3-D CFD analysis of flow in a mixed-flow pump diffuser

Yi Zhao
3-D CFD Analysis of Flow in a Mixed-Flow Pump Diffuser

by

Yi Zhao

A Thesis Submitted in Partial Fulfillment of the Requirement of the Degree of Master of Science in Mechanical Engineering

Approved by: __________________________
Dr. A. Ogut - Thesis Advisor

____________________________
Dr. R. J. Hefner

____________________________
Dr. A. Nye

____________________________
Dr. C. Haines

Department of Mechanical Engineering
College of Engineering
Rochester Institute of Technology
Rochester, New York 14623
1996
Permission Denied

Title of thesis: 3D CFD Analysis of a Mixed-Flow Pump Diffuser

I, Yi Zhao, hereby deny permission to the Wallace Library of the Rochester Institute of Technology to reproduce my thesis in whole or in part.

Date: Oct 23 96  Signature of Author: _____________________
Dedication

I would like to dedicate this work to a very special person in my life who has always been my side through all the unforgettable years, and who has always encouraged me to strive ever upwards, my dear Mother.
Acknowledgments

I would like to thank the Rochester Institute of Technology’s Mechanical Engineering Department faculty of the for the instruction and support of my education in the field of Mechanical Engineering. Especially, I want to thank Dr. Ali Ogut for his guidance as my thesis advisor and Mr. Paul Ruzicka in Goulds Pump Inc. for his support.

I would also like to thank Dr. Robert Hefner for his continuous support through my school years at RIT, especially for giving me the opportunity to work on the Surface Condenser Study Project.

Yi Zhao

October, 1996
ABSTRACT

The primary goal of the project was to gain insight into the flow pattern in a mixed-flow pump diffuser by using FLUENT CFD code and k-ε turbulence model, and improve the design of a diffuser for higher pump efficiency.

The diffuser chosen in this thesis is used in a mixed flow multi-stage pump which collects the nearly radial flow (water) at impeller discharge and converts it to a nearly axial flow at the inlet of the following pump stage. A three-dimensional computational model was developed for three different diffuser designs labeled as Diffuser A, Diffuser B and Diffuser C. CFD results indicated that Diffuser C had better pressure recovery characteristics than both Diffuser A and Diffuser B. These results were confirmed by comparing with the experimental data for these diffusers provided by the pump manufacturer.

The major conclusion resulting from the flow pattern study for Diffuser A, B and C is that the exit angle of the mean flow near the hub side of the diffuser has a strong effect on the diffuser performance. The greater the flow exit angle, the higher the degree of secondary flow formation which tends to reduce static pressure recovery. Based on that finding, several different diffuser geometries were modeled in an effort to reduce the exit flow angle.

It can be concluded from this work that FLUENT CFD code can be used to model internal subsonic flows with a high degree of confidence. There was good correlation between model results and manufacturer’s test data.
# Table of Contents

List of Figures...........................................................................................................iv

List of Tables...........................................................................................................vii

List of Symbols.........................................................................................................viii

1. Introduction

1.1 Project Background...............................................................................................1

1.2 Project Objectives...............................................................................................1

1.3 Project Description..............................................................................................2

1.4 Literature Search.................................................................................................4

2. Turbulent Flow Theory

2.1 Governing Equations..........................................................................................5

2.2 Methodology of Analysis....................................................................................6

2.3 Diffuser Flow.......................................................................................................10

2.4 Secondary Flow..................................................................................................12

3. FLUENT Computational Fluid Dynamics Analysis Code

3.1 FLUENT Code Introduction................................................................................14

3.1.1 Code Application and Features.......................................................................14

3.1.2 Program Structure.........................................................................................15

3.1.3 Modeling Technique......................................................................................15

3.2 Diffuser Modeling Process................................................................................18

3.2.1 Geometry Creation.........................................................................................18
3.2.2 Grid Generation .............................................. 20
3.2.3 Case File Generation ......................................... 23
3.2.4 Computing the Results ...................................... 24

4. Results and Discussions

4.1 General CFD Results ........................................... 25
  4.1.1 Alphanumeric Illustration ................................. 25
  4.1.1 Graphical Illustration ..................................... 27
4.2 Flow Pattern Study ............................................. 28
  4.2.1 Diffuser A .................................................. 28
  4.2.2 Diffuser B .................................................. 32
  4.2.3 Diffuser C .................................................. 35
  4.2.4 Flow Pattern Comparison Of Diffuser A, B & C ...... 38

5. Conclusions and Recommendations ................................ 39

  5.1 Conclusions .................................................. 39
  5.2 Recommendations ............................................ 40

Reference ............................................................ 41
List of Figures

1.1 Mixed-Flow Pump
2.1 Near Wall Model
2.2 Sample Diffuser Geometry
2.3 Schematic of Shroud to Hub Curvature Caused Secondary Flows
2.4 Schematic of PS to SS Curvature Caused Secondary Flows
3.1 Schematic of Diffuser Location Inside a Mixed Flow Pump
3.2 Surface Grid of Diffuser A
3.3 Surface Grid of Diffuser B
3.4 Surface Grid of Diffuser C
3.5 Twisting (Chord) Angle Comparison
3.6 Diffuser Intersection Angle (Between Outlet & SS on Hub) Illustration
4.1 Area Progression of Diffuser A, B & C
4.2 Static Pressure Recovery of Diffuser A, B & C
4.3 Total Pressure Loss of Diffuser A, B & C
4.4A Diffuser A: Velocity Vector Plot of Slice J=2 (Near Hub)
4.4B Diffuser A: Zoom View of Velocity Vector Plot at Outlet of Slice J=2 (Near Hub)
4.4C Diffuser A: Static Pressure Distribution Filled Contour of Slice J=2 (Near Hub)
4.5A Diffuser A: Velocity Vector Plot of Slice J=19 (Near Shroud)
4.5B Diffuser A: Static Pressure Distribution Filled Contour of Slice J=19 (Near Shroud)
4.6A Diffuser A: Velocity Vector Plot of Slice J=11 (Mid-Spanwise)
4.6B  Diffuser A: Static Pressure Distribution Filled Contour of Slice J=11 (Mid-Spanwise)

4.7A  Diffuser A: Velocity Vector Plot of Slice I=2 (Near SS)

4.7B  Diffuser A: Static Pressure Distribution Filled Contour of Slice I=2 (Near SS)

4.8A  Diffuser A: Velocity Vector Plot of Slice I=19 (Near PS)

4.8B  Diffuser A: Static Pressure Distribution Filled Contour of Slice I=19 (Near PS)

4.9A  Diffuser A: Velocity Vector Plot of Slice I=11 (Mid-Pitchwise)

4.9B  Diffuser A: Static Pressure Distribution Filled Contour of Slice I=11 (Mid-Pitchwise)

4.10A Diffuser A: Static Pressure Distribution Filled Contour of Slices K=1, 30, 50, 70, 100

4.11A Diffuser A: Velocity Vector Plot of Slice K=100 (Outlet)

4.11B Diffuser A: Velocity Vector Plot of Slice K=100 Side View (Outlet)

4.11C Diffuser A: Static Pressure Distribution Filled Contour of Slice K=100 (Outlet)

4.12A Diffuser B: Velocity Vector Plot of Slice J=2 (Near Hub)

4.12B Diffuser B: Zoom View of Velocity Vector Plot at Outlet of Slice J=2 (Near Hub)

4.12C Diffuser B: Static Pressure Distribution Filled Contour of Slice J=2 (Near Hub)

4.13A Diffuser B: Velocity Vector Plot of Slice J=19 (Near Shroud)

4.13B Diffuser B: Static Pressure Distribution Filled Contour of Slice J=19 (Near Shroud)

4.14A Diffuser B: Velocity Vector Plot of Slice J=11 (Mid-Spanwise)

4.14B Diffuser B: Static Pressure Distribution Filled Contour of Slice J=11 (Mid-Spanwise)

4.15A Diffuser B: Velocity Vector Plot of Slice I=2 (Near SS)

4.15B Diffuser B: Static Pressure Distribution Filled Contour of Slice I=2 (Near SS)

4.16A Diffuser B: Velocity Vector Plot of Slice I=19 (Near PS)
4.16B Diffuser B: Static Pressure Distribution Filled Contour of Slice I=19 (Near PS)

4.17A Diffuser B: Velocity Vector Plot of Slice I=11 (Mid-Pitchwise)

4.17B Diffuser B: Static Pressure Distribution Filled Contour of Slice I=11 (Mid-Pitchwise)

4.18A Diffuser B: Static Pressure Distribution Filled Contour of Slices K=1, 30, 50, 70, 100

4.19A Diffuser B: Velocity Vector Plot of Slice K=100 (Outlet)

4.19B Diffuser B: Velocity Vector Plot of Slice K=100 Side View (Outlet)

4.19C Diffuser B: Static Pressure Distribution Filled Contour of K=100 (Outlet)

4.20A Diffuser C: Velocity Vector Plot of Slice J=2 (Near Hub)

4.20B Diffuser C: Zoom View of Velocity Vector Plot at Outlet of Slice J=2 (Near Hub)

4.20C Diffuser C: Static Pressure Distribution Filled Contour of Slice J=2 (Near Hub)

4.21A Diffuser C: Static Pressure Distribution Filled Contour of Slices K=1, 30, 50, 70, 100

4.22A Diffuser C: Velocity Vector Plot of Slice K=100 (Outlet)

4.22B Diffuser C: Velocity Vector Plot of Slice K=100 Side View (Outlet)

4.22C Diffuser C: Static Pressure Distribution Filled Contour of K=100 (Outlet)

4.23 Ideal Diffuser Flow Pattern

4.23A Velocity Vector Plot Comparison of Diffusers A, B & C of Slice J=2 (Near Hub)

4.23B Static Pressure Distribution Comparison of Diffusers A, B & C of Slice J=2 (Near Hub)

**List of Tables**

3.1 Model Geometry Comparison

4.1 Summary of Results

4.2 Test Data for Diffuser A & B

4.3 Comparison of CFD Results & Test Data
### List of Symbols

**Nomenclature**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A$</td>
<td>Net Flow Cross Area</td>
</tr>
<tr>
<td>$AR$</td>
<td>Area Ratio</td>
</tr>
<tr>
<td>$C_{1.2}$</td>
<td>Empirical Constants in $\varepsilon$ and $k$ equations</td>
</tr>
<tr>
<td>$C_{\mu}$</td>
<td>Empirical Constant in Viscosity Model</td>
</tr>
<tr>
<td>$C_p$</td>
<td>Pressure Recovery Coefficient</td>
</tr>
<tr>
<td>$C_{p,1}$</td>
<td>Ideal Pressure Recovery Coefficient</td>
</tr>
<tr>
<td>$E$</td>
<td>Log Law Constant</td>
</tr>
<tr>
<td>$G$</td>
<td>Shear Generation Term</td>
</tr>
<tr>
<td>$k$</td>
<td>Turbulent Kinetic Energy</td>
</tr>
<tr>
<td>$k_v$</td>
<td>Von Karmans' Constant</td>
</tr>
<tr>
<td>$k_p$</td>
<td>Near Wall Turbulent Kinetic Energy</td>
</tr>
<tr>
<td>$I$</td>
<td>Turbulence Intensity</td>
</tr>
<tr>
<td>$L$</td>
<td>Diffuser Wall Length</td>
</tr>
<tr>
<td>$I$</td>
<td>Turbulence Length Scale</td>
</tr>
<tr>
<td>$P$</td>
<td>Fluid Pressure</td>
</tr>
<tr>
<td>$P^*$</td>
<td>Dimensionless Pressure</td>
</tr>
<tr>
<td>$Q$</td>
<td>Flow Rate [Gpm]</td>
</tr>
<tr>
<td>$R$</td>
<td>Radius of streamline</td>
</tr>
<tr>
<td>$Re$</td>
<td>Reynolds Number</td>
</tr>
</tbody>
</table>
\[ t \quad \text{Time} \]
\[ u' \quad \text{Friction Velocity} \]
\[ u_i \quad \text{Velocity in i-direction} \]
\[ u^- \quad \text{Dimensionless Velocity} \]
\[ u_p \quad \text{Near Wall Velocity} \]
\[ V \quad \text{Diffuser Inlet Fluid Velocity} \]
\[ V_m \quad \text{Meridional Velocity} \]
\[ V_n \quad \text{Normal Velocity (velocity component in the direction of diffuser channel)} \]
\[ V_t \quad \text{Tangential Velocity (velocity component orthogonal to the direction of diffuser channel)} \]
\[ W \quad \text{Diffuser Width} \]
\[ y^- \quad \text{Dimensionless Normal Distance from Wall} \]

**Greek Letters**

\[ \eta \quad \text{Diffuser Efficiency; Arbitrary Field Variable} \]
\[ \varepsilon \quad \text{Dissipation Rate of } k \]
\[ \theta \quad \text{Divergence Angle of Diffuser} \]
\[ \delta \quad \text{Boundary Layer Thickness} \]
\[ \rho \quad \text{Fluid Density} \]
\[ \pi \quad \text{Pi [3.14159...]} \]
\[ \omega \quad \text{Vorticity} \]
\[ \tau \quad \text{Shear Stress} \]
\( \sigma \)  Empirical Constants

\( \sigma_e \)  Turbulent Prandtl Number

\( \mu \)  Absolute Viscosity

**Subscripts**

\( l \)  Inlet Node

\( 2 \)  Outlet Node

\( I \)  Grid Index in the Spanwise Direction

\( J \)  Grid Index in the Pitchwise Direction

\( K \)  Grid Index in the Streamwise Direction

\( n \)  Normal Direction

\( b \)  Binormal Direction

\( s \)  Streamwise Direction
Chapter 1 Introduction

1.1 Project Background

Diffusers are one of the basic components of centrifugal pump systems. The primary function of a diffuser is to convert the inlet dynamic pressure (kinetic energy) to a static pressure rise. For subsonic flow this is done by decelerating the fluid particles by providing a continuous and gradual increase of the cross-sectional flow area. The desired effect is to recover as much of the inlet dynamic pressure as possible with steady flow conditions.

The scope of this thesis covers the analysis of a turbine-bowl diffuser used in a mixed flow multi-stage pump, and to investigate the applicability of using CFD code as an aid in designing effective diffusers. The multistage pump has a 15 inch outside bowl diameter, and the pump has a specific speed of 4750. (Specific speed is a dimensionless coefficient for compare certain aspects of the different families of turbomachines.)

In order to have optimum performance, a mixed-flow pump, not only requires an optimally designed impeller but also a matching designed diffuser. Currently, there are two versions of the diffuser: Diffuser A and Diffuser B. Diffuser B was shaped by untwisting Diffuser A along the bowl axis approximately five degrees.

1.2 Project Objectives

Present diffuser design methods lack theoretical depth and sufficient empirical data for determining the optimum geometry. This lack of knowledge is in part due to the complex nature of flow in diffusers, and is also due to the difficulty of making the required fluid dynamics analysis needed to correlate theory and experiments.
The project goal at RIT is to modify a mixed flow pump's diffuser geometry to improve the discharge performance by using FLUENT computational fluid dynamics analysis method. The task of this thesis was to develop a three-dimensional turbulent model of current designs as base models, then modify these designs for improvements using FLUENT. The method of increasing the diffuser performance, however should not significantly alter the diffuser geometry.

1.3 Project Description

All centrifugal pumps utilize but one pumping principle: the rotating impeller imparts energy to fluid, building up a velocity head. At the periphery of the pump casing, the fluid is directed into a diffuser. The diffuser most often has a constantly increasing cross-sectional area along its length, so that as the fluid proceeds along the channel its velocity is reduced. Because the energy level of the fluid cannot be substantially dissipated at this point, the conservation of the energy requires that when the fluid loses kinetic energy as it moves along the channel, it must increase the energy related to pressure. That is, the pressure of the fluid increases.

A mixed flow pump, (a type of centrifugal pump), develops head partly by centrifugal force and partly by the lift of the vanes on the liquid. This type of pump has a single inlet impeller with the flow entering axially and discharging in both axial and radial directions. Pumps of this type usually have a specific speed from 4200 to 9000.

Figure 1.1 shows the basic configuration of a mixed flow pump.
The turbine-bowl diffuser in this thesis consists of eight passages which are separated by eight vanes. In order to reduce computational effort, a three-dimensional computational model in FLUENT was developed for one passage of each diffuser design.

Based on the CFD results for Diffuser A and B, different geometries were constructed and modeled in order to improve the design. After consultation with the manufacturer, one of the modeled geometries was chosen and analyzed in this thesis as the final improved design. This is labeled as Diffuser C. The design of Diffuser C is actually an optimized design of Diffusers A and B, constructed by enlarging the intersecting angle between suction side wall and outlet in the hub from $90^\circ$ to $108^\circ$. This modification challenges the traditional diffuser design practice in the industry which is believed that the optimum design for vane passageways to reduce losses is a channel that is as square as possible.
1.4 Literature Search

In spite of an extensive effort for literature search via library and Internet, no article was found related to the subject of mixed flow pump diffuser. However, the research related to mixed flow pump and diffuser computer modeling and flow analysis was insightful which is listed as following:

- Zhang and Sun (1995) presented a method based on a 3-D viscous flow analysis for the performance prediction for the mixed-flow pump\(^1\)
- Favre (1995) introduced a full 3-D flow modeling method to study the mixed-flow pump impeller performance\(^2\)
- Zhang and Garon (1993) presented a 3-D simulation of the passage-averaged vorticity-potential formulation of the incompressible viscous flow field within a mixed-flow pump\(^3\)

******** This area intentionally left blank. ********
Chapter 2 Turbulent Flow Theory

2.1 Governing Equations

Turbulence is one of the most difficult phenomena in the area of physical sciences. In turbulent flow situations, the fluid motion is highly random, unsteady, and three-dimensional. Due to these complexities, the turbulent motion and mass-transfer phenomena associated with it are extremely difficult to describe and thus predict theoretically. It is believed that the solution of the time-dependent three-dimensional Navier-Stokes equations can completely describe turbulent flows. This can be done by describing the fluid flow at every point in the flow regime for all time by taking into consideration the principles of conservation of mass, momentum and energy. These general equations completely describe the fluid flow (the equation for energy conservation is disregarded here due to its irrelevance to the problem).

The nature of the specific flow under consideration allows for some simplification. It is assumed that within the diffuser, the flow can be described as turbulent, steady, incompressible, isothermal, and Newtonian in nature. The conservation of energy equation is disregarded based upon these assumptions, and the equations for conservation of mass and momentum are reduced to:

\[
\frac{\partial u_i}{\partial x_j} = 0 \quad \text{ (Eq. 1)}
\]

\[
\rho \left[ u_j \frac{\partial u_i}{\partial x_j} \right] = -\frac{\partial P}{\partial x_i} + \rho F_i + \frac{\partial}{\partial x_i} \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \rho u_i u_j \right] \quad \text{ (Eq. 2)}
\]
where \( u_i \) is the fluid velocity with \( i, j = 1, 2, \& 3 \) for a three-dimensional problem, \( P \) is the pressure, and \( F \) represents body forces.

### 2.2 Methodology of Analysis

To practically describe turbulent motion, it is necessary to use time averaged quantities rather than instantaneous ones. This approach is based upon the conservation laws for mass and momentum (Eq. 1 and 2). Osborne Reynolds was the first to suggest using a statistical approach where the equations are averaged over a time scale which is comparatively long with that of the turbulent event in question. The resulting equations describe the distribution of the mean velocity and pressure within the control volume.

In this statistical approach, each of the field variables (velocity, \( u_i \), & pressure, \( P \)) are separated into mean and fluctuating quantities which allows for the use of mean values of the field variables (\( \overline{u}_i \) & \( \overline{P} \)) in modeling the large scale flow characteristics. For an arbitrary field variable (\( \eta \)), the mean value can be defined as

\[
\overline{\eta} = \frac{1}{\Delta t} \int_{t}^{t+\Delta t} \eta \, dt
\]

(Eq. 3)

where the averaging time \( \Delta t \) is long compared with the time scale of the turbulent motion. The instantaneous variable \( \eta \) is then given by,

\[
\eta = \overline{\eta} + \eta'
\]

(Eq. 4)

where \( \overline{\eta} \) is the time averaged quantity and \( \eta' \) reflects the small scale fluctuations associated with turbulence. This decomposition is applied to the Navies-Stokes equations which are then integrated over the time interval \( (t, t + \Delta t) \) resulting in the following time averaged equations (Eq. 5 & 6).
Due to the non-linearity of the Navies-Stokes equations, the averaging process introduces a correlation between fluctuating velocities $\overline{u_i u_j}$. Multiplying this term by $\rho$ gives the transport of momentum due to the turbulent motion. The relation

$$\rho \overline{u_i u_j} = \frac{1}{\Delta t} \int_{\tau}^{\tau + \Delta t} \rho u_i u_j \, d\tau$$

(Eq. 5)

describes the transport of $x_i$ momentum in the direction of $x_j$, and acts as a stress on the fluid (Reynolds stress). It summarizes the effect of small scale eddy behavior on the large scale mean flow. To solve the Navies-Stokes equations and Eq. 5 requires a way of determining the turbulence correlation. This determination is the main roadblock in analyzing turbulent flows. A turbulence model which approximates this correlation along with the Navies-Stokes equations forms a closed set of equations which can be solved for the mean values of velocity and pressure.

Generally, the two equation $k$-$\varepsilon$ turbulence model is employed to facilitate the solution, where $k$ stands for turbulent kinetic energy and $\varepsilon$ stands for dissipation rate. In the $k$-$\varepsilon$ model, Reynolds stresses are related to the mean flow via the Boussinesq hypothesis:

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

(Eq. 6)

The effective or "turbulent" viscosity, $\mu_t$, is computed from a velocity scale ($k^2$) and a length scale ($\frac{k^3}{\varepsilon}$) which are predicted at each point in the flow via solution of transport equations for $k$ and $\varepsilon$. 
\[ \frac{\partial}{\partial t} (\rho k) + \nabla \cdot (\rho u_k k) = \frac{\partial}{\partial x_j} \left[ \mu_t \frac{\partial k}{\partial x_j} \right] - G_k + G_\varepsilon - \rho \varepsilon \]  

(Eq. 7)

and

\[ \frac{\partial}{\partial t} (\rho \varepsilon) + \nabla \cdot (\rho u \varepsilon) = \frac{\partial}{\partial x_j} \left[ \mu_t \frac{\partial \varepsilon}{\partial x_j} \right] - C_1 \rho \varepsilon G_k - C_2 \rho \varepsilon^2 k \]

where \( G_k \) is the generation of \( k \) and is given by:

\[ G_k = \mu_t \left( \frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j} \]  

(Eq. 9)

and \( G_\varepsilon \) is the generation due to the buoyancy:

\[ G_\varepsilon = -g \frac{\mu_t}{\rho \sigma_k} \frac{\partial \rho}{\partial x} \]  

(Eq. 10)

where \( \sigma_h \) is the turbulent Prandtl number, \( \frac{\mu_t C_i}{k} \).

The turbulent viscosity is then related to \( k \) and \( \varepsilon \) by the expression:

\[ \mu_t = \rho C_\mu \frac{k^3}{\varepsilon} \]  

(Eq. 11)

The coefficients \( C_1, C_2, C_\mu, \sigma_k, \) and \( \sigma_\varepsilon \) are empirical constants which have the following empirically derived values:

\[ C_1 = 1.44, C_2 = 1.92, C_\mu = 0.09, \sigma_k = 1.0, \sigma_\varepsilon = 1.3 \]

In turbulent flow, the wall boundary layer consists of a laminar sublayer and a so-called log-law region in which the flow is fully turbulent. In the log-law region, the wall shear stress can be computed via the log-law wall function:
\[
\frac{u_p}{u^*} = \frac{1}{k_v} \ln \left( \frac{E}{\gamma} \right)
\]  
(Eq. 12)

where

\( k_v = \text{Von Karmans' constant} (0.42) \)

\( E = \text{log law constant} (9.8) \)

\( u^* = \text{friction velocity} \frac{r_w}{\sqrt{\rho}} \)

\( u_p = \text{near wall velocity} \)

Note that the assumption of equilibrium in the boundary layer (production equal to dissipation) can be used to derive the following expression for \( y \) (dimensionless normal distance from wall):

\[
y = \frac{\rho k_p^2 C_n^4 \Delta y}{\mu}
\]  
(Eq. 13)

where: \( k_p = \text{near wall turbulent kinetic energy} \)

\( \mu = \text{fluid viscosity} \)

\( \Delta y = \text{distance to the wall} \)

In general, the inlet turbulence intensity and characteristic length are a function of the flow parameters upstream. The turbulence intensity is defined as the ratio of the turbulent fluctuations in velocity to the mean flow velocity \( (u' / u_{avg}) \), expressed as a percentage. The inlet values of \( k \) and \( \varepsilon \) are calculated from the specified inlet turbulence intensity, \( I \) as follows:
Figure 2.1 Near Wall Model
\[ k = \frac{3}{2} (u')^2 \quad \text{or} \quad k = \frac{3}{2} (u_{\infty} l)^2 \]  
(Eq. 14)

The dissipation rate is then given by:

\[ \varepsilon = C_\mu^4 \left( \frac{k}{l} \right)^{\frac{3}{2}} \]  
(Eq. 15)

where \( l \) is a length scale characteristic of the turbulence in the inlet flow. The characteristic length is used to compute the mixing length for the small-scale eddies and should be set to the hydraulic radius of the inlet. The inlet turbulence length scale, \( l \), is calculated by:

\[ l = 0.07 R \]  
(Eq. 16)

Note that the factor of 0.07 is derived from the "average" mixing length in turbulent pipe flow, where \( R \) is the radius of the pipe.

The turbulent flow regime can be described by distinct regions based upon the definitions and characteristics from above. The viscous sublayer is the region nearest to the wall where \( y^+ \) is less than or equal to five. The fully turbulent core is near the centerline of the flow where \( y^+ \) is greater than 30. The buffer region is located between the viscous sublayer and the fully turbulent region. Figure 2.1 shows these regions in a graphical form. The regions are defined by the different flow characteristics that are found within each region, which is helpful in discussing the complexities of turbulent flow.

2.3 Diffuser Flow

One of the basic components of a pump is the diffuser. The diffuser's purpose is to convert the inlet dynamic pressure of the fluid to a static pressure rise. For subsonic flow, this is accomplished by decelerating the fluid particles by the application of a gradual
Figure 2.2 Sample Diffuser Geometry
increase of the cross sectional flow area. It is desirable to recover as much of the entering
dynamic pressure as possible. It is also important that the exiting flow be steady and has a
uniform profile for the next impeller stage.

There are several parameters used to describe a diffuser geometry. These
quantities are useful in analyzing the performance of the diffuser. A simple flat diffuser is
shown in Figure 2.2. The geometry of a diffuser is specified by the aspect ratio \( b/W_1 \), the
divergence angle \( 2\theta \), the length-to-width ratio \( L/W_1 \), and the cross-sectional area ratio
\( W_2/W_1 \). The pressure recovery coefficient \( C_p \) describes the performance of a diffuser:

\[
C_p = \frac{P_2 - P_1}{\frac{1}{2} \rho v_t^2}
\]  
(Eq. 17)

where \( P_2 \) is the outlet static pressure, \( P_1 \) is the inlet static pressure, and \( v_t \) is the throat
velocity. Under ideal conditions, the maximum pressure recovery coefficient \( C_{p,\text{ideal}} \) is a
function of the geometry and is given by

\[
C_{p,\text{ideal}} = 1 - \frac{1}{AR^2}
\]  
(Eq. 18)

where \( AR \) is the area ratio. The ratio of the actual pressure recovery coefficient to the ideal
pressure recovery coefficient is known as the diffuser efficiency \( \eta \).

\[
\eta = \frac{C_p}{C_{p,\text{ideal}}}
\]  
(Eq. 19)

In the diffuser, the development of the turbulent boundary layer has a significant
impact on the diffuser performance. If the turbulent boundary layer is thick enough to
create a large throat blockage, separation will occur near the inlet of the diverging section.
The fluid particles decelerate near the wall region under the effect of an increasing pressure gradient and reduced transverse momentum transfer. As the fluid progresses through the diffuser, in the presence of flow separation, excessive blockage occurs, which in turn reduces the diffuser efficiency.

2.4 Secondary Flows

Due to the diffuser wall 3-D curvature, secondary flows can be easily found in a mixed-flow pump diffuser. The cause of secondary flows can be traced to the dynamics of streamwise vorticity, with reference to a right-hand reference system composed of streamwise (s), normal (n), and binormal(b) directions. Under the hypotheses of incompressible stationary flow, the production of streamwise vorticity is:

\[
\frac{\partial}{\partial s} \left( \frac{\omega_s}{v} \right) = \frac{2 \omega_n}{Rv}
\]

where \( \omega \) is vorticity, \( v \) is absolute velocity, and \( R \) is radius of streamline. Eq. 20 can be reduced by use of the Bernoulli’s equation and scalar product in direction b to:

\[
\frac{\partial}{\partial s} \left( \frac{\omega_s}{pv} \right) = \frac{2}{Rpv} \frac{\partial v}{\partial b} = \frac{2}{Rpv^3} \frac{\partial P_t}{\partial b}
\]

where \( P_t \) stands for total pressure.

The production of streamwise vorticity depends thus on the gradients of velocity in binormal direction. These gradients of velocity are typically associated with the presence of boundary layers, which are encountered on the hub and shroud walls as well as on the suction and pressure side walls. It is possible to separate the effects of shroud to hub and pressure to suction side wall curvatures:
a) Shroud to Hub Curvature  Figure 2.3 is a schematic of the mixed flow pump diffuser, the normal \( n \) points inward in radial direction, that is from shroud to hub; the binormal \( b \) is in tangential direction from pressure side (PS) to suction side (SS). Under these conditions, \( (\partial \tau / \partial b) \) is positive on PS (total pressure is increased at the outer PS boundary layer), and negative on the SS (total pressure is decreased at the inner SS boundary layer). Vorticity production (clockwise; displacing \( n \) towards \( b \)) is positive on PS and negative (counterclockwise, displacing \( b \) towards \( n \)) on SS. This means that the direction of secondary flows will be from shroud to hub both on both PS and SS wall surface: low-total pressure fluid accumulates in the hub boundary layer and then recirculates to the shroud at the center of the diffuser passage. This pattern, shown by dashed lines in Figure 2.3 is responsible for the rapid buildup of a large hub boundary layer, which is then subject to further secondary effects.

b) PS to SS  For PS to SS curvature (Figure 2.4), the normal \( n \) is directed in tangential direction from PS to SS; the binormal \( b \) points outward in radial direction, that is from hub to shroud. Under these conditions, \( (\partial \tau / \partial b) \) is positive at the hub (total pressure is increased at the outer hub boundary layer), and negative at the shroud (total pressure is decreased at the inner shroud boundary layer). Vorticity production (clockwise; displacing \( n \) towards \( b \)) is positive at the hub and negative (counterclockwise; displacing \( b \) towards \( n \)) at the shroud. This would mean that secondary flows should circulate from PS to SS at the both hub and shroud (dashed lines in Figure 2.4).
Figure 2.3 Schematic of Shroud to Hub Curvature Caused Secondary Flows
Figure 2.4 Schematic of PS to SS Curvature Caused Secondary Flows
Chapter 3 FLUENT Description and Implementation

3.1 FLUENT CFD code introduction

3.1.1 Code Application and Features

The use of advanced computational fluid dynamics techniques is very helpful in the analysis of complex flow patterns encountered within a centrifugal pump's subcomponents. In this work FLUENT was used for such a purpose. FLUENT is a general purpose computational program for modeling fluid flow, heat transfer, and chemical reactions. FLUENT models this wide range of phenomena by solving the conservation equations for mass, momentum, energy, and chemical species using a control volume based finite-difference method. The governing equations are discretized on a curvilinear grid to enable computations in complex/irregular geometries. A nonstaggered system is used for storage of discrete velocities and nodal pressures. Interpolation is accomplished via a first-order, Power-Law scheme or optionally via the higher order QUICK scheme. The equations are solved using the SIMPLEC algorithm with an iterative line-by-line matrix solver and multigrid acceleration or with the GMRES full-field iterative solver.

In this thesis work, FLUENT was used to analyze regions of stagnation, flow separation, and secondary flow patterns within the various diffuser configurations. The flow was modeled in three dimensions and accounted for the incompressibility and viscous nature of the flow. FLUENT presents several options for the solution of the Reynolds averaged Navier-Stokes equations for turbulent flow. This solution incorporates the two-equation kinetic energy dissipation turbulence \(k-\varepsilon\) model. In this turbulence model, the
turbulence field is characterized in terms of two variables, the turbulent kinetic energy, and the viscous dissipation rate of kinetic energy. The criteria for effectively using finite difference method are numerous in order to ensure a solution that converges and gives realistic results. Among the most important are the choices of grid density and cell types, inlet and boundary conditions, solution techniques, and the extension of the kinetic energy-dissipation model to the near-wall region.

3.1.2 Program Structure

FLUENT is a two part program consisting of a preprocessor - PreBFC, and a main module FLUENT. PreBFC is used to define the geometry and a structured grid for the model. Then the grid information is transferred from PreBFC to FLUENT via a GRID File. Following this transfer, FLUENT is used to define physical models, fluid/material properties, and boundary conditions that describe the problem. This information is added to the grid information and stored in a Case File that is a record of all the inputs for problem definition. Calculation and post-processing are also performed in FLUENT, the results are stored in a Data File.

3.1.3 Modeling Technique

3.1.3.1 Problem Solving Steps

Once the important features of the problem are determined, the basic procedural steps are those shown below:

1. Create or import the model geometry and grid.

2. Choose the basic equations to be solved (i.e. enthalpy, species, turbulence transport).
3. Identify additional models needed (fans, porous media, special boundary conditions, species transport or chemical reaction, etc.).

4. Specify the boundary conditions.

5. Specify the fluid properties.

6. Set up a dispersed phase (optional).

7. Adjust the solution control parameters (optional).

8. Calculate a solution (fluid phase and/or dispersed phase).

9. Examine the results.

10. Save the results.

11. Consider revisions to the numerical or physical model.

3.1.3.2 Choosing a Suitable Grid

The grid represents a discrete approximation of the continuous field phenomena that users must model. The accuracy and numerical stability of the FLUENT calculations depend on these grid characteristics. In other words, the density and distribution of the grid lines determines the accuracy with which the FLUENT model represents the actual physical phenomena.

In FLUENT, the control volume method, sometimes referred to as the finite volume method, is used to discretize the transport equations. In the discrete form of the equations, values of the dependent variables appear at control volume boundary locations. These values have to be expressed in terms of the values at the nodes of neighboring cells in order to obtain solvable algebraic equations. This task is accomplished via an interpolation
practice known as a “differencing scheme” The choice of differencing scheme not only affects the accuracy of the solution but the stability of the numerical method.

3.1.3.2.1 Grid Spacing near Walls

In turbulent flow, the spacing between the wall and the adjacent grid line should be such that the grid line lies in the log-law layer of the turbulent boundary layer. This implies a dimensionless distance from the wall, $y^+$, greater than about 25 and less than about 300-500.

3.1.3.2.2 Non-Uniform Grid Spacing

One way to minimize the number of cells while maintaining a sufficient degree of accuracy in the solution is to use a non-uniform grid. In a non-uniform grid, the grid spacing is reduced in regions where high gradients are expected and increased in regions where the flow is relatively uniform.

The rate of change of grid spacing should be minimized due to stability effects that result. Normally the spacing between adjacent grid lines should not change by more than 20% or 30% from one grid line to the next which implies expansion factors between 0.7 and 1.3. This is an accuracy consideration, primarily impacting the accuracy of the diffusion terms in the governing transport equations.

3.1.3.2.3 Cell Aspect Rations

The aspect ratio of the computational cells is an additional issue that arises during the setup of the computational grid. While large aspect ratios may introduce acceptable degrees of error in some problems, a general rule of thumb might be to avoid aspect ratios in excess of 5:1. This limit can be exceeded without significant consequence when the
gradients in one direction are very small relative to those in the second direction. Conversely, excessive aspect ratios can lead to stability problems, convergence difficulties, and the propagation of numerical errors.

3.1.3.2.4 Grid Skewness

When using body-fitted coordinates, the grid lines may not be orthogonal. While some degree of nonorthogonality is allowable, and is accounted for in the solution process, the computational grid should maintain grid intersection angles close to 90 degrees whenever possible.

3.1.3.2.5 Weighting Factors for Grid Redistribution

Weighting factors can be used to control the grid density at each endpoint of a segment. A weighting factor greater than 1.0 implies that the grid density will be increased, thus a finer (denser) grid at the endpoint can easily be achieved.

3.1.3.3 Solver Selection

The default $k - \varepsilon$ model is a semi-empirical model that has been proven to provide engineering accuracy in a wide variety of turbulent flows including flows with planar shear layers such as jet-flows, duct flows, etc. This model was found suitable for the conditions present in this project.

3.2 Diffuser Modeling Process:

Since the flow pattern in each diffuser channel is identical, only one channel was modeled in each of the three diffuser designs.

3.2.1 Geometry Creation:
The geometry data for Diffuser A, B and C is given in Table 3.1. This table provides the axial (Z), radial (R), and chord angle (θ) coordinates for the points along the two sides of one of the eight blades in the diffuser. The inner side is the intersection of SS and the hub side, and the outer side is the intersection of SS and the shroud side. Chordal thickness ($T_h$) of inner & outer side represents the diffuser PS and SS wall thickness. All the data was supplied in English units, thus geometries were constructed in English units in Cadkey and PreBFC and converted into SI units in FLUENT. Figure 3.1 shows the diagram of the diffuser location inside a mixed flow pump.

To reduce the computational and analysis effort, it was critical to find and locate the inlet and outlet plane normal to the diffuser channel. This became one of the most difficult challenges due to the diffuser’s complex geometry. Finding these planes exceeded the capability of the PreBFC package, which constructs geometry only based on coordinates. Cadkey 3-D software was introduced into the project to find the normal planes. The raw geometries were built up based on geometry data, and then the normal inlet and outlet planes were located using the recently introduced surface package offered in Cadkey version 7. The excess portion of the geometry was removed from the model along the normal inlet and outlet planes. The resulting diffuser channel point coordinates were input into the PreBFC to generate the CFD models.

**Diffuser A:** This was the original design. As shown in Figure 3.5, from outlet to inlet: the twisting angle of the PS and SS walls on the hub side ranged from 0 to 62.36 degrees, on the shroud side increased from 3.35 to 77 degrees; the hub radius varied from 1.912 in to 4.875 in, while the shroud radius varied from 4.909 in to 7.026 in.
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>12.507</td>
<td>1.912</td>
<td>0</td>
<td>0.125</td>
<td>14.035</td>
<td>4.909</td>
<td>3.35</td>
<td>0.218</td>
</tr>
<tr>
<td>2</td>
<td>11.864</td>
<td>2.451</td>
<td>1</td>
<td>0.138</td>
<td>12.879</td>
<td>5.475</td>
<td>4.35</td>
<td>0.295</td>
</tr>
<tr>
<td>3</td>
<td>10.866</td>
<td>3.288</td>
<td>10</td>
<td>0.182</td>
<td>11.867</td>
<td>6.055</td>
<td>10</td>
<td>0.39</td>
</tr>
<tr>
<td>4</td>
<td>10.2</td>
<td>3.847</td>
<td>20</td>
<td>0.223</td>
<td>10.806</td>
<td>6.539</td>
<td>20</td>
<td>0.47</td>
</tr>
<tr>
<td>5</td>
<td>9.526</td>
<td>4.359</td>
<td>30</td>
<td>0.254</td>
<td>9.787</td>
<td>6.858</td>
<td>30</td>
<td>0.479</td>
</tr>
<tr>
<td>6</td>
<td>8.786</td>
<td>4.699</td>
<td>40</td>
<td>0.238</td>
<td>8.725</td>
<td>7.056</td>
<td>40</td>
<td>0.491</td>
</tr>
<tr>
<td>7</td>
<td>8.042</td>
<td>4.857</td>
<td>50</td>
<td>0.227</td>
<td>7.672</td>
<td>7.125</td>
<td>50</td>
<td>0.477</td>
</tr>
<tr>
<td>8</td>
<td>7.21</td>
<td>4.875</td>
<td>62.36</td>
<td>0.22</td>
<td>6.347</td>
<td>7.125</td>
<td>62.36</td>
<td>0.432</td>
</tr>
</tbody>
</table>

**Diffuser B**

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>12.507</td>
<td>1.9118</td>
<td>0</td>
<td>0.125</td>
<td>14.035</td>
<td>4.909</td>
<td>3.35</td>
<td>0.1875</td>
</tr>
<tr>
<td>2</td>
<td>11.647</td>
<td>2.6335</td>
<td>1</td>
<td>0.245</td>
<td>12.5411</td>
<td>5.6706</td>
<td>1</td>
<td>0.5</td>
</tr>
<tr>
<td>3</td>
<td>10.9983</td>
<td>3.1779</td>
<td>5</td>
<td>0.255</td>
<td>11.2989</td>
<td>6.3345</td>
<td>5</td>
<td>0.55</td>
</tr>
<tr>
<td>4</td>
<td>10.4526</td>
<td>3.6358</td>
<td>10</td>
<td>0.2416</td>
<td>10.5518</td>
<td>6.6308</td>
<td>10</td>
<td>0.543</td>
</tr>
<tr>
<td>5</td>
<td>9.6504</td>
<td>4.2799</td>
<td>20</td>
<td>0.2395</td>
<td>9.4302</td>
<td>6.9391</td>
<td>20</td>
<td>0.458</td>
</tr>
<tr>
<td>6</td>
<td>8.8753</td>
<td>4.6682</td>
<td>30</td>
<td>0.244</td>
<td>8.353</td>
<td>7.0943</td>
<td>30</td>
<td>0.3632</td>
</tr>
<tr>
<td>7</td>
<td>8.1631</td>
<td>4.842</td>
<td>40</td>
<td>0.2472</td>
<td>7.3604</td>
<td>7.125</td>
<td>40</td>
<td>0.3908</td>
</tr>
<tr>
<td>8</td>
<td>7.5531</td>
<td>4.875</td>
<td>50</td>
<td>0.2502</td>
<td>6.455</td>
<td>7.125</td>
<td>50</td>
<td>0.4121</td>
</tr>
<tr>
<td>9</td>
<td>7.1773</td>
<td>4.875</td>
<td>57.1889</td>
<td>0.2534</td>
<td>5.8621</td>
<td>7.1124</td>
<td>57.1889</td>
<td>0.433</td>
</tr>
<tr>
<td>10</td>
<td>7.5531</td>
<td>4.875</td>
<td>57.1889</td>
<td>0.2534</td>
<td>5.059</td>
<td>7.0264</td>
<td>67.9884</td>
<td>0.3807</td>
</tr>
</tbody>
</table>

**Diffuser C**

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>12.507</td>
<td>1.9118</td>
<td>-9.2436</td>
<td>0.125</td>
<td>14.035</td>
<td>4.909</td>
<td>0</td>
<td>0.1875</td>
</tr>
<tr>
<td>2</td>
<td>11.9707</td>
<td>2.3619</td>
<td>-5</td>
<td>0.1367</td>
<td>12.5411</td>
<td>5.6706</td>
<td>1</td>
<td>0.5</td>
</tr>
<tr>
<td>3</td>
<td>11.3719</td>
<td>2.8644</td>
<td>0</td>
<td>0.1525</td>
<td>11.2989</td>
<td>6.3345</td>
<td>5</td>
<td>0.55</td>
</tr>
<tr>
<td>4</td>
<td>10.8591</td>
<td>3.2974</td>
<td>5</td>
<td>0.1853</td>
<td>10.5518</td>
<td>6.6308</td>
<td>10</td>
<td>0.543</td>
</tr>
<tr>
<td>5</td>
<td>10.4273</td>
<td>3.657</td>
<td>10</td>
<td>0.2219</td>
<td>9.4302</td>
<td>6.9391</td>
<td>20</td>
<td>0.458</td>
</tr>
<tr>
<td>6</td>
<td>9.6503</td>
<td>4.284</td>
<td>20</td>
<td>0.2395</td>
<td>8.353</td>
<td>7.0943</td>
<td>30</td>
<td>0.3632</td>
</tr>
<tr>
<td>7</td>
<td>8.8753</td>
<td>4.6682</td>
<td>30</td>
<td>0.244</td>
<td>7.3604</td>
<td>7.125</td>
<td>40</td>
<td>0.3908</td>
</tr>
<tr>
<td>8</td>
<td>8.1631</td>
<td>4.842</td>
<td>40</td>
<td>0.2472</td>
<td>6.455</td>
<td>7.125</td>
<td>50</td>
<td>0.4121</td>
</tr>
<tr>
<td>9</td>
<td>7.5531</td>
<td>4.875</td>
<td>50</td>
<td>0.2503</td>
<td>5.8621</td>
<td>7.1124</td>
<td>57.1889</td>
<td>0.433</td>
</tr>
<tr>
<td>10</td>
<td>7.1773</td>
<td>4.875</td>
<td>57.1889</td>
<td>0.2534</td>
<td>5.059</td>
<td>7.0264</td>
<td>67.9884</td>
<td>0.3807</td>
</tr>
</tbody>
</table>
Figure 3.1 Schematic of Diffuser Location Inside a Mixed Flow Pump
**Diffuser B:** This was an improved design developed purely by experience, no CFD analysis was done on Diffuser B prior to this thesis work. As shown in Figure 3.5, from outlet to inlet: the twisting angle of the PS and SS walls on the hub side ranged from 0 to 57.19 degrees, on the shroud side increased from 0 to 67.98 degrees; the hub radius varied from 1.912 in to 4.875 in, while the shroud radius varied from 4.909 in to 7.026 in.

**Diffuser C:** Diffuser C is an optimized design combining aspects of Diffuser A and B. As shown in Figure 3.6, Diffuser C has an enlarged, $108^0$ vs. $90^0$, intersecting angle between SS and outlet in the hub. As shown in Figure 3.5, from outlet to inlet: the twisting angle of the PS and SS walls on the hub side was increased from -9.24 to 57.19 degrees, on the shroud side ranged from 0 to 67.98 degrees; the hub radius varied from 1.912 in to 4.875 in, and the shroud radius varied from 4.909 in to 7.026 in.

**3.2.2 Grid Generation:**

The grid density is the most important criteria for providing a realistic and converging solution. In order to obtain the optimal grid density, many parameters of the region had to be taken into consideration. This was accomplished through trial-and-error. A Body-Fitted Coordinate Based grid with a curve segmentation size of $I = 20$, $J = 20$ and $K = 100$ (total 44541 cells) was chosen for the model. As shown in Figure 3.2, $I$ is taken along the pitchwise (PS to SS) direction, $J$ is taken along spanwise (hub to shroud) direction, and $K$ is taken along streamwise (inlet to outlet) direction. (The grid selection is the same for Diffuser B and Diffuser C as shown in Figure 3.3 and Figure 3.4.) The $20 \times 20 \times 100$ grid provided adequate resolution and the value of $y^+$ was approximately 40 at grid points adjacent to the PS and SS walls. The value of $y^+$ was approximately 45 at grid
points adjacent to the shroud and hub surfaces which ensured the grid line to lie in the log-law layer of the turbulent boundary layer as discussed in paragraph 3.1.3.2.1 - Grid Spacing near Walls. The major steps involved in generating the grid are as follows:

1. Specify the node distribution along the boundaries (map the boundaries).
2. Adjust the mapped nodes along the boundaries.
3. Create the grid inside the domain via interpolation.
4. Smooth the interpolated grid.
5. Display and verify the grid.

It was determined that the grid needed to be finer in regions where properties of interest were changing rapidly. It was also important to note that care must be exercised in the design of the computational mesh to ensure a smooth spatial distribution of nodal points throughout the entire flow domain. If the transition in grid density was abrupt, especially along the direction of flow, spurious spatial oscillation in the flow variables may be observed causing a divergent numerical solution.

Since the majority of losses, flow separation and vortices occur near the walls, dense cell layers should be set close to walls to catch those phenomena. In other words, to effectively capture viscous effects, grids are clustered near wall surfaces.

This can be done by redistributing nodes according to specified weighting factors. Dividing the distance between nodes in the neighborhood of the selected adjustment points by the weighting factor will redistribute the nodes according to a smooth hyperbolic tangent function. The following grid weighting factors are found suitable for the diffuser model:
Figure 3.2 Surface Grid of Diffuser A
Figure 3.3 Surface Grid of Diffuser B
Figure 3.4 Surface Grid of Diffuser C
Figure 3.5 Twisting (Chord) Angle Comparison
Figure 3.6 Diffuser Intersection Angle (Between Outlet & SS on Hub) Illustration
<table>
<thead>
<tr>
<th>Hub (J1)</th>
<th>PS (I1)</th>
<th>Shroud (J20)</th>
<th>SS (I20)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet:</td>
<td>10-10</td>
<td>10-10</td>
<td>10-10</td>
</tr>
<tr>
<td>Outlet:</td>
<td>10-10</td>
<td>10-10</td>
<td>10-10</td>
</tr>
</tbody>
</table>

A grid weighting factor of 3 - 3 was also applied along the channel from inlet to outlet to account for any flow pattern change in the inlet and outlet region. Six-point interpolation method was used for grid generation. A grid verification is performed after the grid generation, which includes cell type check, cell volume check, relative cell size check and skewness check. The models passed all the standard check requirements which are the program defaults: 20 to 1 ratio of maximum volume ratio between adjacent cells, and 60 degrees maximum deviation from orthogonal.

******* This area intentionally left blank. *******
3.2.3 Case File Generation (Boundary Conditions and Assumptions):

In this phase of the modeling, constraints and variables such as geometric unit conversion, fluid properties, and boundary conditions were applied to the model. For this problem the SI system was the unit base chosen for analysis. Water was used as the fluid medium with density $= 1000 \text{ kg/m}^3$, and absolute viscosity $= 9.8 \times 10^{-3} \text{ kg.s/m}$ at $20^\circ C$. The module chosen in FLUENT only required the inlet boundary condition to fully define the model. The no slip condition on the wall surfaces was also input into the program.

In order to predict flow patterns for the inlet boundary conditions at the design flow rate, normal velocities toward the inlet plane are adopted since the incidence angle effects were unknown. The normal velocity was calculated based on 4500 gpm flow rate:

$$V_n = \frac{Q}{A} = \frac{4500 \text{gpm} \times 3.7854 \times 10^{-3} \text{m}^3 / \text{gallon}}{8 \times 4.12 \times 10^{-3} \text{m}^2} = 516.81 \text{m/min} = 8.61 \text{m/sec}$$

For each channel the inlet surface area was $4.12 \times 10^{-3} \text{ m}^2$:

$$\text{Re} = \frac{\rho V D}{\mu} = \frac{1000 \text{ kg/m}^3 \times 8.61 \text{m/sec} \times 0.064 \text{m}}{9.8 \times 10^{-3} \text{ kg.s/m}} = 56229$$

Given this high Reynolds number, it is clear that turbulent flow is present within the diffuser. The $k-\epsilon$ turbulent flow model was used, which requires inlet turbulence intensity and characteristic length input. A turbulence intensity of 10% (based on the design flow of 4000 to 5000 gpm), characteristic length of 0.032 m were used as inlet condition. Due to the adiabatic nature of the diffuser, isothermal flow conditions were assumed.
3.2.4 *Computing The Results:*

For each model, approximately 1000 iterations were performed to reach the FLUENT default convergence criteria. The programmed criteria proved adequate for this study and did not require alteration. Data files were created during the program analysis which contained the solution for the CFD problem. Results were presented in both graphical (velocity vector plot, pressure distribution contour plot) and numerical format.

******** This area intentionally left blank. ********
Chapter 4 Results & Discussion

4.1 General CFD Results

4.1.1 Alphanumeric Illustration

The goal of a diffuser is to convert the inlet dynamic pressure of the fluid to a static pressure rise. It is desirable to recover as much of the entering dynamic pressure as possible. It is also important that the exiting flow be steady and has a uniform profile for the next impeller stage.

Table 4.1 lists the general results from the FLUENT program for Diffusers A, B and C.

<table>
<thead>
<tr>
<th>Diffuser</th>
<th>AR</th>
<th>Static Pressure Recovery $\Delta P$ (Pa)</th>
<th>$C_p$</th>
<th>$C_{p,\text{ideal}}$</th>
<th>Efficiency $\eta$</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>1.303</td>
<td>7745</td>
<td>0.209</td>
<td>0.41</td>
<td>50.90%</td>
</tr>
<tr>
<td>B</td>
<td>1.406</td>
<td>13415</td>
<td>0.303</td>
<td>0.49</td>
<td>61.33%</td>
</tr>
<tr>
<td>C</td>
<td>1.426</td>
<td>14202</td>
<td>0.330</td>
<td>0.51</td>
<td>65.03%</td>
</tr>
</tbody>
</table>

Compared with Diffuser A, Diffuser B has approximately an 8% area ratio increase. The static pressure recovery coefficient, $C_p$, which is the measure of the diffuser performance, has improved from 0.209 to 0.303; diffuser efficiency $\eta$ increases from 50.9% to 61.33%. Compared with Diffuser B, Diffuser C has roughly a 1.5% area ratio increase; $C_p$ increases from 0.303 to 0.33; the efficiency $\eta$ increases from 61.33% to 65.03%. The improvements shown here are very significant for the pump industry, which strives for every percentage improvement in diffuser performance.
In order to validate the reliability of the CFD results, a correlation was performed to test data as stated below.

Both Diffuser A and B were tested in a mixed flow pump at design flow rate.

Table 4.2 Test Data for Diffuser A & B

<table>
<thead>
<tr>
<th></th>
<th>Pump with Diffuser A</th>
<th>Pump with Diffuser B</th>
</tr>
</thead>
<tbody>
<tr>
<td>Test RPM</td>
<td>1770</td>
<td>1770</td>
</tr>
<tr>
<td>GPM</td>
<td>4500</td>
<td>4500</td>
</tr>
<tr>
<td>Static Pressure Recovery (kPa)</td>
<td>405</td>
<td>411</td>
</tr>
<tr>
<td>Pump Efficiency</td>
<td>79.2%</td>
<td>81.1%</td>
</tr>
</tbody>
</table>

The test data cannot be used to make direct comparison with the CFD results of each diffuser since the tests were done by using a pump which has other components such as motor and impeller besides diffuser. However, relative comparison can still be made by following: the test data (Table 4.2) indicated that at the design flow condition, the static pressure recovery improvement from Diffuser A to Diffuser B is approximately 6000 Pa (411kPa 405kPa). From CFD results (Table 4.1): the static pressure recovery improvement from Diffuser A to Diffuser B is 5670 Pa.

Table 4.3 Comparison of CFD Results & Test Data

<table>
<thead>
<tr>
<th>Static Pressure Recovery Improvement from Diffuser A to B ΔP (Pa)</th>
<th>CFD</th>
<th>Test</th>
</tr>
</thead>
<tbody>
<tr>
<td>5670</td>
<td>5670</td>
<td></td>
</tr>
<tr>
<td>Static Pressure Recovery Improvement from Diffuser B to C ΔP (Pa)</td>
<td>787</td>
<td>N/A</td>
</tr>
</tbody>
</table>

As shown in Table 4.3, the CFD results had excellent correlation with the test data.
4.1.1 Graphical Illustration

Based on the complex nature of the diffuser geometry, it is important to examine the relationship between pressure recovery and area progression of the diffuser.

Figure 4.1 shows the area progression for the three diffusers in a graphical form: all three diffusers have positive area progression up to 3/4 of the total channel length reference to K direction grid increment. Figure 4.2 shows the static pressure recovery for the three diffusers. Corresponding to the area progression chart, all three diffusers have positive pressure recovery slope up to about 3/4 of the whole length of the diffuser. Both Diffuser B and Diffuser C have steeper static pressure recovery slope compared with Diffuser A. Among the three curves, although Diffuser B has the highest static pressure magnitude, Diffuser C has the highest static pressure recovery value (14202 Pa) by comparing the difference between the end point and start point for each curve (Diffuser B: 13415 Pa, Diffuser A: 7745 Pa). Figure 4.3 shows the total pressure loss (energy loss) for the three diffusers. Diffuser C shows a smoother energy loss slope compared with Diffuser B and A, it also has the lowest total pressure loss value (4499 Pa) by comparing the difference between the end point and start point for each curve (Diffuser B: 4846 Pa, Diffuser A: 6333 Pa). So Diffuser C has the highest static pressure recovery and lowest energy loss among the three diffusers.
Figure 4.1 Area Progression of Diffuser A, B & C
Figure 4.2 Static Pressure Recovery of Diffuser A, B & C
Figure 4.3 Total Pressure Loss of Diffuser A, B & C
4.2 Flow Pattern Study

In order to obtain a clear view of flow pattern in three dimensional domain, certain slices along J (spanwise: hub to shroud), I (pitchwise: pressure side to suction side) and K (streamwise: inlet to outlet) near the hub, shroud, suction side (SS), pressure side (PS), inlet, outlet and mid-channels were selected for study.

In the following case: slices I=2 (the layer approximately 0.05 mm away from SS; dimensionless normal distance \( y^* = 40 \)); I=19 (the layer approximately 0.05 mm away from PS; dimensionless normal distance \( y^* = 40 \)); J=2 (the layer approximately 0.05 mm away from the hub, dimensionless normal distance \( y^* = 45 \)); J=19 (the layer approximately 0.05 mm away from the shroud, dimensionless normal distance \( y^* = 45 \)) were chosen to demonstrate the flow patterns. Since their \( y^* \) values are greater than 25 and less than 300, these slices are located in the log-law layer of the turbulent boundary layer. K=1, K=30, K=50, K=70 and K=100 (the slices of inlet, outlet and along the channel), I=11 (the center slice between PS and SS: also is referred to as mid-pitchwise layer), J=11 (the center slice between hub and shroud which intersects PS and SS, also referred to as a mid spanwise layer) were selected for each model as well.

4.2.1 Diffuser A

4.2.1.1 Spanwise Flow Pattern

For slice J=2, adjacent layer to hub wall, Figure 4.4A shows that velocity magnitudes start to decrease dramatically at about 1/3 of the way along the channel; velocity vector directions start to turn at about half way along the channel. This is a sign
of secondary flows formation which can be explained by the reduction in hub width along the channel and the large curvature of PS. Under these conditions it is very difficult to avoid the formation of secondary flows. Figure 4.4B is an enlarged view of Figure 4.4A at the outlet region. The exit flow angle is quite large, $\approx 55^\circ$, this angle is measured between the centerline velocity vector and its normal component. The magnitude of this value indicates that the PS and SS walls are incapable of realigning the flow to the axial direction. This phenomenon is a clear sign that energy has been divided by large tangential velocity components which significantly reduces the potential of static pressure recovery. Figure 4.4C indicates that static pressure is not fully recovered in the center area of outlet region, and that the static pressure distribution is not homogeneous near the inlet and suction side.

For slice $J=19$, adjacent layer to shroud, Figure 4.5A indicates that near the corner of shroud and SS, velocity magnitude increases to nearly equal the inlet velocity magnitude. This indicates that dynamic pressure increases significantly in that particular area. Figure 4.5B shows that near the SS half of the channel, static pressure is not recovered at all, on the contrary, static pressure decreases near the outlet.

For slice $J=11$, the mid spanwise slice between the hub and shroud, Figure 4.6A shows the velocity magnitude is not reduced along PS. Near the later portion of the SS, the flow is turning towards SS, this is an indication of secondary flows formation. In Figure 4.6B, it can be seen that the static pressure recovery near PS is poor.
Figure 4.4A Diffuser A: Velocity Vector Plot of Slice J=2 (Near Hub)
Figure 4.4B Diffuser A: Zoom View of Velocity Vector at outlet of Slice J=2 (Near Hub)
Figure 4.4C Diffuser A: Static Pressure Distribution Filled Contour of Slice J=2 (Near Hub)
Figure 4.5A Diffuser A: Velocity Vector Plot of Slice J=19 (Near Shroud)
Figure 4.5B  Diffuser A: Static Pressure Distribution Filled Contour of Slice J=19 (Near Shroud)
Figure 4.6A Diffuser A: Velocity Vector Plot of Slice J=11 (Mid-spanwise)
Figure 4.6B Diffuser A: Static Pressure Distribution Filled Contour of Slice J=11 (Mid-spanwise)
4.2.1.2 Pitchwise Flow Pattern

For slice I=2, the adjacent layer to SS, Figure 4.7A shows that near the outlet and hub region, flow starts to rotate away from hub which features the shortest curvature, this is an indication of secondary flows formation. The radial component of the velocity which directs from hub to shroud grows continuously when the flow velocity decreases near the outlet. The radial component of the velocity has highest value near the mid-layer. The flow along the shroud reduces speed in the middle of the channel and recovers to almost full speed at the outlet. Figure 4.7B clearly shows that the static pressure has not recovered at all and there is in fact a pressure void near the shroud outlet area.

For slice I=19, the adjacent layer to PS, Figure 4.8A shows that as the flow starts to turn towards hub the radial velocity component continuously increases when flow rate reduces and the velocity magnitude decreases near the hub region. Figure 4.8B shows static pressure is well recovered near the outlet/shroud, and the static pressure distribution is fairly homogeneous.

For slice I=11, the mid pitchwise layer between SS and PS, Figure 4.9A indicates that flow speeds up near the shroud outlet region, and flow near the hub, turns towards the hub itself starting approximately 3/4 of the way along the channel. Figure 4.9B shows that near the hub side of the diffuser the static pressure recovery does not form in a desirable manner. The static pressure recovery near the outlet is also poor.

4.2.1.3 Streamwise Flow Pattern
Figure 4.7A Diffuser A: Velocity Vector Plot of Slice I=2 (Near SS Blade)
Figure 4.7B Diffuser A: Static Pressure Distribution Filled Contour of Slice I=2 (Near SS Blade)
Figure 4.8A Diffuser A: Velocity Vector Plot of Slice I=19 (Near PS Blade)
Figure 4.8B  Diffuser A: Static Pressure Distribution Filled Contour of Slice I=19 (Near SS Blade)
Figure 4.9A Diffuser A: Velocity Vector Plot of Slice I=11 (Mid-pitchwise)
Figure 4.9B  Diffuser A: Static Pressure Distribution Filled Contour of Slice $l=11$ (Mid-pitchwise)
For streamwise slices $K=1$, $K=30$, $K=50$, $K=70$ and $K=100$, Figure 4.10A shows that the static pressure recovery near SS and hub is poor along the channel.

At outlet $K=100$, Figure 4.11A shows a significant clockwise secondary flow pattern near PS/hub/SS which can be contributed to both shroud to hub curvature and PS to SS curvature as illustrated in Figure 2.3 and 2.4. There is also an insignificant counterclockwise (directs from PS to SS) flow pattern along shroud can be observed, this is clearly caused by the PS to SS curvature as shown in Figure 2.4, however its magnitude was much smaller than the clockwise vortex. Figure 4.11B is shown to present the side view of Figure 4.11A in the interest of clarity. Figure 4.11C shows that the pressure recovery near the SS half of the channel is poor, the worst occurs at the corner of SS/shroud.
Figure 4.10A Diffuser A: Static Pressure Distribution Filled Contour of Slice K=1, 30, 50, 70, 100
Figure 4.11A  Diffuser A: Velocity Vector Plot of Slice K=100 (Outlet)
Figure 4.11B  Diffuser A: Velocity Vector Plot of Slice K=100 Side View (Outlet)
Figure 4.11C  Diffuser A: Static Pressure Distribution Filled Contour of Slice K=100 (Outlet)
4.2.2 **Diffuser B**

### 4.2.2.1 Spanwise Flow Pattern

For slice \( J = 2 \), adjacent layer to hub wall, compared with Diffuser A (Figure 4.4A), Figure 4.12A shows Diffuser B has the same flow pattern with the exception that the velocity magnitudes in the last half of the channel are smaller; it also shows that the hub of Diffuser B is wider than Diffuser A in the inlet region. These observations contribute to Diffuser B’s better performance than Diffuser A. Figure 4.12B is a zoom view of Figure 4.12A near the outlet. The exit flow angle is measured same as Diffuser A which is 55°. this tells that there is room for improvement. Figure 4.12C also shows the same pattern of Diffuser A (Figure 4.4C) with the exception that the overall static pressure recovery is 5654 Pa higher.

For slice \( J = 19 \), adjacent layer to shroud, Figure 4.13A indicates that in the outlet region velocity magnitudes distribution is fairly homogeneous, which is a significant improvement compared with Diffuser A (Figure 4.5A). Figure 4.13B shows that the static pressure recovery near the outlet/SS corner has been improved compared with Diffuser A (Figure 4.5B).

For slice \( J = 11 \), the mid spanwise slice between hub and shroud, Figure 4.14A shows the velocity magnitude has been reduced along PS in the outlet region, which is also a big improvement compared with Diffuser A (Figure 4.6A). Figure 4.14B shows that the pressure recovery near PS is still poor, but has also been improved compared with Diffuser A (Figure 4.6B).
Figure 4.12A Diffuser B: Velocity Vector Plot of Slice J=2 (Near Hub)
DIFFUSER-B

Velocity Vectors (Meters/Sec)

Lmax = 9.837E+00  Lmin = 0.000E+00

Figure 4.12B Diffuser B. Zoom View of Velocity Vector Plot at Outlet of Slice J=2 (Near Hub)
Figure 4.12C Diffuser B: Static Pressure Distribution Filled Contour of Slice J=2 (Near Hub)
Figure 4.13A Diffuser B: Velocity Vector Plot of Slice J=19 (Near Shroud)
Figure 4.13B  Diffuser B: Static Pressure Distribution Filled Contour of Slice J=19 (Near Shroud)
Figure 4.14A Diffuser B: Velocity Vector Plot of Slice J=11 (Mid-Spanwise)
Figure 4.14B  Diffuser B: Static Pressure Distribution Filled Contour of Slice J=11 (Mid-Spanwise)
4.2.2.2 Pitchwise Flow Pattern

For slice $I=2$, adjacent layer to SS wall, Figure 4.15A shows the velocity magnitudes in the outlet region are smaller and distribute in a homogeneous manner as compared with Diffuser A (Figure 4.7A). Figure 4.15B shows that the static pressure has recovered in a more desirable way near the hub and shroud of the outlet region compared with Diffuser A (Figure 4.7B).

For slice $I=19$, adjacent layer to PS, Figure 4.16A shows that the velocity magnitudes are smaller along the channel compared with Diffuser A (Figure 4.8A). Figure 4.16B shows a more uniform pressure recovery distribution near shroud compared with Diffuser A (Figure 4.8B).

For slice $I=11$, mid pitchwise layer between SS and PS, Figure 4.17A shows that velocity magnitudes near shroud in the outlet region are smaller than Diffuser A (Figure 4.9A). Figure 4.17B shows that near hub side the static pressure recovery is worse than Diffuser A (Figure 4.9B).

4.2.2.3 Streamwise Flow Pattern

For streamwise slices $K=1$, $K=30$, $K=50$, $K=70$ and $K=100$, Figure 4.18A shows that the static pressure recovery near the suction side and hub is poor along the channel, and there is no noticeable improvement compared with Diffuser A (Figure 4.10A).

At outlet $K=100$, Figure 4.19A shows a similar secondary flows pattern as of Diffuser A (Figure 4.11A) with the exception that the counterclockwise secondary flow near the shroud is more obvious, which is an indication that PS to SS curvature caused
secondary flows are more severe than in Diffuser A. Figure 4.19B is the side view of Figure 4.19A. Figure 4.19C shows that the static pressure distribution is very poor along SS, and there is a noticeable pressure void near SS.

********* This area intentionally left blank. *********
Figure 4.15A Diffuser B: Velocity Vector Plot of Slice I=2 (Near SS Blade)
Figure 4.15B  Diffuser B: Static Pressure Distribution Filled Contour of Slice 1=2 (Near SS Blade)
Figure 4.16A Diffuser B: Velocity Vector Plot of Slice I=19 (Near PS Blade)
**DIFFUSER-B**

**Static Pressure (Pascals)**

$L_{\text{max}} = 6.586 \times 10^4$

$L_{\text{min}} = -3.825 \times 10^3$

---

**Figure 4.16B** Diffuser B: Static Pressure Distribution Filled Contour of Slice 1=19 (Near PS Blade)
Figure 4.17A  Diffuser B: Velocity Vector Plot of Slice I=11 (Mid-Pitchwise)
Figure 4.17B Diffuser B: Static Pressure Distribution Filled Contour of Slice I=11 (Mid-Pitchwise)
Figure 4.18A  Diffuser B: Static Pressure Distribution Filled Contour of Slice K=1, 30, 50, 70, 100
Figure 4.19A Diffuser B: Velocity Vector Plot of Slice K=100 (Outlet)
Figure 4.19B  Diffuser B: Velocity Vector Plot of Slice K=100 Side View (Outlet)
Figure 4.19C  Diffuser B: Static Pressure Distribution Filled Contour of Slice K=100 (Outlet)
4.2.3 Diffuser C

4.2.3.1 Spanwise Flow Pattern

For slice J=2, adjacent layer to hub wall, Compared with Diffuser A (Figure 4.4A), Figure 4.20A shows Diffuser C has the similar flow pattern as Diffuser B (Figure 4.12A) with the exception that it has more severe flow vortex near the center of SS. Figure 4.20B is a zoom view of Figure 4.20A at the outlet region; the exit flow angle is measured only as 44° which is a significant improvement compared with Diffuser A (Figure 4.4B) and B (Figure 4.12B). Figure 4.20A and 4.20B clearly demonstrate the negative and positive effects caused by enlarging the intersecting angle between suction side blade and outlet in the hub. The traditional design approach is maintain the angle to 90°, while in this diffuser the angle was enlarged to 108° in the hope to reduce the large flow exit angle which is commonly observed in the 90° design. Figure 4.20A and Figure 4.20B indicate that this method works: the flow was straightened near the outlet region due to the smaller curvature on SS. The side effect of this method can also be easily observed in Figure 4.20A: the rotational flow from the pressure side hits the relative straight SS, a flow vortex easily forms, this nevertheless reduce the diffuser efficiency to a certain degree. According to the numerical data listed in Table 4.1, the positive effect is superior to that of the side effect which indicates that this is indeed a useful approach to improve the diffuser performance. As a proof, Figure 4.20C shows that the static pressure recovery in the outlet region has improved greatly compared with Diffuser A (Figure 4.4C) and B (Figure 4.12C).
For slice J=19, adjacent layer to shroud, and slice J=11, the mid pitchwise slice between hub and shroud, the flow patterns were found identical to that of Diffuser B.

4.2.3.2 Pitchwise Flow Pattern

For slice I=2, adjacent layer to SS; slice I=19, adjacent layer to PS; slice I=11, mid pitchwise layer between SS and PS, flow patterns were found identical to that of Diffuser B.

********* This area intentionally left blank. *********
Figure 4.20A  Diffuser C: Velocity Vector Plot of Slice J=2 (Near Hub)
Figure 4.20B  Diffuser C: Zoom View of Velocity Vector Plot at Outlet of Slice J=2 (Near Hub)
Figure 4.20C Diffuser C: Static Pressure Distribution Filled Contour of Slice J=2 (Near Hub)
4.2.3.3 Streamwise Flow Pattern

For streamwise slices $K=1$, $K=30$, $K=50$, $K=70$ and $K=100$, Figure 4.21A shows that the static pressure recovery near the suction side and hub is well improved compared with Diffuser A (Figure 4.10A) and B (Figure 4.18A).

At outlet $K=100$, Figure 4.22A shows a similar secondary flows pattern as of Diffuser B (Figure 4.19A) with the exception that the velocity magnitude of the secondary flows near hub has reduced from a maximum value of approximately 8.3 m/s to a maximum of approximately 5.3 m/s. Figure 4.22B is the side view of Figure 4.22A. Figure 4.22C shows that the static pressure recovery near hub has been improved compared with Diffuser A (Figure 4.11C) and B (Figure 4.19C), however the recovery near SS is still poor.

********** This area intentionally left blank. **********
Figure 4.21A  Diffuser C: Static Pressure Distribution Filled Contour of K=1, 30, 50, 70, 100
Figure 4.22A  Diffuser C: Velocity Vector Plot of Slice K=100 (Outlet)
Figure 4.22B Diffuser C: Velocity Vector Plot of Slice K=100 Side View (Outlet)
Figure 4.22C  Diffuser C: Static Pressure Distribution Filled Contour of Slice K=100 (Outlet)
4.2.4 Flow Pattern Comparison of Diffusers A, B & C

As shown in Figure 4.23, for a diffuser to reach its maximum static pressure recovery capacity, all the dynamic loss has to be transformed into static pressure recovery, in another words, inside the passage, the flow has to align perfectly to the channel, and the exit flow has to leave the channel perfectly normal to the outlet plane.

From the individual flow pattern study made above, it is found that the slice J=2 (near the hub) have the most representation of the difference among the three diffusers, so the velocity vector plots for the three diffusers were all placed in Figure 4.23A and static pressure distribution for the three diffusers were all placed in Figure 4.24B for easy comparison.

Figure 4.23A shows that inside Diffuser C, the velocity magnitude decreases right after the flow enters the passage, in another words. Diffuser C has the most efficient diffusion process among the three diffusers. Diffuser A has the worst diffusion process. It is also observed that all three diffusers have energy loss into tangential velocity components in the outlet region. Diffuser C has the least exit flow angle.

Figure 4.23B shows that near the inlet region, Diffuser A has a significant pressure void area; the void area has been reduced inside Diffuser B; Diffuser C has no pressure void area at all. In the outlet area, the pressure is not well recovered in the center area inside Diffuser A and B; on the contrary, the Diffuser C presents a very uniform pressure distribution in the outlet region. Figure 4.23B is a strong evidence showing that Diffuser C is a superior design compared with Diffuser A and B.
Figure 4.23 Ideal Diffuser Flow Pattern
Figure 4.23B Static Pressure Distribution Comparison of Diffusers A, B, & C of Slice J = 2 (Near Hub)
Figure 4.23A Velocity Vector Plot Comparison of Diffusers A, B, & C of Slice J = 2 (Near Hub)
Chapter 5  Conclusions and Recommendations

5.1 Conclusions:

Based on the results presented in Chapter 4, the following conclusions are made:

1. Compared with Diffuser A, Diffuser B has approximately an 8% area ratio increase. The static pressure recovery coefficient, $C_P$, which is the measure of the diffuser performance, has improved from 0.209 to 0.303; diffuser efficiency $\eta$ increases from 50.9% to 61.33%.

2. Compared with Diffuser B, Diffuser C has roughly a 1.5% area ratio increase; $C_P$ increases from 0.303 to 0.33; the efficiency $\eta$ increases from 61.33% to 65.03%.

3. The CFD results have good correlation with test data: at the design flow condition, the static pressure recovery improvement from Diffuser A to B is 5670 Pa from CFD results and 6000 Pa from test data.

4. The major finding from the flow pattern study of Diffuser A and Diffuser B indicates that the flow exit angle (angle between centerline exiting flow and its normal component) near the hub side of the diffuser affects the diffuser performance significantly. A small exit flow angle reduces secondary flows formation which improves static pressure recovery.

5. The flow patterns study based on Figure 4.4A though Figure 4.22C in chapter 4 also confirms that the performance of Diffuser C is superior than Diffuser A and
B. It can be concluded that the Design of Diffuser C is an optimized design of Diffuser A and B.

6. Through this project the FLUENT CFD package was found to be very suitable for curved geometry designs and diffuser performance evaluation. Especially, it offers the capability to predict the model’s performance based on different geometric configurations and inlet boundary conditions. It gives the user a glimpse into the actual flow processes occurring inside the diffuser.

Overall, this study suggests a very practical method for prediction of mixed-flow pump diffuser performance at a given design flow condition by studying the theoretical flow patterns that takes place inside the diffuser flow passages.

5.2 Recommendations

Several studies can be recommended as following:

- A full diffuser model consisting of eight channels with simulated inlet and outlet region could be modeled to observe more realistic flow simulation inside the pump.

- A corresponding test which measures the flow parameters for an individual diffuser stage could be done to provide inlet flow conditions for refining the CFD models.

- A study focusing on off-design flow conditions.
Reference


