortex.

6.7.2 Mass Averaged Total Pressure Loss

The mass averaged total pressure losses were plotted versus a radial distance parameter, A, shown in Figure 6.22. The plot was established for the entire diffuser from entrance to exit. A parameter of A = -0.24 corresponds to plane 2, which is at r/r_2 = 1.05, A = 0 refers to the leading edge of the diffuser vane, A = 1 refers to the trailing edge, A = 1.52 refers to plane 3, which is at r_3/r_2 = 1.79, and A = 2 is where the throttle begins. The
Figure 6.18  Total Pressure Losses, Cpt, on circumferential planes I=1,4
Figure 6.19  Total Pressure Losses, Cpt, on circumferential planes I=12,18
Figure 6.20  Total Pressure Losses, Cpt, on circumferential planes I=22, 26
Figure 6.21 Total Pressure Losses, Cpt, on circumferential planes I=33,41
peak total pressure losses decrease further into the diffuser passage as the flow mixes out but the low loss values remain constant. However, the mass averaged values show that the overall level of losses increases from entrance to exit. The entry region, 2 - C, produces 56% of the losses, the passage, C - D, produces 26% of the losses, and the exit region, D - 3, produces 18% of the losses. Also, in the entry region, 20% of the total pressure losses occur in the semi-vaneless space.

6.7.3 Summary of the Flow Development

The inlet skew and high losses on the hub surface cause a leading edge recirculation at the hub. This captures the flow with the highest losses at the inlet. The highest losses are produced in the inlet region causing an increase in mass averaged loss from 0.098 to 0.164. In the passage, the mass averaged total pressure loss increases due to both mixing and boundary layer growth from 0.164 to 0.194. The size of the low loss region decreases and the magnitude of the minimum loss slowly increases. Secondary flows carry the high loss region, which began at the hub, to the pressure side of the blade. A much smaller high loss region in the shroud boundary layer moves from the suction side to mid-passage. The large secondary flows in the hub region begin as inlet skew. However, the throat and exit planes show the lowest static pressures on the pressure side of the blade. This gives rise to secondary flows in the hub and shroud boundary layers, moving the near wall flow from the suction side to the pressure side of the passage. In the exit region, the mass averaged total pressure loss increases from 0.194 to 0.215.

The inlet of the diffuser is a very important area in the diffuser. This area
contributes the highest losses, 56%, and the highest amount of static pressure rise, 54%. The passage contributes 26% of the losses and 39% of the static pressure rise, while the exit region contributes 18% of the losses and only 7% of the static pressure rise.

Velocity vectors were also plotted for the entire range of the diffuser. Figure 6.23 shows vectors on the pressure surface and suction surface. It is evident that the flow almost stagnates at the hub on the pressure surface due to the low momentum of the pressure side boundary layer over most of the blade length. The convection of the flow from the hub to the shroud due to the secondary flow on the pressure surface can also be seen. Figures 6.24 and 6.25 shows velocity vectors on the hub and shroud walls.
Figure 6.23  Velocity vectors on blade suction surface and pressure surface
Figure 6.24 Velocity vectors on the hub wall
7.0 CONCLUSIONS

A 3D CFD flow program, MEFP, was used to predict the flow field and performance of a vaned radial diffuser in a centrifugal compressor. The 25 degree diffuser operating with an impeller tip Mach number of 0.75 and a flow coefficient of 0.064 was chosen from the study performed by Dalbert et al. (3). The computational study was carried out and compared to the performance measured by Dalbert et al. and Hunziker and Gyarmathy (19), who also performed a study using the same diffuser.

The computational grid was established in accordance with the geometry and inlet conditions provided by Dalbert et al. After an initial calculation was run, it was evident that the given inlet flow angles had to be adjusted to match the specified flow rate of 1.46 kg/s. The inlet flow angles were reduced by 4.1 degrees. The inlet flow angle analysis indicated the sensitivity of the flow to the incidence. The reduction of the flow angle reduced the mass flow rate from 1.76 kg/s for the initial calculation to 1.46 kg/s. The change in flow angle also had a significant effect on the radial velocity profile but not on the tangential velocity profile.

The results of the computational study were verified by comparing with the static pressure distribution measured on the shroud wall by Dalbert et al. Good agreement was obtained with the static pressure distribution throughout the diffuser and especially in the important inlet region. The test case diffuser had a static pressure rise of 0.22 bars and the computational diffuser had a rise of 0.21 bars. The measured static pressure rise coefficient, Dp, was 0.60 compared with a computed value of 0.56.

The diffuser was broken down into three subcomponents, the inlet region (diffuser
inlet at $r/r_2 = 1.05$ to the throat plane), the vaned passage (throat plane to exit plane), and the exit region (exit plane to $r_2 = 1.79$). The static pressure rise and mass-averaged total pressure losses in each component were evaluated. The inlet region produces 54% of the pressure rise and 56% of the losses, the passage produces 39% of the pressure rise and 26% of the losses, and the exit region produces 7% of the pressure rise and 18% of the losses. Since the static pressure rise is the main goal of a diffuser, it is important to observe that the inlet region produces the most pressure rise in the diffuser.

The throat plane was studied more closely to observe the effects of the inlet distortion and to evaluate the throat blockage. A large region of high loss was evident at the hub/pressure side corner due to the inlet losses at the hub and the hub-corner stall at the leading edge of the diffuser vane. A smaller region of high loss was also evident at the shroud/suction side of the throat plane. The flow incidence was positive over the hub half of the diffuser vanes and negative over the shroud half, due to inlet skew. A low pressure region was observed on the pressure side of the throat plane near the shroud due to the negative incidence. The suction side boundary layer was thicker than the pressure side layer because of the longer blade length up to the throat.

The throat blockage, which is another important parameter when studying diffusers, was determined for the computational diffuser and for the test case diffuser. The throat blockage is due to the inlet distortion and boundary layer growth on the suction side and on the hub and shroud walls. The data given in Hunziker and Gyarmathy was initially used in order to determine the procedure for solving for the throat blockage. Using core-flow properties did not give agreement with the throat blockage plot given in
Hunziker and Gyarmathy. The blockage was then evaluated with mass averaged properties which gave better agreement. The calculations were evaluated in a similar manner and similar results were obtained. The blockage for a shroud averaged static pressure evaluated with mass-averaged properties was determined to be 0.17 for the calculations and 0.11 for the given data, which is an overprediction of 55%. The blockage evaluated using mid-height properties was determined to be 0.23 for the calculations and 0.19 for the given data, which is an overprediction of 20%; but these values were not close to the blockage plot given in Hunziker and Gyarmathy.

The flow development and secondary flow mechanisms were studied in the diffuser. At the inlet, a leading edge recirculation is evident at the hub, which is where the highest losses were reported by Dalbert et al. Secondary flows carry the high losses at the hub to the pressure side of the blade, further into the diffuser passage. A small high loss region, which began at the leading edge at the shroud, is moved to mid-passage. Therefore, there exist two regions of high losses which are circulated within the diffuser by the secondary flow patterns.

In summary, the computational diffuser predicted the performance in close agreement to the test case diffuser studied by Dalbert et al. The use of CFD enabled the visualization of the 3D flow in the diffuser. The computational study provided a good understanding of the flow and performance in a vaned radial diffuser.
BIBLIOGRAPHY


APPENDIX A

Program which solves for cartesian coordinates for the computational diffuser vane
MathCAD Program
MathCAD program to solve for x-y coordinates for the diffuser vane:
Solve: 5 equations and 5 unknowns, yc, xc, R, x2, y2

ra := 1.15 (radius ratio at leading edge)
rb := 1.54 (radius ratio at trailing edge)
r2 := 140 mm (impeller exit radius)
R1 := ra*r2  R1 = 161 mm
R2 := rb*r2  R2 = 215.6 mm
x1 := 161 mm
y1 := 0 mm

a1 := 25 \times \frac{\pi}{360}  a1 = 0.436 rad
a2 := 39 \times \frac{\pi}{360}  a2 = 0.681 rad

Initial guesses for unknowns
yc := -800
xc := -1500
R := 2000
x2 := 175
y2 := -150

GIVEN (Equations to be satisfied)

\frac{yc}{x1 - xc} = \tan(a1)

x^2 + y^2 = R^2

(x2 - xc)^2 + (y2 - yc)^2 = R^2

(x1 - xc)^2 + (y1 - yc)^2 = R^2

\frac{x2 \cdot (x2 - xc) + y2 \cdot (y2 - yc)}{R} = R \cdot \cos(a2)
\[
\begin{bmatrix}
  y_c \\
  x_c \\
  R \\
  x_2 \\
  y_2 \\
\end{bmatrix} = \text{FIND}(y_c, x_c, R, x_2, y_2)
\]

**Solutions:**

\[
\begin{align*}
y_c &= -200.313 \\
x_c &= -269.545 \\
R &= 475.164 \\
x_2 &= 193.573 \\
y_2 &= -94.936
\end{align*}
\]
APPENDIX B

Program which initializes the inlet boundary conditions for the computational diffuser
FORTRAN Program
* PROGRAM WHICH CALCULATES THE INLET BOUNDARY CONDITIONS *
* FOR THE COMPUTATIONAL DIFFUSER *
* *
* OUTPUT: *
* VRAD2 : RADIAL VELOCITY (m/s) *
* V2 : ABSOLUTE VELOCITY (m/s) *
* VTHETA2 : TANGENTIAL VELOCITY (m/s) *
* TT2 : STAGNATION TEMPERATURE (K) *
* T2 : STATIC TEMPERATURE (K) *
* RHO2 : DENSITY (kg/m^3) *
* M : MASS FLOW RATE (kg/s) *
* ******************************************************************

DOUBLE PRECISION A,B,C,D,E,F,A1,B1,C1,K,B2,M
DOUBLE PRECISION INC,PROD,MDOT
DIMENSION PT2(20)
DIMENSION A2(20)
DIMENSION A2R(20)
DIMENSION B2(20)
DIMENSION INC(20)
DIMENSION M(20)

***** Given stagnation pressure from Dalbert et al. *****
DATA PT2 /1383.0,1409.0,1428.0,1456.0,1483.0,1520.0, *
* 1553.0,1583.0,1608.0,1623.0,1623.0,1617.0, *
* 1607.0,1601.0,1600.0,1599.0,1597.0,1564.0, *
* 1536.0,1493.0/

***** Adjusted flow angles (4.1 degrees) *****
DATA A2 /0.2,0.6,1.5,5.4,10.1,14.8,18.4,20.0,21.9, *
* 22.7,23.8,25.4,25.8,26.7,28.0,28.6,28.2, *
* 28.0,27.0,25.4/

***** Relative span of the diffuser from hub to shroud *****
DATA B2 /0.000,0.025,0.060,0.120,0.175,0.235,0.295, *
* 0.355,0.418,0.475,0.538,0.593,0.655,0.713, *
* 0.773,0.835,0.895,0.948,0.973,1.000/

***** Incremental distance for summing the mass flow rate *****
DATA INC /0.21,0.504,0.798,0.966,0.966,1.008,1.008, *
* 1.0332,1.008,1.008,0.9912,0.9828,1.008, *
* 0.9912,1.0248,1.0248,0.9492,0.6552,0.4368, *
* 0.2268/

OPEN (UNIT = 1, FILE = './DISKB/dico/inlet.out')

WRITE (1,10) 'b/b2', 'P2', 'Pt2', 'alpha2', 'Vr2', 'V2', *
* 'Vth2', 'Tt2', 'T2', 'rho2'

* A,6X,A)
GIVEN:
P2 = 1280.0
TT1 = 297.15
U2 = 272.11
R2 = 0.147
CP = 1003.5
R = 287.0
K = 1.40
PI = 3.14159

DO 20 I = 1, 20
    A2R(I) = A2(I)*(PI/180.0)

*** Equations to set up Equation 5.1.9 in the form of the quadratic formula to solve for the radial velocity. ***

A = (P2/PT2(I))**((K-1)/K)
B = TT1
C = U2/(CP*(TAN(A2R(I))))
D = TT1
E = 1.0/(2.0*CP*((SIN(A2R(I))**2.0))
F = U2/(CP*(TAN(A2R(I))))

A1 = E
B1 = (A*C)-F
C1 = (A*B)-D

***** Equations to solve for the inlet conditions *****

VRAD2=(-B1+(B1**2.0-(4.0*A1*C1)**0.5))/(2.0*A1)
V2=VRAD2/SIN(A2R(I))
VTETA2=VRAD2/TAN(A2R(I))

TT2=TT1+((U2*VTETA2)/CP)
T2=TT2-((V2**2.0)/(2.0*CP))

RHO2=(P2*100.0)/(R*T2)
M(I)=(RHO2*VRAD2)*INC(I)

WRITE (1,30) B2(I),P2,PT2(I),A2(I),VRAD2,V2,VTETA2,TT2,T2,RHO2

20      CONTINUE

30      FORMAT (F8.3,3F8.1,5F8.2,F8.4)

***** Summing the mass flow rate *****

PROD = 0
DO 40 I = 1, 20
    PROD = PROD + M(I)
40    CONTINUE

MDOT = 2.0*PI*R2*(PROD/1000.0)
WRITE (1,50) 'mass flow rate (kg/s) = ', MDOT
50    FORMAT (3X,A,F8.4)

END
VITA

Dikran Akseraylian was born to Garabed and Armen Akseraylian on October 7, 1972 in Beirut, Lebanon. He moved to Richmond, Virginia a year later. He received his Bachelor of Science Degree in Mechanical Engineering from The Virginia Military Institute in 1994. In August of 1994, he began his graduate studies in Mechanical Engineering at Virginia Polytechnic Institute and State University.