

151801

ÇUKUROVA UNIVERSITY
INSTITUTE OF NATURAL AND APPLIED SCIENCES

PhD THESIS

Adnan ÖZTÜRK

**A STUDY OF IMPELLER-DIFFUSER INTERACTION
IN A CENTRIFUGAL PUMP**

DEPARTMENT OF MECHANICAL ENGINEERING

151 801

ADANA, 2004

**ÇUKUROVA ÜNİVERSİTESİ
FEN BİLİMLERİ ENSTİTÜSÜ**

**A STUDY OF IMPELLER-DIFFUSER INTERACTION IN
A CENTRIFUGAL PUMP**

**Adnan ÖZTÜRK
DOKTORA TEZİ**

MAKİNA MÜHENDİSLİĞİ ANABİLİM DALI

Bu tez 13/12/2004 Tarihinde Aşağıdaki Jüri Üyeleri Tarafından Oybirliği/Oyçokluğu İle Kabul Edilmiştir.

İmza:.....
Prof.Dr. Kadir AYDIN
DANIŞMAN

İmza:.....
Prof.Dr. Beşir ŞAHİN
ÜYE

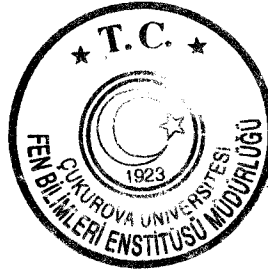
İmza:.....
Doç.Dr. Osman BABAARSLAN
ÜYE

İmza:.....
Doç.Dr. Ali PINARBAŞI
ÜYE

İmza:.....
Yrd.Doç.Dr. Hüseyin AKILLI
ÜYE

Bu tez Enstitümüz Makina Mühendisliği Anabilim Dalında hazırlanmıştır.

Kod No: 869



İmza:.....
Prof. Dr. Fikri AKDENİZ
Enstitü Müdürü
İmza Mühür

Bu Çalışma Çukurova Üniversitesi Bilimsel Araştırma Projeleri Birimi Tarafından Desteklenmiştir.

Proje No: FBE.2001D111

Not: Bu tezde kullanılan özgün ve başka kaynaktan yapılan bildirişlerin, çizelge, şekil ve fotoğrafların kaynak gösterilmeden kullanımı, 5846 sayılı Fikir ve Sanat Eserleri Kanunundaki hükümlere tabidir.

ABSTRACT

PhD THESIS

A STUDY OF IMPELLER-DIFFUSER INTERACTION IN A CENTRIFUGAL PUMP

Adnan ÖZTÜRK

**DEPARTMENT OF MECHANICAL ENGINEERING
INSTITUTE OF NATURAL AND APPLIED SCIENCES
UNIVERSITY OF ÇUKUROVA**

Supervisor Prof.Dr. Kadir AYDIN

Year : 2004, Pages : 215

Jury Prof.Dr. Kadir AYDIN

Prof.Dr. Beşir ŞAHİN

Assoc.Prof.Dr. Osman BABAARSLAN

Assoc.Prof.Dr. Ali PINARBAŞI

Assist.Prof.Dr. Hüseyin AKILLI

In this study, a centrifugal pump with a diffuser which has different radial gaps (20% of impeller radius) and a volute a thickness of 8 mm which is distinct from traditional pumps was investigated numerically with FLUENT package program using Finite Volume Method. Flow was assumed to be three dimensional, viscous, incompressible, turbulent steady and unsteady. The present centrifugal pump has an impeller with five backward curved blades and nine diffuser vanes. The impeller runs at 890 rpm. The centrifugal pump was modeled with various solid modeling programs. The investigations for three-dimensional geometry have been carried out for 15 different volume flow rates, three different radial gaps and both steady and unsteady flows. The performance curves have been obtained for each gap to observe the effect of different radial gaps on the pump performance. The results were compared with the published experimental results obtained with PIV technique for five different flow rates for a 20% radial gap. On top of that, for the cases of 15% and 10% radial gaps, the flow mechanisms through the impeller and diffuser were numerically investigated. From the present results, it can be concluded that the 20% radial gap is most suitable for the pump and the flow in the pump is affected adversely by decreasing radial gaps and causing pressure fluctuations in the pump.

Keywords: Centrifugal pump, Computational Fluid Dynamics (CFD), Finite Volume Method, Impeller-Diffuser interaction, Radial gap effect

ÖZ

DOKTORA TEZİ

**SANTRİFÜJ POMPADA ÇARK-DİFÜZÖR ETKİLEŞİMİ
ÜZERİNE BİR ÇALIŞMA**

Adnan ÖZTÜRK

**ÇUKUROVA ÜNİVERSİTESİ
FEN BİLİMLERİ ENSTİTÜSÜ
MAKİNA MÜHENDİSLİĞİ ANABİLİM DALI**

Danışman Prof.Dr. Kadir AYDIN

Yıl : 2004, Sayfa sayısı : 215

Jüri Prof.Dr. Kadir AYDIN

Prof.Dr. Beşir ŞAHİN

Doç.Dr. Osman BABAARSLAN

Doç.Dr. Ali PINARBAŞI

Yrd.Doç.Dr. Hüseyin AKILLI

Bu çalışmada, geleneksel pompalardan farklı bir radyal açıklığa ve salyangoz kalınlığına (çark yarıçapının %20 si kadar ve 8 mm salyangoz kalınlığı) sahip olan difüzörlü bir santrifüj pompa içindeki akış, Sonlu Hacimler Yöntemini içeren FLUENT paket programıyla nümerik olarak incelenmiştir. Akış üç boyutlu, viskoz sıkıştırılmaz, türbülanslı, daimi ve daimi olmayan koşullarda kabul edilmiştir. Bu pompa, beş geriye dönük kanatlı çark ve dokuz vanalı difüzörden oluşur. Çark devri 890 devir/dakikadır. Santrifüj pompa farklı katı model programları kullanılarak modellenmiştir. Akış yapısı üç boyutlu geometride 15 farklı debide, üç farklı radyal açıklıkta, daimi ve daimi olmayan akış durumları için incelenmiştir. Farklı radyal açıklıkların pompa performansı üzerine etkisini gözlemlemek amacıyla, her bir açıklık için performans eğrileri elde edilmiştir. Nümerik çalışmanın geçerlilik durumunu belirlemek amacıyla nümerik sonuçlarla deneysel PIV ile elde edilen sonuçların mukayesesi yapılmıştır. Deneysel çalışmada %20 radyal açıklık durumu için beş farklı debi değeri alınmıştır. Ayrıca %15 ve %10 radyal açıklık durumları için çark ve difüzörde akış yapısı nümerik olarak incelenmiştir. Elde edilen sonuçlardan, %20 radyal açıklığın optimum pompa performansı için daha uygun olduğu, açıklığın azaltılması durumunda pompa içerisinde akışın olumsuz olarak etkilendiği ve pompada basınç çalkantılarına yol açtığı görülmüştür.

Anahtar Kelimeler : Santrifüj pompa, Hesaplamalı Akışkanlar Dinamiği (HAD), Sonlu Hacimler Yöntemi, Çark-Difüzör etkileşimi, Radyal açıklık etkisi

ACKNOWLEDGEMENT

I would like to express my sincere appreciation to my supervisor, Prof.Dr. Kadir AYDIN, for his support, guidance, encouragement, invaluable understanding and patience through the study. He showed the right way with his experiences and great knowledge when constructing the model and at the analysis.

I owe special thanks to Prof.Dr. Beşir ŞAHİN, Head of Department of Mechanical Engineering, for his help. I really appreciate his providing everything necessary to complete this work and keeping me on the right track by making valuable suggestions on many aspects of this research.

My sincere thanks go to Assoc.Prof.Dr. Ali PINARBAŞI, who introduced me to the subject of this thesis, and helped at the analysis and conclusion of experimental results.

My special thanks go to Assist.Prof.Dr. Hüseyin AKILLI for his support at the Department for satisfying the required conditions to study.

I must thank Research Assist. K. Melih GÜLEREN of Mechanical Engineering Department of Cumhuriyet University for technical support.

In addition, I would like to thank all the staff of the Department of Mechanical Engineering of Çukurova University and Cumhuriyet University and especially Research Assist. Cahit GÜRLEK, Mustafa ÖZCANLI, Durmuş Ali BİRCAN, N. Adil ÖZTÜRK, Ahmet FERTELLİ, Mr. M. İlteriş SARIGEÇİLİ and Mr. Cevdet YILDIRIM.

Finally, I would like to thank wholeheartedly my wife, Dr. Emine ÖZTÜRK, my sons, Ozan and Eren, and my parents for their endless patience and trust. My wife also deserves special thanks for her support during this study.

CONTENTS

Page

ABSTRACT	I
ÖZ	II
ACKNOWLEDGEMENT	III
CONTENTS	IV
LIST OF TABLES	VI
LIST OF FIGURES	VII
NOMENCLATURE	XII
1. INTRODUCTION	1
2. PREVIOUS STUDIES AND OUTLINE OF PRESENT STUDIES	9
2.1 Previous Studies	9
2.1.1. Experimental Studies	17
2.1.2. Numerical Studies	23
2.2 Outline of Present Study	32
2.2.1 Outline of Dissertation	33
3. MATERIAL AND METHODS	34
3.1 Pump Geometry and Modeling	35
3.2 Experimental Results Used in This Study	46
3.3 Pump Models and Grid Generation	50
3.3.1 Grid Generation	50
3.3.2 Grid Refinement Study	57
3.3.3 The Pump Models Grid	62
3.4 Governing Equations	71
3.4.1 The Mass Conservation Equation	72
3.4.2 Momentum Conservation Equations	72
3.4.3 Transport Equations	73
3.5 Modeling of Flow	73
3.5.1 The MRF Model	75
3.5.2 Sliding Mesh Model	77
3.5.3 Rotating Reference Frame	78

3.5.4 Turbulent Modeling and RNG $k - \varepsilon$ Model	81
3.6 Boundary Conditions	84
3.6.1 Solid and Fluid Conditions	85
3.6.2 Flow Inlets and Exits	86
3.6.2.1 Constant velocity inlet boundary conditions	86
3.6.2.2 Transient velocity inlet boundary conditions	87
3.6.3 Pressure Outlet Boundary Conditions	90
3.6.4 Wall Boundary Conditions	90
3.7 Structure of a CFD Code	91
3.8 Numerical Schemes	93
3.9 Discretization Methods	96
3.10 Numerical Solution Control	100
4. RESULTS AND DISCUSSIONS	102
4.1 Performance Characteristics	106
4.2 Comparison of Experimental and Numerical Results	110
4.3 Comparison of Steady and Unsteady Flow	138
4.4 Radial Gap Effect	144
4.5 Pressure Fluctuations Distribution inside the Pump	188
5. CONCLUSIONS	198
5.1 Future Studies	204
REFERENCES	205
CURRICULUM VITAE	

LIST OF TABLES

Page

Table 3.1	The pump dimensions and operating conditions	39
Table 3.2	Quality classification based on skewness	58
Table 3.3	The 2-D Grid size of all fluid zones and complete domain	68
Table 3.4	The 3-D Grid size of all fluid zones and complete domain	69



LIST OF FIGURES

Page

Figure 1.1.	Nomenclature of impeller and diffuser geometry	1
Figure 2.1	The impeller	9
Figure 2.2	Separated jet-wake flow in impeller	10
Figure 2.3	Vaned diffuser	13
Figure 2.4	Vaned diffuser configurations	14
Figure 2.5	The leading edge-trailing edge vortex	15
Figure 3.1	Machine drawing of the impeller	40
Figure 3.2	The pump geometry and location of the PIV measurements	41
Figure 3.3	A schematic description of the pump loop	42
Figure 3.4	The pump model and fluid zones (2-D)	42
Figure 3.5	a) Wire model b) Solid model c) Mesh model of the impeller (3-D)	43
Figure 3.6	The 3-D Pump Model, a) wire model b) solid model and fluid zones	45
Figure 3.7	Schematic of the optical setup used to create a laser light sheet	47
Figure 3.8	The performance and efficiency curve of the centrifugal pump with diffuser vanes	49
Figure 3.9	Grid components	52
Figure 3.10	Cell types	56
Figure 3.11	A view of used grids for grid independency a) 200.000 elements b) 300.000 elements c) 400.000 elements	59

Figure 3.12	The performance curves for different grids	60
Figure 3.13	The positions of impeller-diffuser and gaps	61
Figure 3.14	Mesh elements	63
Figure 3.15	A view of model mesh a) 2D mesh b) 3D mesh	66
Figure 3.16	The views of the distribution of the mesh elements in the 3D pump model	67
Figure 3.17	Coordinate system for relative velocity	76
Figure 3.18	Impeller-diffuser interaction	80
Figure 3.19	a) The C code of velocity profile b) Average velocity magnitude on inlet (for one cycle)	89
Figure 3.20	Overview of the Segregated Solution Method	95
Figure 3.21	Control volume used to illustrate discretization of a scalar transport equation	99
Figure 4.1	Static pressure calculation points; between impeller and diffuser, after diffuser, and between diffuser blades	104
Figure 4.2	A sample of histories of residuals	105
Figure 4.3	Typical pump head-flow curve, BEP at constant speed	107
Figure 4.4	a) The Experimental and numerical $H - Q$ curves of the centrifugal pump b) The Experimental and numerical $\eta - Q$ curves of the centrifugal pump for (Case I)	109
Figure 4.5	a) Averaged velocity b) Instantaneous velocity, for 3.80 l/s	118
Figure 4.6	a) Averaged velocity b) Instantaneous velocity, for 3.48 l/s	119
Figure 4.7	a) Averaged velocity b) Instantaneous velocity, for 3.16 l/s	120

Figure 4.8	a) Averaged velocity b) Instantaneous velocity, for 2.86 l/s	121
Figure 4.9	a) Averaged velocity b) Instantaneous velocity, for 2.51 l/s	122
Figure 4.10	a) Contours of static pressure b) Contours of velocity magnitude c) Velocity vectors d) Contours of vorticity magnitude e) Contours of TKE (for 3.80 l/s) – Case I	123
Figure 4.11	a) Contours of static pressure b) Contours of velocity magnitude c) Velocity vectors d) Contours of vorticity magnitude e) Contours of TKE (for 3.48 l/s) – Case I	128
Figure 4.12	a) Contours of static pressure b) Contours of velocity magnitude c) Velocity vectors d) Contours of vorticity magnitude e) Contours of TKE (for 3.16 l/s)	131
Figure 4.13	a) Contours of static pressure b) Contours of velocity magnitude c) Velocity vectors d) Contours of vorticity magnitude e) Contours of TKE (for 2.86 l/s) – Case I	134
Figure 4.14	a) Contours of static pressure b) Contours of velocity magnitude c) Velocity vectors d) Contours of vorticity magnitude e) Contours of TKE (for 2.51 l/s) – Case I	137
Figure 4.15	Steady and unsteady performance curves for a) Case I, b) Case II, and c) Case III	142
Figure 4.16	For unsteady flow a) Velocity b) Mass flow rate c) Pressure, (for 4.12 l/s)	143
Figure 4.17	The centrifugal pump a) $H - Q$ curves b) $\eta - Q$ curves c) Error rates (%) relating to experimental values, for Case I, Case II, Case III	157

Figure 4.18	a) Contours of static pressure b) Contours of velocity magnitude c) Velocity vectors d) Contours of vorticity magnitude e) Contours of TKE (for 4.74 l/s) – Case I	158
Figure 4.19	a) Contours of static pressure b) Contours of velocity magnitude c) Velocity vectors d) Contours of vorticity magnitude e) Contours of TKE (for 4.12 l/s) – Case I	161
Figure 4.20	a) Contours of static pressure b) Contours of velocity magnitude c) Velocity vectors d) Contours of vorticity magnitude e) Contours of TKE (for 4.12 l/s) – Case II	164
Figure 4.21	a) Contours of static pressure b) Contours of velocity magnitude c) Velocity vectors d) Contours of vorticity magnitude e) Contours of TKE (for 4.12 l/s) – Case II	167
Figure 4.22	a) Contours of static pressure b) Contours of velocity magnitude c) Velocity vectors d) Contours of vorticity magnitude e) Contours of TKE (for 3.48 l/s) – Case II	170
Figure 4.23	a) Contours of static pressure b) Contours of velocity magnitude c) Velocity vectors d) Contours of vorticity magnitude e) Contours of TKE (for 4.74 l/s) – Case III	173
Figure 4.24	a) Contours of static pressure b) Contours of velocity magnitude c) Velocity vectors d) Contours of vorticity magnitude e) Contours of TKE (for 4.12 l/s) – Case III	176
Figure 4.25	a) Contours of static pressure b) Contours of velocity magnitude c) Velocity vectors d) Contours of vorticity magnitude e) Contours of TKE (for 3.48 l/s) – Case III	179
Figure 4.26	The static pressure contour on hub and shroud for 4.12 l/s	182

Figure 4.27	The velocity contour on hub and shroud for 4.12 l/s	183
Figure 4.28	The static pressure and velocity contours on impeller blades for 4.12 l/s	184
Figure 4.29	The static pressure and velocity contours on impeller for 4.12 l/s	185
Figure 4.30	Three-dimensional view of inside the pump flow for 4.12 l/s a) Static pressure b) Velocity contour c) Velocity vector	186
Figure 4.31	Pressure distributions between impeller and diffuser for Case I, II, III at six flow volume flow rates (1.96, 2.86, 3.48, 4.12, 4.74, 5.38 l/s)	195
Figure 4.32	Pressure distributions between diffuser vanes for Case I, II, III at six flow volume flow rates (1.96, 2.86, 3.48, 4.12, 4.74, 5.38 l/s)	196
Figure 4.33	Pressure distributions after diffuser for Case I, II, III at six flow volume flow rates (1.96, 2.86, 3.48, 4.12, 4.74, 5.38 l/s)	197

NOMENCLATURE

$\vec{a}$	Acceleration (m/s^2)
A	Area (m^2), Velocity Amplitude (dimensionless constant)
$\vec{A}$	Area surface vector
$C_{1\varepsilon}, C_{2\varepsilon}$	Constants for $k - \varepsilon$ turbulence model
D	Diameter (m)
$\vec{F}$	Force vector (N)
$\vec{g}$	Gravitational acceleration (m/s^2)
h	Enthalpy (J/kg)
I	Unit tensor
H	Head (Pa)
m	Mass (kg)
p	Pressure (Pa)
Q	Flow Rate (m^3/s)
Re	Reynolds number
r	Radius (m)
$\vec{r}$	Position vector
r_v	Logarithmic radius of volute (m)
S	Source term
t	Time (s)
u, v, w	Velocity magnitude (m/s) (directional subscripts x, y, z, r, t)
V	Volume (m^3)

Greek Symbols

α	Thermal diffusivity (m^2/s)
Δ	Change in variable, final-initial
∇	Gradient
δ	Delta function (units vary)
ε	Turbulent dissipation rate (m^2/s^3)
μ	Dynamic viscosity ($Pa \cdot s$)
ν	Kinematic viscosity (m^2/s)
ρ	Fluid density (kg/m^3)
k	Kinetic energy per unit mass (J/kg)
θ	Angle (degree)

φ	Angular position (degree)
ϕ	Equivalence ratio (dimensionless)
Ω	Angular velocity (s^{-1})
ω	Specific dissipation rate (s^{-1})
τ	Stress tensor (Pa)

Abbreviations

1D, 2D, 3D	One, two, three dimensional
BEP	Best Efficiency Point
CAD	Computer Aided Drawing
CFD	Computational Fluid Dynamics
CPU	Central Processing Unit
FDM	Finite Differences Method
FEM	Finite Elements Method
FVM	Finite Volume Method
LDA	Laser Doppler Anemometer
LES	Large Eddy Simulation
MRF	Multiple Reference Frame
NSEs	Navier-Stokes Equations
PIV	Particle Image Velocimetry
RANS	Reynolds-Averaged-Navier-Stokes
RNG	Renormalization Group Theory
RSM	Reynolds Stress Model
SM	Sliding Mesh
TKE	Turbulent Kinetic Energy
UDF	User Defined Function

1. INTRODUCTION

Pumps are used in a wide range of industrial and residential applications. Pumping equipment is extremely diverse, varying in type, size, and materials of construction. There have been significant new developments in the area of pumping equipment since the early 1980s. Pumps are divided into two fundamental types based on the manner in which they transmit energy to the pumped media: kinetic or positive displacement. In kinetic displacement, a centrifugal force of the rotating element, called an impeller, “impels” kinetic energy to the fluid, moving the fluid from pump suction to the discharge. On the other hand, positive displacement uses the reciprocating action of one or several pistons, or a squeezing action of meshing gears, lobes, or other moving bodies, to displace the media from one area into another (i.e., moving the material from inlet to outlet).

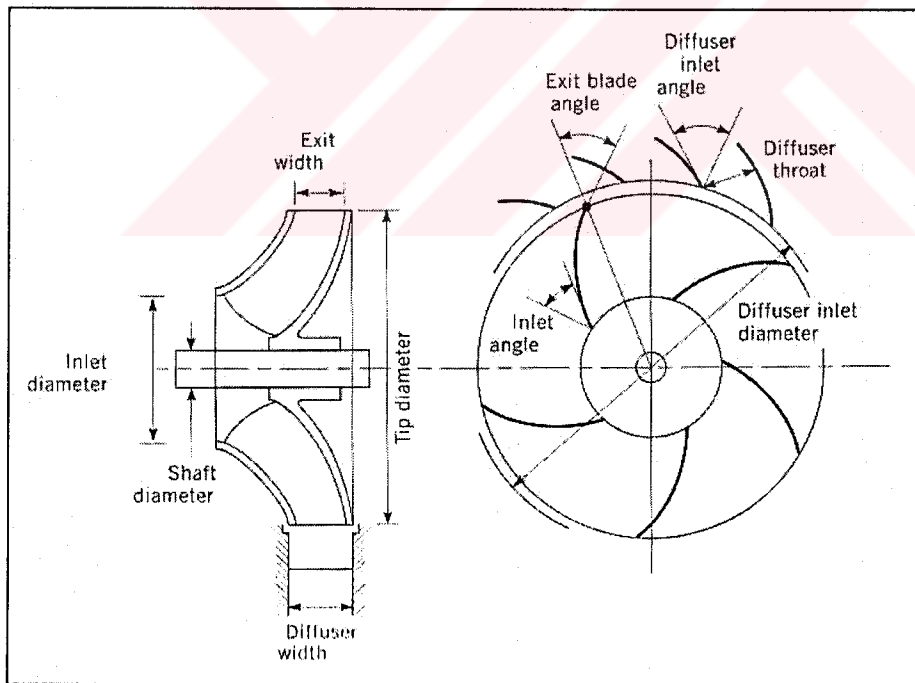


Figure 1.1 Nomenclature of impeller and diffuser geometry (Tuzson, 2000)

A calculation procedure to estimate the theoretical performance of a pump is an indispensable tool in pump design. Performance needs to be known, not only at

the rated, best efficiency point, but also off design. Pump specifications often impose special requirements, such as head at shutoff, maximum power demand, rate of head rise to assure stability, and so on. A good pump design process requires trial-and-error iteration, a check on anticipated performance with a trial geometry, and progressive approximation to be optimal design configuration. Generally, the more accurate and detailed these calculations are, the greater the number of input variables needed: the desired head, flow rate, and rotational speed, but also the details of the geometrical description of the impeller and housing.

The effect of individual variables, such as number of impeller blades, inlet diameter, and impeller exit width etc. (figure 1.1) on pump performance is complex and often contradictory, depending on the value of other variables. Their effect on performance is difficult to predict from memory or from empirical rules, because of the complexity of their interdependence (Tuzson, 2000).

Flow in radial pumps very complex and not clearly understood. Numerical solutions of the flow equations for the rotating part of pumps have become an important tool for designers during the last decade. It is wanted that the flow pattern and properties inside an impeller to be clarified to improve the efficiency and performance. That is why in this study it is attempted to find numerical solutions different kind of impellers where experimental performances are available.

Flow in centrifugal pumps produces a complex three-dimensional phenomenon involving turbulence, secondary flows, unsteadiness, etc. Moreover, the geometry is complex and asymmetric due to the volute shape. The relative movement between impeller and volute generates an unsteady interaction which affects not only the overall pump performance (flow structure, losses), but is also responsible for pressure fluctuations. Pressure fluctuations interact with the volute casing or even with the circuit and give rise to dynamic effects (mainly unsteady forces) over the mechanical parts, which are one of the most important sources of vibration and hydraulic noise. Both experimental and numerical approaches have been reported and have contributed to the understanding of the highly complex flow interactions that occur in a centrifugal pump.

The complexities of the flow in a centrifugal pump command attention when the energy level or power input for a given size becomes relatively large. Fluid phenomena such as recirculation, cavitation, and pressure pulsations become important; “hydraulic” and mechanical interactions-involving stress, vibration rotor dynamics, and the associated design approaches, as well as the materials used-become critical; and operational limits must be understood and respected. The complexity of any study of the flow in a centrifugal impeller is obvious and has stirred the research work over the last decades. Many studies have been carried out, but even nowadays some flow occurrences are still under study and far from being fully understood (Engeda, 1998).

The unsteadiness of the flow at the leading edge of a vaned diffuser represents a source of flow efficiency and instability in a centrifugal pump. Furthermore, the internal flow of the impeller can be affected by asymmetric downstream conditions, which results in extra flow unsteadiness and instabilities. The improvement of machine performances can only be achieved if there is a progress in the comprehension of the nature of the complex flow that develops at the gap between the impeller and the diffuser.

Modern turbomachinery operates under very complex three-dimensional flow conditions, and further improvement requires detailed knowledge of the flow structure. Particularly, the need to estimate off-design conditions, secondary flows, and heat transfer forces us to look at viscous models. Even if the absolute level in accuracy of the Computational Fluid Dynamics (CFD) prediction depends on the complexity of the problem under investigation, important savings in costs and time have been obtained in design by the use of CFD.

In the last decade, Fluid Dynamics has undergone impressive evolution in and understanding complex flow features. In this process, CFD has played an increasingly important role. Two and three-dimensional viscous solvers have become widely used as basic tools to tackle the most challenging aspect of analysis and design. Recently, they have also been successfully coupled to advanced experimentation, with the measurement and the understanding of complex flow structures facilitated by a preliminary theoretical prediction (Gonzalez et al., 2003).

Viscous solvers can be used for optimization in the final part of the component design, as well as for choosing between different configurations, therefore reducing costs related to a redesign. The real flow inside a turbomachine is unsteady and strongly influenced by impeller-diffuser interactions, wakes and clearance effects. However, the design process takes place in several phases that goes from one-dimensional to three-dimensional and from study to ones that account for unsteadiness and critical frequency. In each of these phases, CFD plays an important role, and the successful design of the next generation of turbomachines will be strongly influenced by the existence of the reliable methods at the various levels of modeling. Combined with measurements it provides a complementary tool for simulation, design, optimization and, most important of all, analysis of the flow field inside the turbomachine. CFD is the most convenient possibility to obtain detailed flow information in rotating parts, in which measurements would be extremely expensive or even impossible to realize.

CFD techniques are very useful tools for design and analysis of complex fluid machinery. Although most of the current CFD research activities are on steady flows, there is an increasing interest in unsteady flow computations. Unsteady blade row interaction is an inherent phenomenon for all kinds of turbomachinery due to relative motion of impeller-diffuser blade rows. The effects of the unsteadiness on time-averaged performance, as well as on blade forced responses, are of great interest for designs of modern turbomachinery.

Among its direct advantages are fewer builds and shorter times to market. CFD simulations for internal flows and turbomachines in particular, must, however, contend with:

- disparate scale of gradients near walls end in the flow core;
- complex turbulent flows and their modeling;
- complex physical phenomena such as shocks, separation, choking, and combustion;
- different time scales for some of these phenomena;
- adjacent rotating and non-rotating components and their unsteady interaction;
- problems of a multidisciplinary nature such as aeroelasticity and aeroacoustics.

In an industrial environment large-scale numerical simulations for stress and fluids are used daily in the analysis and design of turbomachinery. The CFD analysis/design codes currently in use are based on Finite Volume Method (FVM) and Finite Element Method (FEM). Components designed using CFD include stators, rotors, diffusers, ducts, mixers, combustors and cooling passages. CFD-based designs rely upon the code(s) to judge performance and, when feasible, absolute performance values for a component. Cost-effective designs require judicious balance between design complexity and the design time, keeping in mind the current accuracy limitations of numerical predictions. As a consequence, the goal is to develop accurate and efficient tools to be used in each of the various stages of the design.

The equations of fluid dynamics can be solved numerically with the aid of modern computers. The set of techniques and procedures to achieve this has led to the development of what is known as CFD. Although it is not possible to leave out real experiments, numerical experiments using CFD allow the determination of certain trends and give an insight into the physics of complex problems that would be impractical to study experimentally. The subject has now advanced to the point where commercial packages are available and can be used to simulate or predict complex fluid dynamics problems. Any discrepancy between predictions and experiments can only be explained in terms of the assumptions made in the development of the code. The technique is very powerful and spans a wide range of industrial and non-industrial application area.

The present research addresses comparisons between detailed flow visualization using the technique of Particle Image Velocimetry (PIV) and CFD modeling, in a centrifugal pump with diffuser vanes. Two classes of unsteady flow phenomena i.e. the gap effect within the diffuser and impeller blades and the effect of flow rate level will be investigated. The observed flow structures and turbulence measurements, which obtained by Sinha (1999) are then used as tools to test and examine assumptions underlying turbulence models that are used for flow predictions, and to deduce fundamental properties of the flow that may serve as guidance in model development.

CFD probably plays a greater part in the hydraulic design of pumps than it does in any other engineering application. For many years the design of a modern pump or compressor has been unthinkable without the help of CFD and this dependence has increased as more of the flow becomes amenable to numerical prediction. The benefits of CFD range from shorter design cycles to better performance and reduced cost and weight.

In spite of the considerable efforts devoted to the study of vaneless space effect in compressor and pumps, the mechanics of the phenomenon is not yet well understood. Most of the publications show that, the effect of the gap ratio for the pump flow is dependent on the impeller geometry and all the experimental and computational work were performed at different gap ratio. Therefore, the optimum distance is still not yet well understood and will be helpful for future pump designer. Especially on the effect of the gap between the impeller and diffuser of the centrifugal pump, there is still very little experimental and numerical data. Most of the researches consist of special blade geometry. Therefore it will be beneficial work to solve reason of the gap effect for the future pump designer.

A pump model it is usually designed to achieve optimum performance for the customer with minimum cost and time. The numerical solution of the internal flow field plays an important role to achieve design objectives. With the helpful of CFD codes the designer is able to produce more engineered and more clearly understood designs. The commercial advantages arising from the use of CFD in design process as follows.

- More accurate prediction of the flow pattern,
- Natural physics limits of the flow (flow separation, cavitation),
- The use of graphical visualization techniques which enables the designer a better understanding of the flow field,
- Shorter development times without using experimental prototypes.

The design of turbomachines, such as pumps and compressors, has reached the stage where improvements can only be achieved through a detailed understanding of the internal flow. The prediction of the flow in such equipment is very complicated due to the three-dimensionality of the flow and the highly curved passages in rotating

impellers. Furthermore, turbomachines show unsteady behavior, especially under off design conditions, as a result of interaction between impeller and diffuser or impeller and volute. Considering these complexities, computer simulations are becoming increasingly important.

Pump manufacturers to design more-efficient and longer-lived products are using CFD, a simulation tool originally developed for academic research and aerospace. CFD is used to solve the equations that describe fluid flow and generate a three-dimensional model of that flow. The development of more-advanced CFD codes, coupled with the introduction of more-powerful personal computers, has encouraged CFD's use to study the behavior of fluid inside a pump. Armed with this knowledge, pump manufacturers can design more-efficient pumps, thus improving the efficiency of the fluid process the pump serves. Superior pump design also minimizes cavitation, which occurs when the local pressure drops below the vapor pressure of the fluid. Cavitation causes turbulence that wears out the impeller and blades. By reducing cavitation, pump designers reduce maintenance downtime and costs.

Diffuser pumps are found in a wide range of applications, such as process industry and space engineering. These modern applications require that the pump systems can be operated over a wide range, which, therefore, necessitates knowledge of pump performance from the shut-off to run-out flow. However, most previous research, especially numerical approaches, focused on the design condition or the near-design flow region.

There are a lot of parameters like efficiency, pressure rise, flow coefficient, work input or geometry ratios that specify the design and typify the operation of centrifugal pumps. Obtaining these parameters by testing is sufficient for conventional repetitive design and for highly developed individual machines, but it is not adequate when new levels of performance are sought for a reasonable investment and when precise predictability for new design are demanded. The determination of above-mentioned parameters by means of measuring cannot be replaced but only accompanied by means of CFD. However, rather expensive experiments are not adequate for preliminary testing of new developments or for parameter studies of

new design variants. These are the main application fields of CFD in industrial use. In the last years new insights are achieved by the introduction of the CFD in the field of turbomachinery.

This study presents the validity of CFD methods for predicting design and off-design performance of diffuser pumps. In both design and off-design conditions, the effects of impeller-diffuser interaction and the pump system characteristics become, significant, and both need to be taken into account. An arbitrary sliding interface was exploited to consider the influence of impeller-diffuser interaction.

The present study has demonstrated that the characteristic curve of the pump and the pump CFD model produce a good agreement for both pump design and off-design performance. The CFD methods can provide an alternative to dealing with pump design and off-design performance.

2. PREVIOUS STUDIES AND OUTLINE OF PRESENT STUDIES

The rotating impeller imparts energy to the fluid. It is the most important, the only rotating element of the pump. The diffuser, following the impeller, can transform kinetic energy into pressure energy but cannot increase the total energy of the fluid. The impeller contains radial flow passages, inclined toward the axial direction in mixed flow pumps, formed by rotating blades arranged in a circle. A disk, the shroud, covers the blades on the front. Another disk, the hub, in the back, connects the impeller assembly to the shaft. The flow enters axially near the center of rotation and turns in the radial direction inside the impeller as seen in figure 2.1 (Tuzson, 2000).

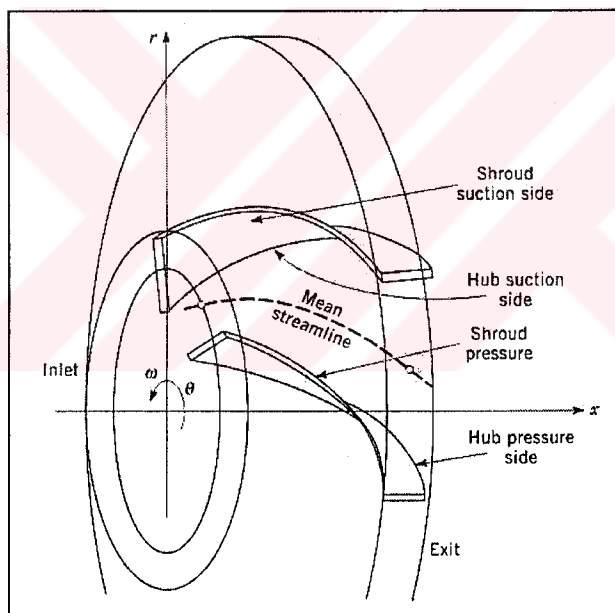


Figure 2.1 The impeller (Tuzson, 2000)

In mixed flow pumps and in the inlet of radial flow pumps, the streamlines are inclined from the axial direction toward the radial direction. The shape of the blades and the resulting flow pattern in the impeller determine how much energies transferred by a given size impeller and how efficiently it operates. To find the values of the exit angular momentum, the velocities must be examined at the inlet of

the impeller, the leading edge of the blades, and at the impeller exit, the trailing edge of the blades. The flow follows certain streamlines inside the rotating impeller, approximately parallel to the blade surface.

It appears that the energy added to the fluid is proportional to the pressure difference across the blade, which is usually referred to as the blade loading. In the flow passage between two blades, the pressure decreases in the circumferential direction, approximately linearly, from the pressure side of the blade facing in the direction of rotation to the suction side. As will be seen, there is a corresponding increase in the velocity in the impeller. If the blade loading, and therefore the pressure difference, becomes very high, the velocity at the pressure side of the blade can become very small or vanish, which results in flow separation and losses.

A shear layer, or separation streamline, between two flow regions of different fluid energy is stabilized if the acceleration and corresponding pressure rise perpendicular to the shear layer are directed from the low-energy region toward the high energy region. When separation appears in the impeller, two flow regions of different energy can be distinguished: the separated region and the main stream, as shown in figure 2.2.

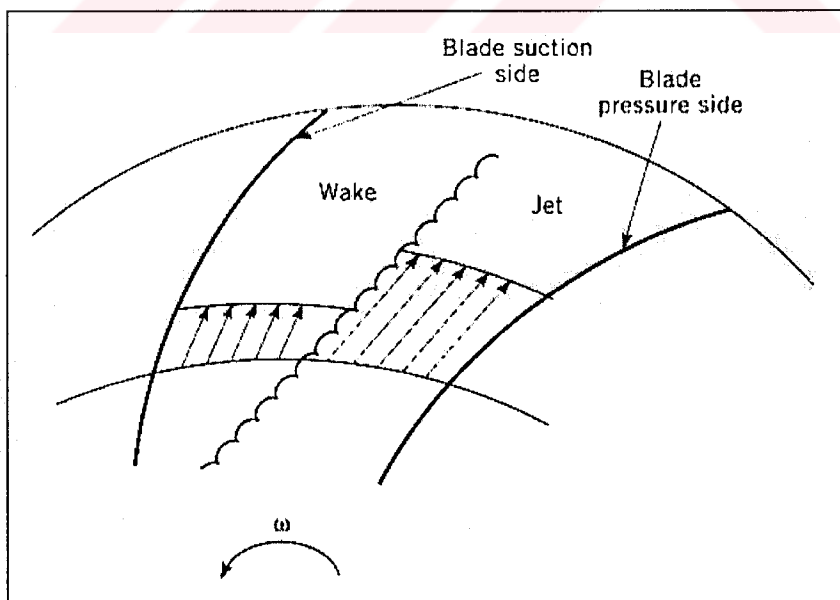


Figure 2.2 Separated jet-wake flow in impeller (Tuzson, 2000)

The relative velocity remains constant, but different, on either side of the streamline separating the two regions. The acceleration, and corresponding pressure increase, is directed from the suction side to the pressure side. A separated region of low energy will be stable and can then persist on the suction side. A low-energy separated region -for example, a boundary layer- on the pressure side would be unstable and would tend to migrate along the hub and shroud surfaces to the suction side. Such secondary flows do exist and have been observed in impellers.

In the case of heavy blade loading, potential flow calculations may predict that flow separation will occur on the pressure side. However, since a separated region cannot persist on the pressure side, the flow switches to a jet-wake flow pattern. Consequently the ideal potential flow model gives a false picture of the flow pattern. A flow pattern consisting of a stable separated region or wake on the suction side and a main stream or jet near the pressure side of the blade is called a jet-wake flow pattern. Such a flow pattern appears only under certain conditions.

The flow inside centrifugal pump is characterized of a jet-wake structure. Therefore the prediction by numerical simulation has turned out to be very challenging.

Although usually thin, impeller blades have a definite thickness and leave a wake in the flow at the impeller exit. It can be assumed that the cross section of the flow passage between the blades of the rotating impeller suddenly increases by the amount of the blade thickness as the flow leaves the impeller. Since the blade thickness is considerably less than the passage width, the area increase and the corresponding loss are relatively small and may or may not be negligible. A sudden expansion loss calculation can provide a rough estimate of the magnitude of such a blade-wake mixing loss. There is a reason to believe, however, that such a flow model is not valid and that the mixing loss is relatively smaller than would be expected.

In reality the flow leaving the impeller is unsteady when considered by a stationary observer in an absolute frame of reference. The absolute flow velocity at the impeller exit is not parallel to the wake. The mixing process is therefore entirely different from a steady wake mixing. The fluid in the wake is not low energy as in

the case of wake behind a stationary object, but can move with a higher velocity than the main flow, approximately with the tip speed. In addition, trailing vortices are likely to be present in the corners between the blade and the hub and shroud surfaces.

In some pumps a radial, vaned diffuser follows the impeller exit. A short radial distance, a vaneless space, precedes the diffuser. The diameter of the diffuser inlet is usually about 5% to 10% greater than the impeller diameter. As described above, the pressure and relative velocity vary from the pressure side of the blades to the suction side at the impeller exit. In the stationary frame of reference a fluctuating flow comes off the impeller. If the leading edge of the diffuser vanes is too close to the impeller, the flow fluctuations impinge on the diffuser vanes and can produce undesirable noise and pressure pulsations at the pump exit. The vaneless space serves to allow the flow to mix, become more uniform, and adjust to the inlet of the vaned diffuser. The pressure pulsations can be reduced by slanting the leading edge of the diffuser vanes in the circumferential direction with respect to the trailing edge of the impeller blades. The trailing edge of the impeller blades then sweeps gradually across the leading edge of the diffuser vanes instead of overlapping exactly at a certain moment. On small multistage pumps sometimes a few widely spaced diffuser vanes strongly inclined in the tangential direction are positioned relatively close to the impeller exit. Their purpose is to attempt to slow down the high-speed flow immediately after the impeller exit and recover some pressure. The radial component of velocity in the diffuser is imposed by the continuity equation, the ratio of the flow rate, and the available cross-sectional area. Diffusion is accomplished primarily by reducing the tangential component of the velocity.

The direction of the diffuser leading edge and the diffuser throat size, the cross-sectional area between two vanes perpendicular to the velocity, must match the magnitude and direction of the velocity approaching from the impeller exit. The expected pressure recovery in the vaned diffuser follows the traditional diffuser experience. The effectiveness of diffusers consisting of widely spaced airfoil-shaped vane is more difficult to estimate. General airfoil theory might provide a measure of the turning of the velocity, and corresponding diffusion that could be expected.

At off-design flow rates the impeller exit velocity is not aligned with the diffuser inlet, and leading edge separation occurs. Also, a mismatch appears between the approach velocity and throat velocity, which results in a loss.

Considerable kinetic energy remains in the flow leaving the impeller, a portion of which should be converted to pressure head with a vaned diffuser (figure 2.3). In multistage pumps the flow leaving the impeller must be returned to the inlet of the next stage, and some pressure recovery is desirable in the process. The inlet section of the return passage often acts as a diffuser. The inlet leading edge of the diffuser, or of return passage, must align with the direction of the absolute velocity leaving the impeller. The exit velocity and its components in the radial or meridional and tangential direction are known from the impeller exit velocity diagram.

The inlet of vaned diffuser follows closely the impeller exit, the radial distance being perhaps 5% of the impeller radius. In the case of a return passage, more space is left, and a correction needs to be made when calculating the parameters.

The number of diffuser or return passage vanes usually exceeds by one the number of impeller blades. The intensity of blade passing noise also depends on the spacing (radial gap) between the impeller and the diffuser inlet.

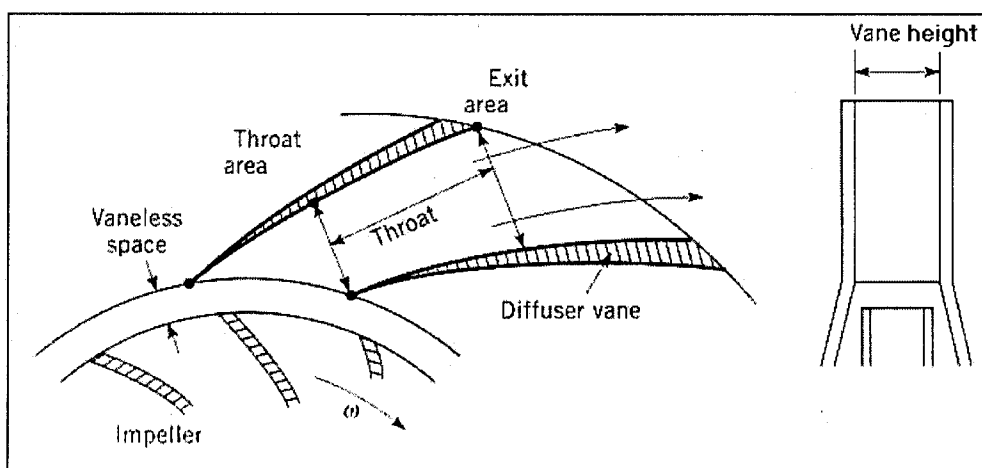


Figure 2.3 Vaned diffuser

A radial vaned diffuser can consist of passages with straight, diverging walls, inclined to the radial direction as shown in figure 2.4. The diffuser area ratio usually remains below 2. As an alternative, the diffuser passages can be formed by airfoils, or slightly curved, circular partitions, arranged in a circle. The location immediately after the impeller exit, where the velocity is high, offers the greatest potential for pressure recovery. Even a modest diffusion can have a significant effect. If the radial space is limited, widely spaced, thin, curved partitions or airfoils are used close to the impeller exit.

Often a wide space remains between the impeller exits and return passage inlet. It can be assumed that the fluid rotates in this space with a constant regular momentum. This space merges with the clearance space between the hub and shroud faces of the impeller and the housing.

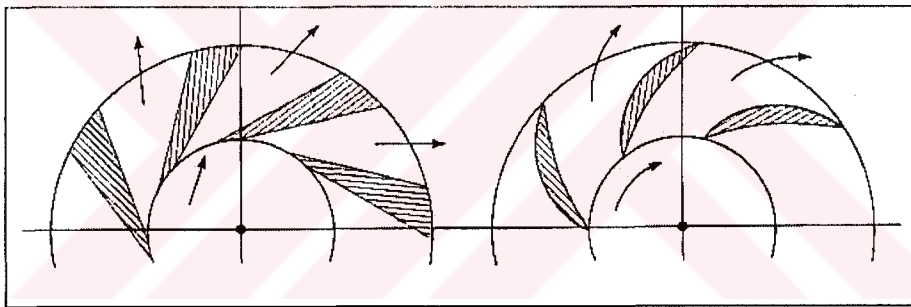


Figure 2.4 Vaned diffuser configurations

The impeller often discharges directly into the volute, which is a spiral-shaped flow passage usually of circular or trapezoidal cross section. Its cross section increases gradually around the impeller periphery, starting from the volute tongue and ending at the throat volute. The volute tongue directs the total flow, collected from around the impeller, through the throat to the pump exit flange. When the flow rate is greater than the design flow rate, the flow generally accelerates in the volute, and the pressure increases circumferential direction. When the flow rate is less than the design flow rate, the volute velocity tends to decrease, and the pressure increases circumferentially around the impeller. Because of this pressure increase or decrease, a transverse pressure force appears on the impeller at off-design conditions,

which must be resisted by the pump bearings. As in the case of the vaned diffuser, a radial space is left between the impeller periphery and the volute tongue. The circle centered on the center of rotation and touching the volute tongue, the base circle, usually has a diameter 10% greater than that of the impeller. At flow rates less than the design flow rate, the volute tongue deflects some of the flow approaching the volute throat, which passes between the volute tongue and the impeller is very large, too much flow returns unnecessarily, and losses increase. If the space is too small, strong pressure fluctuations at the blade passing frequency can occur.

The flow separates at the leading edge of the impeller blade if the direction of the relative velocity does not align with the leading edge. Correct pump design usually assures correct alignment between the velocity vector and the blade at the inlet at the best efficiency flow rate. At flow rates below design flow rate, separation occurs on the suction side of the blade. The separated region on the pressure side of the blade becomes unstable. The flow pattern in the pump is three-dimensional, and three dimensional flow calculations would be desirable.

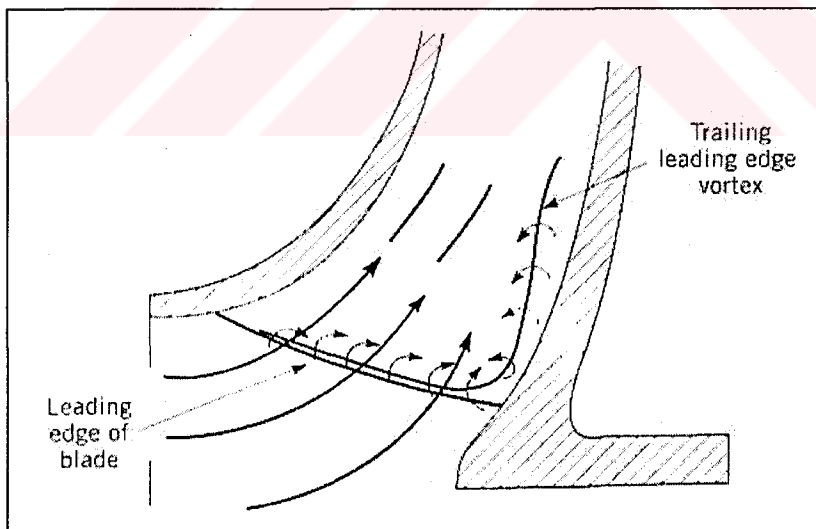


Figure 2.5 The leading edge-trailing edge vortex

The real flow conditions at the impeller blade inlet at incidence are often much more complicated than those of the two-dimensional separation model. If the

flow velocity meets the leading edge at a slant, a tip vortex can develop in the separated region along the leading edge, which trails away toward the hub (figure 2.5). High-specific-speed pumps are particularly sensitive to inlet conditions because the through flow velocities are relatively high and the inlet diameter is also relatively large. The circumferential velocity on the shroud far exceeds the circumferential velocity at the hub. Meridional velocities are high on the shroud and low on the hub because of the curvature of the streamlines. Since flow losses vary with the square of the velocity, high velocities indicate potentially high losses. The most critical conditions are encountered on the shroud streamline.

In both design and operation of pumps for liquid handling applications, the effects of cavitation are important consideration. Cavitation has been defined as the phenomena when the local absolute static pressure of the fluid somewhere in the flow drops below the vapor pressure of the fluid and vapor bubbles appear. As the bubbles are convected downstream, they reach a location where the pressure increases again above the vapor pressure. They collapse and create a sudden, very high-pressure pulse, which can damage the material of a nearby flow passage wall. In centrifugal pumps the lowest pressure usually appears on the impeller blades at the inlet, especially at flow rates higher than the design flow rate, when the flow separates on the leading, pressure side of the blades. It is known that the conditions for the appearance of cavitation in pumps and the extent of the resulting damage. Prediction of these items remains difficult because of the complexity of the phenomena and because several factors affecting cavitation in a certain application remain unknown, such as the air content of the water handled by the pump (Tuzson, 2000).

The interaction between impeller and diffuser considered having strong influence on the flow in highly loaded centrifugal pump. However the knowledge about this influence is still not satisfying.

Radial gap is an often-discussed design parameter influencing pressure ratio, efficiency mass flow range, mechanical loading, noise emissions and size of a flow machines. Many researchers presented investigations varying the radial gap. But

corresponding recommendations to find an optimum radial gap do not exist yet to the large number of contradicting parameters influenced by its.

The objective this study is to the better achieves understanding of the flow phenomena connected with a variation of the radial gap and the entire pump. Furthermore, the general knowledge about the highly unsteady flow field between the impeller exit region and diffuser throat shall be improved.

In order to investigate the impeller-diffuser interaction, the radial gap between impeller exit and diffuser vane inlet was varied at 20%, 15% and of 10% of the impeller radius. The gap very frequently has profound effect on impeller performance. It can cause the impeller to work very inefficiently and cause large vibrations if it is designed improperly.

2.1 Previous Studies

2.1.1 Experimental Studies

The flow in turbomachines involves unsteady wakes, flow separation, high straining, interactions between blade rows etc. These complexities associated with turbulent flows in realistic geometries have, so far, inhibited the development of successful and truly predictive models. Hence, a strong need continues to exist to test carefully closures and basic modeling ideas. Although the actual flow patterns within a centrifugal pump are three-dimensional (3D) and unsteady varying, it is fairly easy, on a one (1D) / two-dimensional (2D), steady flow basis, to make the connection between the basic energy transfer and performance relationships and the geometry or what is commonly termed the “hydraulic design” (more properly “fluid dynamical design”) of impellers and stators and stationary passageways of these machines. When an impeller blade passes by an upstream diffuser blade, it experiences a fluctuation in the fluid forces that act upon it. Considerable effort has already been invested in studying these interactions between the flow within the impeller and the flow between the diffuser vanes of centrifugal turbomachines.

The flow field in turbomachines is extremely complex. This flow field is highly unsteady, with a time scales, ranging from a fraction of the shaft speed to the Kolmogorov time scales of a turbulent flow within a machine (Adamczyk, 1985). The length scales of the flow are also diverse. They range from the circumference of a machine to the Kolmogorov length scale of a turbulent eddy. The flow in turbomachines involves unsteady wakes, flow separation, high straining, interactions between blade rows etc. These complexities associated with turbulent flows in realistic geometries have, so far, inhibited the development of successful and truly predictive models. Hence, a strong need continues to exist to test carefully closures and basic modeling ideas. Instrumentation has evolved rapidly in recent years, in particular PIV, which allows measurement of entire 2D velocity fields (Adrian, 1991), and of 3D fields using Holographic PIV (Barnhart et al., 1995; Zhang et al., 1997). Such experimental data bases can be applied to the problem of studying models for complex and turbulent flow in at least two distinct fashions: (i) One may use many realizations of PIV fields to compute ensemble averaged velocity fields and Reynolds stress fields, which are then compared to results of numerical simulations in the same geometry. The simulation employs a particular turbulence model that is being tested. (ii) One may also use the PIV data directly in order to test and examine assumptions underlying a model, and deduce fundamental properties of the flow that may serve as guidance in model development.

As measured and discussed in Dring et al. (1982) for axial machines, the interactions can be divided into potential flow effects, i.e. the circulation and blockage of a blade (or vane), and wake effects. The former affects both the impeller and the stator and decreases with increasing distance between blade rows. The wake impacts the flow downstream and should persist over larger distances. In cases with larger gaps between blade rows of axial compressors (60% of chord), Gallus et al. (1980) measured weak potential flow interactions; as a result, the stator pressure fluctuations were higher than those of rotors. Arndt et al. (1989, 1990) examined the unsteady pressure and lift in pumps with small gaps between the impeller and the leading edge of the diffuser vanes (1.5% and 4.5% of the impeller tip radius) of a pump. They measured high-pressure fluctuations at the same order as the pressure

rise within the impeller on the suction side of the diffuser vanes, and fluctuations in the lift force which were higher than the mean value. Similarly, Iino and Kasai (1985) measured high-pressure fluctuations in the impeller of centrifugal compressors that depended on blade and vane angles. Akin and Rockwell (1994) used PIV to study the flow in the gap between the impeller and the diffuser vane of a simulated centrifugal pump. They demonstrated the shedding of vortices from the trailing edge of the impeller blade that interacted and caused unsteady separation at the leading edge of the diffuser blade. Eisele et al. (1997) and Muggli et al. (1997) measured and computed the flow for a pump with a narrow gap (8% of the impeller radius). They also demonstrated substantial impact of the impeller orientation on both the mean flow and the turbulence levels. The periodic unsteadiness that depended on the impeller orientation decreased rapidly downstream of the diffuser throat; whereas the turbulence level increased.

A number of authors have treated the problem of the interaction of the impeller and its surrounding. Miner (1989) used a laser velocimeter to measure velocities within the impeller and the volute of a centrifugal pump and found that the relative velocity components distribution varies with the circumferential angle relative to the volute cutwater. Liu (1994) has also used LDA for the internal flow investigation and found that the asymmetric volute alters the relative flow, the flow rate from each impeller passage varied with the volute circumferential position by up to 25% at an off-design flow rate. Inoue (1984), Sideris (1987) and Arndt et al. (1989-1990) have been concerned with the action of the diffuser. For internal flow studies, a large number of the experimental investigations that revealed the presence of a jet-wake structure at the discharge of centrifugal rotors were concerned with compressors as done by Eckardt (1975-1976), Johnston (1976), Johnson (1978), and more recently Rohne (1991) and Ubaldi (1993). However, Krain (1988) found a velocity profile that differed widely from the jet-wake type flow. Previously, the authors (El Hajem et al. (1998)) found a jet-wake structure developing at the impeller exit. During this study, Laser Doppler Anemometer (LDA) measurements, revealed the presence of a jet-wake flow structure. The location and the extension of the wake seem to be affected by the proximity of the diffuser vanes.

Sinha et al. (1998) conducted PIV measurements in order to investigate the flow structure and turbulence in a centrifugal pump with a vaned diffuser and demonstrated the entire flow is dominated by a series of wakes, generated by the impeller, diffuser vanes the unsteady separation phenomena. They also demonstrate the impact of the impeller blade orientation on the flow structure.

Sinha et al. (2001) performed the experiments in a facility that enables flow visualization in the impeller between the diffuser vanes and in the volute. They observed high flow separation between impeller and diffuser vanes. When the gap of 20% of the impeller radius (15.4% of the diffuser chord length) which is larger than most of the available data for pumps. Especially at the below design conditions, there is a consistent high-speed leakage flow in the gap between the impeller and diffuser from the exit side to beginning of the volute.

Kurukawa and Amasaka (1983) studied the flow in a 2D volute and demonstrated that the static pressures and the velocities in the volute were not uniform circumferentially, that is, not axisymmetrical under off-design operating conditions. Miner et al. (1989) measured the flow inside a centrifugal impeller by using a laser velocimeter. They demonstrated that the flow was not uniform circumferentially, even at design point, because of the interaction between the impeller and the volute.

As for pressure fluctuations, Iino and Kasai (1985) measured the blade pressure fluctuations in a centrifugal pump and found that the fluctuations depended on flow rates and blade/vane angles. Arndt et al. (1989-1990) measured the pressure fluctuations on the blades of a 2D centrifugal impeller. They reported that the largest blade-pressure fluctuations occurred at the trailing edge, independently of the diffuser vane configuration, but that the magnitudes depended greatly on the vane number and vane angle. Tsukamoto et al. (1995) measured the pressure fluctuations inside the diffuser of a centrifugal pump and reported that the fluctuations on the suction side of the diffuser vane's leading edge were large.

When a dynamic study is proposed, difficulties go basically in two directions: geometric variables are quite complex and hence flow interactions have different nature (Brennen, 1994). Pressure fluctuations interact with the volute casing

or even with the circuit and give rise to dynamic effects (mainly unsteady forces) over the mechanical parts, which are one of the most important sources of vibration and hydraulic noise (Chu et al., 1995).

Investigation of fluid forces in turbomachinery has been a key research topic for a long time because of the increasing requirements for high reliability, low vibration, and low noise. The search includes a study of the impeller-diffuser interaction force, a typical fluid force that can cause not only serious vibration problems in shafts and pipelines but also failure of impellers and bearings. Many experimental studies have been done on the impeller-diffuser interaction in centrifugal pumps: flows and pressures inside volutes, diffusers, and impellers were studied by Arndt et al. (1989, 1990), Eisele et al. (1997), Iino and Kasai (1985), Kurokawa and Amasaka (1983), Uno et al. (1995), rotordynamic fluid forces acting on impellers were studied by Ohashi et al. (1998), Ohashi and Shoji (1984), Zhang et al. (2000), and vibrations of shafts and impellers were studied by Guo et al. (1999).

As far as radial turbomachines are concerned, pronounced impeller-diffuser interactions are expected, since the spacing between impeller and diffuser rows is normally quite small. In centrifugal pumps and compressors, many researchers observed that a highly distorted flow was discharged from an impeller exit (Eckardt (1976) and Krain (1988)). Also the pressure field in frontal part of a diffuser passage may be very in non-uniform condition. Both would result in a strong unsteady interaction between the blade rows.

Impeller-diffuser interaction may be divided into two different mechanisms: potential flow interaction and wake interaction. Because the radial gap between the impeller blade trailing edge and the diffuser vane leading edge was small (about 1.1% of the outlet diameter of the impeller), potential flow interaction was assumed to be predominant by Guo and Okamoto (2003).

In a diffuser pump, the centrifugal impeller interferes with its successive diffuser vanes and produces pressure fluctuations downstream of the impeller. In the case of a small radial gap between the impeller and diffuser vanes, the magnitude of these pressure fluctuations may become as large as the total pressure rise across the pump (Arndt et al. (1989, 1990), Tsukamoto et al. (1995) and Akhras et al. (2000)).

These fluctuations not only generate noise and vibration that cause unacceptable levels of stress and reduce component life due to fatigue, but also introduce unfavorable characteristics of pump performance even at or near the design point. Therefore, there is a need to understand the sources of unsteadiness (i.e., potential and wake interactions) to control the pressure fluctuations and to improve the overall pump performance and reliability. Dring et al. (1982) indicated two distinct mechanisms of impeller-diffuser interaction; wake interactions and potential interactions. Potential interactions are presently fully understood as potential flow effects induced by inviscid interaction due to the relative motion between impeller blades and diffuser vanes. Wake interaction is far more complicated than potential interaction. It originates from the impingement and convection of wakes shed from the impeller passages and moving through the successive diffuser passages.

Impeller-diffuser interactions have been studied extensively in turbomachines. However, only a few studies have focused on impeller-diffuser interaction in hydraulic machines, especially in centrifugal pumps. Particle Image Velocimetry has been used for the analysis of the flow velocities in the outlet part of the impeller and in the vaneless diffuser of a centrifugal pump. A lot of experimental and numerical results have been already published for that impeller. 2D measurements have been performed using PIV. The results are analyzed in various operating conditions by Wuibaut et al. (2001).

Some experimental contributions to wake interactions in pumps may be attributed to the extensive PIV measurements by Dong et al. (1992), and Akin et al. (1994), and hot wire measurement by Ubaldi et al. (1996). However, it is difficult to understand the flow phenomena due to impeller-diffuser interaction from only experimental studies because of the complicated flow structures in centrifugal pumps. A 2D unsteady flow calculation was presented by Fortes-Patella et al. (1995), who studied the interaction between the impeller and volute in a volute pump. Shi et al. (2001) numerically investigated the pressure fluctuations caused by impeller-diffuser interaction in a diffuser pump. Bert et al. (1996) carried out an unsteady flow calculation in a centrifugal pump using a Finite Element Method (FEM). Schulz et al.

(1996) did calculations of steady/unsteady 3D viscous flow in hydrodynamic torque converters.

These studies have given us a comprehensive understanding of the mechanisms of the pressure fluctuations induced by impeller-diffuser interactions. However, our understanding of the relationships among the volute pressures, the impeller pressures, and the radial fluid forces as a result of the integration of the impeller pressures is insufficient as yet. The pressure fluctuations and the radial fluid forces acting on the impeller, as well as the vibration of shaft in a centrifugal, vaned diffuser pump were measured simultaneously, and their relationship were investigated by Guo and Okamoto (2003). The experiment was performed for various diffuser vanes, flow rates, and rotating speeds. Furthermore, the rotating directions of pressure waves, the whirling directions of radial fluid forces, and the most predominant frequency components are discussed, and an equation is introduced for predicting them.

Investigations of the flow field in radial centrifugal pumps with simple blade curvature and constant impeller width for different operation points are performed by means of PIV and CFD. The aim is to validate a CFD tool for the efficiency optimization of pumps. Heilmann et al. (2002) have shown that a relatively good agreement between CFD and PIV for the radial sections in the impeller at the three different operating points.

2.1.2 Numerical Studies

The design of turbomachines presents one of the most advanced technologies in the world. The products are numerous and include challenging machines such as turbochargers, gas turbine engine and hydraulic machines. The successful development of efficient turbomachines requires an understanding of flow conditions, which are exceedingly complex. Appreciation of this factor, together with a concerted effort to harness the flow efficiently, has offered one of the most exacting problems faced by any design engineer. In attempting to control the design parameters involved, many of which are as yet little understood, the design engineer

requires both sophisticated software (for the prediction of flow processes) and hardware (for testing of components). Fierce competition to produce more efficient and cheaper turbomachines has led to large technical advances in their major components and, in particular, the compressor and turbine. The cost of manufacturing and testing these components in the development of new engines and the improvement of existing units is one of the largest problems facing manufacturers in today's market. To reduce the required manpower and overall cost and to speed up the development, an alternative solution has become essential. The ever-increasing power of computers and new advances in computational techniques is likely to provide the current answer to the problem. The very complex flow in turbomachines results from; the development of the boundary layer on the end walls and blade surfaces, centrifuging of the boundary layers, tip leakage flows and their interaction with the end wall and blade, boundary layers and strong secondary flows.

A clear understanding of this is necessary for a researcher to assess the performance of a blade configuration and to optimize the blade profile so as to minimize losses. The complete simulation of any flow can be achieved by application of the Navier-Stokes Equations (NSEs). Over the past few years, great advances have been made in CFD. However, the analysis as applied to a pump has not yet developed to a level permitting design solely on the basis of computational results. Experimental data for the internal flow are needed to verify computational results. Velocity magnitudes over a wide range of geometries are needed to understand the flow mechanics and to verify CFD predictions.

CFD has proved to be a very useful tool in the analysis of these turbomachines, both in design and performance prediction. Many research has been done: Denton and Dawes (1999), Croba and Kueny (1996) and Valenti (1996) give list of selected papers up to that date; Denus and Göde (1999) give a more extended and up to date bibliography, but no so specific. However, due to the difficulties of the task, most of the studies begin doing heavy simplifications of the problem either in the geometry or in the flow characteristics. Research slowly trends towards more the complete simulations.

Numerical simulation of centrifugal pumps is not easy due to the usual CFD difficulties; turbulence, separation, boundary layer, etc. Although there are also specific problems; complex geometry: a great number of cells are needed and, due to skewness, usually unstructured grids give better convergence than structured ones. Energy transfer is generated mainly by the centrifugal force in the impeller. A cascade simulation is not valid and these force source terms must be included in the equations of the moving zone.

The interaction between impeller and volute requires an unsteady solution processes to calculate the time derivatives of the equations. Also the blade position respect the volute the tongue is to be taken into account. This can be partially accomplished in a quasi-steady way: recalculating the solution with different grid positions; it is desirable that the code could do an unsteady calculation at the same time that slides the impeller grid each time step.

In order to understand the flow pattern through on impeller of centrifugal pump for the 3D turbulent flow under design and off-design operating conditions is numerically investigated by Shuliang et al. (1999). The SIMPLEC algorithm with body-fitted coordination and staggering grid system is used for the solution procedure of the discretized equations based on the NSEs and the standard $k-\varepsilon$ model. The calculated result for pressure distribution is compared with experimental data, which is measured on the spring casing in suction side of the pump. Good agreement is shown between the calculated result and experimental one.

Volute very frequently has profound effect on the impeller performance. It can cause the impeller to work very inefficiently and cause large vibrations if it is designed improperly. Although advanced CFD methods have been used extensively in impeller design, the volute has been neglected or considered independently. A FVM computational method is used to solve the 3D Reynolds-Averaged flow field of an impeller in the volute by Chen and Liaw et al. (1997). The study is aimed at the understanding of the effect of volute on the performance of impeller and its associated with unsteady effect. As its configuration and solution procedure of the physical problem is more complicated than a traditional single blade passage analysis, the grid generation and solution procedure will be described in more detail.

A traditional single blade passage without volute and at steady state condition is calculated first. Later, volute is added and calculated results are compared with the single-passage calculation. It is found that the predicted mean performance of the impeller and volute agrees better with the test data, while the fluctuation of the flow is significant, especially while the trailing edges of impeller is passing the volute tongue.

Differential numerical methods for the treatment of the RANS equations have been widely developed. An extensive list of applications where the numerical solution of the flow has proven its worth can be found in the work of Gunzburger et al. (1993). Although predictions on the flow field are always valuable, the numerical study in a pump is not easy due to the usual CFD difficulties: turbulence modeling, flow separation, boundary layer, etc (Lakshminarayana, 1991). There are also some particular problems like the extremely difficult geometry and complex flow dynamic effects. The numerical solutions of the dynamic effects on the centrifugal pumps can be seen in the studies by Tsukamoto et al. (1995) and Shi et al. (2001). Gonzalez et al. (2002) studied a combination of experimental measurements and numerical modelization of the flow using a RANS commercial code. The focus of the study is the dynamic response inside the volute of a centrifugal pump and the flow interactions at different axial positions at the impeller exit.

Both the passage-averaged and RANS approaches still suffer from uncertainties associated with the validity of the turbulence closures employed. As is well known, the main difficulty is that even the largest eddies, which do not have a universal structure, have to be modeled (Speziale, 1991). For example, large-scale structures in a hub vortex are very different from those in the wake, or in the tip-leakage vortex. Thus, it is necessary to adjust the model on a case-by-case basis, which makes it impractical for truly predictive purposes. These difficulties have motivated the development of a variety of RANS models with increasing complexity. Overall, they have improved, but have not led to breakthroughs in our ability to predict these flows. Therefore, for the purpose of providing fundamentally more accurate simulations of turbulent flow in turbomachines, it is of interest to consider Large-Eddy-Simulation (LES) and the modeling issues associated with it. For RANS

of multistage turbomachinery, it is very difficult to define unique phase-relationships among stages, which requires additional averaging (Adkins and Smith (1982), Gallimore and Cumpsty (1986), Rai, (1987) and Lakshminarayana, (1991)). This issue is the main ingredient of the passage-averaged approach introduced by Adamczyk (1985) and described in more detail by Adamczyk et al. (1990) and Rhie et al. (1995).

Cao et al. (1999) investigated the 3-D turbulent flow through the impeller at off design operating condition by numerical flow analysis, in order to understand the flow patterns through a semi-open impeller of centrifugal pump for solid-liquid two-phase flow. They used the solution procedure of the discretized equation based on the NS equations and the standard $k-\varepsilon$ model. The comparison between the numerical and experimental results has showed that this kind of calculation can be used for performance prediction and optimization of the impeller geometry.

There have been some experimental and numerical studies of the unsteady blade row interaction problems in centrifugal machines at a high-speed flow condition (Yamane and Nagashima (1997) and Filipenco et al. (1998)). The simulations have been performed as incompressible, 2D and 3D, unsteady and with sliding mesh in the impeller, with the $k-\varepsilon$ turbulence model by Blanco-Marigorta et al. (2000a, 2000b). Most of research experiments are operated in a low speed flow condition because of limitation in costs and difficulties in instrumentation at high speeds (Inoue and Cumpsty (1984) and Ubaldi et al. (1996)). For pumps in which the working fluid is essentially incompressible, it was suggested that the amplitude of the pressure fluctuations due to the unsteady blade row interaction may have the same order of magnitude as the total pressure rise across the pump (Arndt et al. (1990)). This would affect both the fluid dynamic and the structural performances, especially in relation to the minimum unsteady pressure associated cavitations. Blade row interaction is also responsible for the noise generation in the pumping system (Akin and Rockwell (1994)).

Fisher (1981) compared with theoretical predictions of the diffuser field for diffuser geometries by experimental flow data recorded on a low speed centrifugal compressors rig. Studies with a vaneless diffuser confirmed that, although the

absence of compressibility effects probably produces minor differences between the rig and high speed machines, the principal high speed features of jet-wake flow are adequately reproduced. Data recorded on the rig for a range of different vaned diffusers plays a major role in establishing the circumferential variation in mean flow conditions. They presented the experimental results for the diffuser geometries. These results show large fluctuations in the vicinity of the blade leading edge, which maybe explained by consideration of the flow round the leading edge of an aerofoil.

Centrifugal pump is based on the rotating motion of the impeller relative to the stationary pump casing. In the essence it working is simple. The fluid enters the pump in axial direction through the suction nozzle into eye of the impeller or rotor. It is subsequently accelerated into a circular motion by the rotating impeller blades and collected in the volute or diffuser. The fluid finally leaves the pump through the discharge nozzle. The inlet of the pump as well as the volute (or diffuser region) is part of the pump casing. The impeller contains several blades or vanes which are fitted to hub (or back shroud) of the impeller. The impeller may be equipped with a (front) shroud, or constructed as an open impeller. The discharge region of the pump collects the fluid as it leaves the impeller. The shape of this region is such that the high velocity of the fluid is partly converted to static pressure by gradual expansion. Starting from the tongue of volute, the channel through-flow area gradually increases towards the throat of the discharge nozzle to account for the increasing amount of fluid. For volute casings the major part of the conversion from the kinetic energy to pressure occurs in the discharge nozzle. In a with diffuser pump, the impeller discharges into a channel equipped with stationary vanes. The conversion of energy predominantly takes place between the diffuser vanes (BPM Van Esch (1999)).

A centrifugal pump is a rotating machine in which flow and in which flow and pressure are generated dynamically. The inlet is not walled off from the outlet as is the case with positive displacement pumps, whether they are reciprocating or rotary in configuration. Rather, a centrifugal pump delivers useful energy to the fluid or “pumpage” largely through velocity changes that occur as this fluid flows through the impeller and the associated fixed passageways of the pump; that is, a “rotodynamic” pump. All impeller pumps are rotodynamic, including those with

radial-flow, mixed-flow and axial flow impellers: the term “centrifugal pump” tends to encompass all rotodynamic pumps.

Pumps are used in varied applications and are integral to many industries. Yet, in spite of their prevalence and relatively simple configurations compared to other turbomachines, designing an efficient and durable pump remains a challenge. Two effects that challenge a designer are the force loads on the impeller and the hydraulic efficiency. Small changes in the volute design have been shown to affect one or both of these parameters significantly (Baun (2000)). If the efficiency is too low, the energy consumption over the life of the pump can cost an industry a significant amount of money (Hergt (1999) and Baun and Flack (2003)).

The impeller is an important part of a pump. For improving pump feature and efficiency, it is of great importance to carry out investigation on the flow behavior inside impeller. With the currently increasing computer capability and computation fluid dynamics developed, numerical simulation of the 3-D turbulent flow through impeller is becoming possible (Quin et al. (1994)).

CFD analysis is being increasingly applied in the design of centrifugal pumps. With the aid of the CFD approach, the complex internal flows in water pump impellers, which are not fully understood yet, can be well predicted, to speed up the pump design procedure. Thus, CFD is an important tool for pump designers.

A numerical model is developed for calculating the 2D, unsteady, and incompressible and turbulent flow within the rotating impeller and stationary volute of an industrial centrifugal pump. The objective is the investigation and comprehension of the instantaneous behavior of centrifugal pumps, aiming at the reduction of vibrations, radial forces and hydraulic noise. The computation is performed within a blade-to-blade stream tube for the impeller and a tube normal to the axis of rotation for the volute. The equations to be solved are the unsteady RANS equations along with the continuity equation and the unsteady $k-\varepsilon$ equations for turbulence modeling. The FVM is applied for space discretization and an implicit scheme for time discretization. A multidomain overlapping grid technique is used for matching together the relative flow field calculated within the rotating impeller and the absolute one calculated within the stationary volute. In this way the impeller and

volute interaction is directly taken into account. The numerical model is validated for a centrifugal pump of under design flow conditions. Comparisons between calculation and measurements show fairly good agreement (Croba and Kucny (1996)).

Kaupert et al. (1996), Potts and Newton (1998), and Sun and Tsukamoto (2001) studied pump off-design performance using the commercial CFD codes. Although these researchers predicted reverse flow in the impeller shroud region at small flow rates numerically, some contradictions still existed. For example, Kaupert's experiments showed the simultaneous appearance of shroud-side reverse flow at the impeller inlet and outlet, but his CFD results failed to predict the numerical outlet-reverse flow. Sun and Tsukamoto (2001) validated the predicted results of the head-flow curves, diffuser inlet pressure distribution, and impeller radial forces by revealing the experimental data over the entire flow range, and predicted back flow at small flow rates, but they did not show an exact back-flow pattern along the impeller outlet.

Goto (1992) presented a comparison between the measured and computed exit-flow fields of a mixed flow impeller with various tip clearances, including the shrouded and unshrouded impellers, and confirmed the applicability of the incompressible version of the 3D Navier-Stokes code developed by Dawes (1986) for a mixed-flow centrifugal pump.

Qin and Tsukamoto (1997) calculated an unsteady flow caused by impeller-diffuser interaction in a diffuser pump. They used RANS code to calculate the pressure fluctuations downstream of the diffuser pump impeller. Full NSEs with $k-\varepsilon$ turbulence model were solved within a diffuser pump stage and the results were compared with experimental data to validate the CFD calculations. The results were also compared with values predicted. These comparisons lead to identify the contributions to pressure fluctuation resulting from various sources (potential and wake diffuser interactions), and will enhance our understanding of the mechanism of impeller-diffuser interaction in diffuser pumps.

Many CFD studies concerning the complex flow in all types of centrifugal pumps have been reported. Oh and Ro (2000) used a compressible time marching

method, a traditional SIMPLE method, and a commercial CFD program to simulate flow pattern through a water pump and compared the differences among these methods in predicting the pump's performance.

The effects of the impeller-diffuser blade row interaction on the performance of radial turbine stages are investigated numerically by Sato and He (2001). A three-dimensional unsteady incompressible Navier-Stokes method based on the dual-time stepping and the pseudo compressibility method is developed for the performance prediction. A centrifugal pump with a vaned diffuser is calculated for validation purposes, and the predicted unsteady flow results show reasonable agreement with the experimental data. The method is applied to analysis of hydraulic turbine stages. A generic turbine rotor is combined with a row of nozzle guide vanes with three settings of radial gap and numerical flow simulations are conducted for the performance evaluations. The predicted efficiency of the hydraulic turbine stages deteriorates if the radial gap between blade rows is reduced although the difference is very small.

From such literature, it was found that most previous research, especially research based on numerical approaches, had focused on the design or near-design state of pumps. Few efforts were made to study the off-design performance of pumps. Centrifugal pumps are widely used in many applications, so the pump system may be required to operate over a wide flow range in some special applications. Thus, knowledge about off-design pump performance is a necessity. On the other hand, it was found that few researches had compared flow and pressure fields among different types of pumps. Therefore, there is still a lot of work to be done in these fields.

The physical mechanisms of impeller-diffuser interactions in turbomachines are yet not completely understood, for supersonic and transonic flows, and even for incompressible flows. There is a direct relation between the resulting flow unsteadiness and the rotor-stator clearance according to blade loading. For weak interactions, time averaged characteristics of machine are not modified: the machine works in quasi static operating conditions. For strong unsteadiness, the time averaged

characteristics are notably modified and rotor-stator interactions can change the level of losses and energy transfer. Noise and performance drop often occurs, but sometimes increase of performances can be observed. This improvement is due to delay of the stall separation on the blades. One can also expect, by better understanding of physical mechanisms of the rotor-stator interactions, to reduce noise by techniques like active control. This entire works contributed to the understanding of impeller-diffuser interaction in pumps; however, it has not been reached the stage in which the unsteady flow due to the impeller-diffuser interaction can be predicted with sufficient accuracy.

2.2 Outline of Present Study

For the present study, FLUENT computational solver was used in order to understand the complex unsteady flow phenomena in the pump. FLUENT is a software package that can predict laminar flow, turbulent flow, and heat transfer. It has been widely used in the field of turbomachinery, and the simulation results have been proven by many researches to be reliable. A 3D relative flow analysis between the impeller and diffusers was modeled in a rotating frame of reference using a CFD this software package. In this package program, the mass conservation and momentum equations were solved numerically using a FVM-based algorithm. The internal flows inside the centrifugal pump at design and off-design point conditions are analyzed by solving the fully 3D NSEs coupled with the Renormalization Group Theory (RNG) $k - \varepsilon$ turbulence model.

Many scientists frequently use a PIV measuring technique in a pump. In this work, experimental data on the centrifugal pump are used. For validation of this CFD study, calculated numerical results are compared with the experimental results. Experimental results are obtained by Sinha (1999). Then, the numerical study continues for different gaps.

The purpose of the present study is to show a numerical study impeller-diffuser interaction considering the 3D steady and unsteady flow in various radial

gap geometries (20%, 15%, and 10% of the impeller radius). In the present study, radial gap means the radial clearance between impeller outlet diameter and diffuser outlet diameter (see figure 3.13a). The present work intends to show how impeller-diffuser interaction and its associated flow structures relate to characteristics like peak efficiency, stability and noise production of a pump, by means of the experimental data and CFD for the flow behavior in the impeller-diffuser interaction zone. The ultimate goal of this work is to show how steady and unsteady impeller-diffuser interaction flow features can be integrated in the design procedure of a pump.

2.2.1 Outline of Dissertation

In this thesis, introduction part introduces general information on the flow structure in centrifugal pumps, and CFD importance of the present work.

Previous study parts concerning the numerical and experimental studies on the flow in the centrifugal pumps are given in section 2.

Material and method part of present studies is divided into three parts:

i) The first part reports the details of the experimental studies performed by Sinha (1999).

ii) The second part describes the produced pump models and the grid generation.

iii) Detailed information on numerical method and solution technique provided by CFD package program (FLUENT) is reported in third part.

In results and discussion parts, experimental and numerical results are compared and discussed in details. Later, numerical results are given for various different gap geometries and parameters in the same pump model.

Finally, conclusion and recommendations part summarize the main finding of the present work and provides recommendation for further in investigations as a future work.

3. MATERIAL AND METHODS

Experimental fluid dynamics has played an important role in validating and defining the limits of the various approximations to the governing equations. The wind tunnel, for example, as a piece of experimental equipment, provides an effective means of simulating real flows. Traditionally this has provided a cost effective alternative to full-scale measurement. However, in the design of equipment that depends critically on the flow behavior, for example the aerodynamic design of an aircraft, full-scale measurement, as part of the design process is economically impractical. This situation has led to an increasing interest in the development of the studies.

Computational Fluid Dynamics (CFD) has proven capability in predicting the detailed flow behavior for wide-ranging engineering applications, leading typically to an improved equipment or process design. CFD is used for early conceptual studies of new designs, for detailed product development, for scale-up, for troubleshooting and for system retrofitting.

Pump manufacturers are, now using CFD a simulation tool originally developed for academic research and aerospace. CFD is used to solve the RANS Equations that describe fluid flow and to generate a 3D model of that flow, making possible more efficient pumps and minimizing cavitation, which causes turbulence that wears out the impeller and blades.

Yet to be achieved is a realistic model of turbulence, expected to take decades of fundamental work. CFD must also be able to model the transition of laminar to turbulent flow so it can be obtained that quantitative agreement between CFD and laboratory data for many critical turbomachinery problems.

The object of CFD is to use computers to solve the previously intractable conservation equations for fluids in order to accurately simulate flows. This typically involves discretizing the problem in a finite set of elements, applying the conservation of mass, momentum, energy, and chemical species (where necessary) to these elements, placing additional boundary conditions at the edges of the computational grids, and solving the resultant algebraic equations in a iterative

fashion. While the ideal is for CFD simulations that do not require actual scale-model experiments, in fact, some scale-model experiments are generally still necessary in order to validate the accuracy of the CFD code. Still, the number of necessary experiments can be greatly reduced as computations fill gaps between experiments and numerical models can simulate situations that would be impossible to set-up experimentally.

Thus, CFD allows the analysis of fluid flow problems in detail, faster and earlier in the design cycle than possible with experiments, costing less money and lowering the risks involved in the design process. This trend is only likely to grow more pronounced in the future as computers become increasingly cheaper and more powerful while traditional forms of testing become increasingly expensive. For improving pump feature and efficiency, it is great importance to carry out investigation on the flow behavior indeed. With the currently increasing computer capability and CFD developed, numerical simulation of the 3D turbulent flow through pump is becoming possible.

The purpose of the present study is to show a numerical study of a centrifugal pump taking into account the whole 3D geometry, the effect on the flow inside the pump and the impeller-diffuser interaction of the radial gap. It has been done with the commercial software package FLUENT v 6.0. This code uses the FVM and solves 3D NSEs with ability to handle unstructured grids, include relative reference frames and unsteady calculations.

3.1 Pump Geometry and Modeling

Pumps occur in a wide variety of configurations and they add to energy to the fluid. The dynamic interaction between a fluid and a solid is often based on flow and fluid/solid interaction forces. Centrifugal pumps are classified as axial-flow, radial flow, or mixed flow machines depending on predominant direction of the fluid motion relative to the impeller's axis as the fluid passes the blades. One of the most

common radial flow turbomachines is the centrifugal pump. This type of pump has two main components: an impeller attached to a rotating shaft, and a stationary volute (and diffuser) enclosing the impeller. The impeller consists of a number of blades (usually curved) arranged in a regular pattern around the shaft. As the impeller rotates, fluid is sucked in through the eye of casing and flows radially outward. Energy is added to the fluid by the rotating blades, and both pressure and absolute velocity are increased as the fluid flows from the eye to the periphery of the blades. For the simplest type of centrifugal pump, the fluid discharges directly into a volute-shaped casing. The casing shape is designed to reduce the velocity as the fluid leaves the impeller, and this decrease in kinetic energy is converted into an increase in pressure. The volute shape casing, with its increasing area in the direction of flow, is used to produce an essentially uniform velocity distribution as the fluid moves around the casing into the discharge opening. For large centrifugal pumps, it is used the diffuser vanes which surround the impeller. The diffuser vanes decelerate the flow as the fluid is directed into the pump casing. This type of centrifugal pump is referred to as a diffuser pump.

Centrifugal pump design is a highly developed field, with much known about pump theory and design procedures. However, due to the general complexity of the flow through a centrifugal pump, the actual performance of the pump cannot be accurately predicted on a completely theoretical basis because of losses. Actual pump performance is determined experimentally through tests on the pump. From these tests, pump characteristics are determined and presented as pump performance curves. It is this information that is most helpful to the designer responsible for incorporating pumps into a given flow system. The overall pump efficiency is affected by the hydraulic losses in the pump, and in addition, by the mechanical losses in the bearings and seals. There may also be some power loss due to leakage of the fluid between the back surface of the impeller hub plate and casing, or through other pump components. This leakage contribution to the overall efficiency arises from the sources, the hydraulic efficiency, mechanical efficiency, and volumetric efficiency.

There will be pump noise and vibration, pitting damage to the impeller, and a sharp drop off in pump head and discharge. In some liquids this deterioration starts before actual boiling, as dissolved gases and light hydrocarbons are liberated. This is sharp drop off and is indicative of non-recoverable losses of the three types: impeller recirculation loss significantly only at low flow rates, friction losses on the blade and passage surfaces, which increase monotonically with the flow rate and “shock” loss due to mismatch between the blade angles and the inlet flow direction, especially significant at high flow rates.

The pump consists of an impeller rotating within a casing. Fluid enters axially through the eye of casing, is caught up in the impeller blades, and is whirled tangentially and radially outward until it leaves through all circumferential parts of the impeller into the diffuser part of the casing. The diffuser, or scroll, section of the casing decelerates the flow and further increases the pressure.

There are complicated 3D flow effects and hence are difficult to predict. Although, as mentioned, CFD techniques are becoming more important, modern performance prediction is still a blend of experience, empirical correlations, idealized theory, and CFD modifications.

The formulas from pump theory are merely definitions of performance parameters and cannot be used in ant predictive mode. To actually predict the head, power, efficiency, and flow rate of a pump, two theoretical approaches are possible: (1) simple one-dimensional flow formulas and (2) complex digital computer models, which account for viscosity and three-dimensionality. Many of the best design improvements still come from testing and experience, and pump research remains a very active field. The last 10 years have seen considerable advances in CFD.

In this study, the geometrical data of impeller, diffuser and volute and operational conditions of the pump given in Table 3.1 have been used as a initial data for numerical analysis. The work has been carried out at different vaneless space ratio in order to look into the effect of the impeller and diffuser interactions on the pump performance.

As seen in figure 3.1, the impeller has five backward swept blades with a logarithmic profile, each with an exit angle of 19° relative to the tangent. At the

inlet, the impeller blade inlet angle at the hub relative to the circumferential tangent is 14.1° , while the blade inlet angle at the shroud surface is about -4.2° . The inlet and discharge (blade tip) diameters of the impeller are 8.51 cm and 20.32 cm, respectively. The diffuser with 9 vanes has an inside diameter of 24.45 cm and a discharge diameter of 30.5 cm. The circular arc vanes have a chord length of 13.44 cm with the convex side radius of 141.5 mm and a concave side radius of 24.79 cm. The solidity ratio, i.e. the ratio between the diffuser passage length and the average passage width, is 3.08. The inlet and exit angle that the mean camber line makes with the tangential direction is 4.42° and 17.38° respectively. The volute with a constant axial width of 1.27 cm is transparent at the bottom and along the periphery. The setup allows illumination and flow visualization in both the horizontal (parallel to the shrouds) and vertical planes. The configuration of the external contour (perimeter) of the volute is a logarithmic spiral with the radius in cm given by;

$$r_v = 17.78 + 24.46(\theta / 360) \quad (3.1)$$

where θ is the angle in degrees (between 0° and 360°). In the present configuration, (the tongue section is removable) the tip of the tongue is located at $\theta = 29^\circ$ (see figure 3.2) Beyond $\theta = 360^\circ$ the outer perimeter is straight. The rectangular 31.06 cm^2 (24.46×1.27) exit area is gradually expanded up to a circular area of 81.07 cm^2 (diameter 10.16 cm, area ratio = 2.61) by imposing a maximum expansion angle of less than 7° to avoid separation. The pump has radial gap 20% of impeller radius and 15.4% of the diffuser chord length. In this the numerical study, gap states 20%, 15% and 10% of impeller radius are investigated, and these radial gap geometries have been defined Case I, Case II, Case III, respectively. A centrifugal pump with a vaned diffuser has been tested during the study. The centrifugal pump test model with a vaned diffuser operates at 890 rpm and powers of 3.7 HP. The pump is totally suspended and isolated by air cushions and two coiled flexible hoses from the rest of the pump loop. The design specific speed of the pump, which is in the characteristic range for radial turbomachines, is 0.49 at the estimated design flow rate of 4.12 l/s

and a head 31.9 kPa. The experimental study which contained the static pressure measurements and PIV for performance and efficiency curves in a pump flow visualization facility, is made by PIV technique at Johns Hopkins University in United States by Sinha (1999). A transparent centrifugal pump test rig, which allows flow visualization, is illustrated schematically in figure 3.3.

Table 3.1 The pump dimensions and operating conditions (Sinha, 1999)

<i>Impeller</i>	
<i>Inlet / Discharge diameter</i>	<i>85.1/203.2 mm</i>
<i>Number of blades</i>	<i>5 backswept</i>
<i>Inlet / Exit angle of blade</i>	<i>14.1°/19°</i>
<i>Diffuser</i>	
<i>Number of vanes</i>	<i>9</i>
<i>Inside/ Discharge diameter</i>	<i>244.5/305 mm</i>
<i>Inlet / Outlet angle of vane</i>	<i>10.6°/10.95°</i>
<i>Chord length</i>	<i>134.4 mm</i>
<i>Suction side radius</i>	<i>141.5 mm</i>
<i>Pressure side radius</i>	<i>247.9 mm</i>
<i>Axial width</i>	<i>12.7 mm</i>
<i>Operating conditions</i>	
<i>Impeller speed</i>	<i>890 rpm (93.15 rad/s)</i>
<i>Design flow rate</i>	<i>5.67 l/s</i>

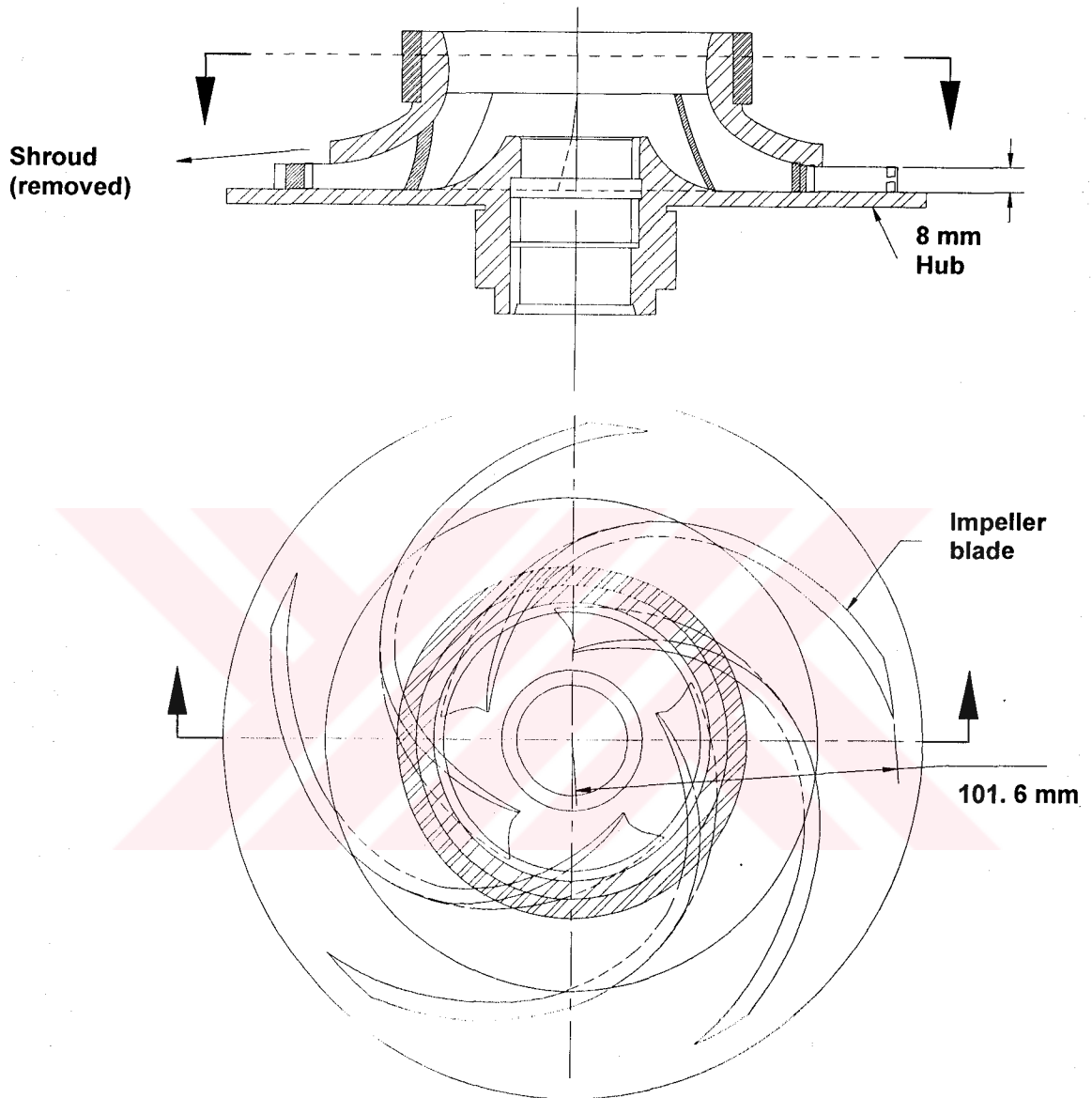


Figure 3.1 Machine drawing of the impeller (cut-section and bottom view). Note in the cut-section above the stainless steel shroud was removed and replaced with an acrylic one (the direction of rotation for the figure is counter-clockwise) (Sinha, 1999)

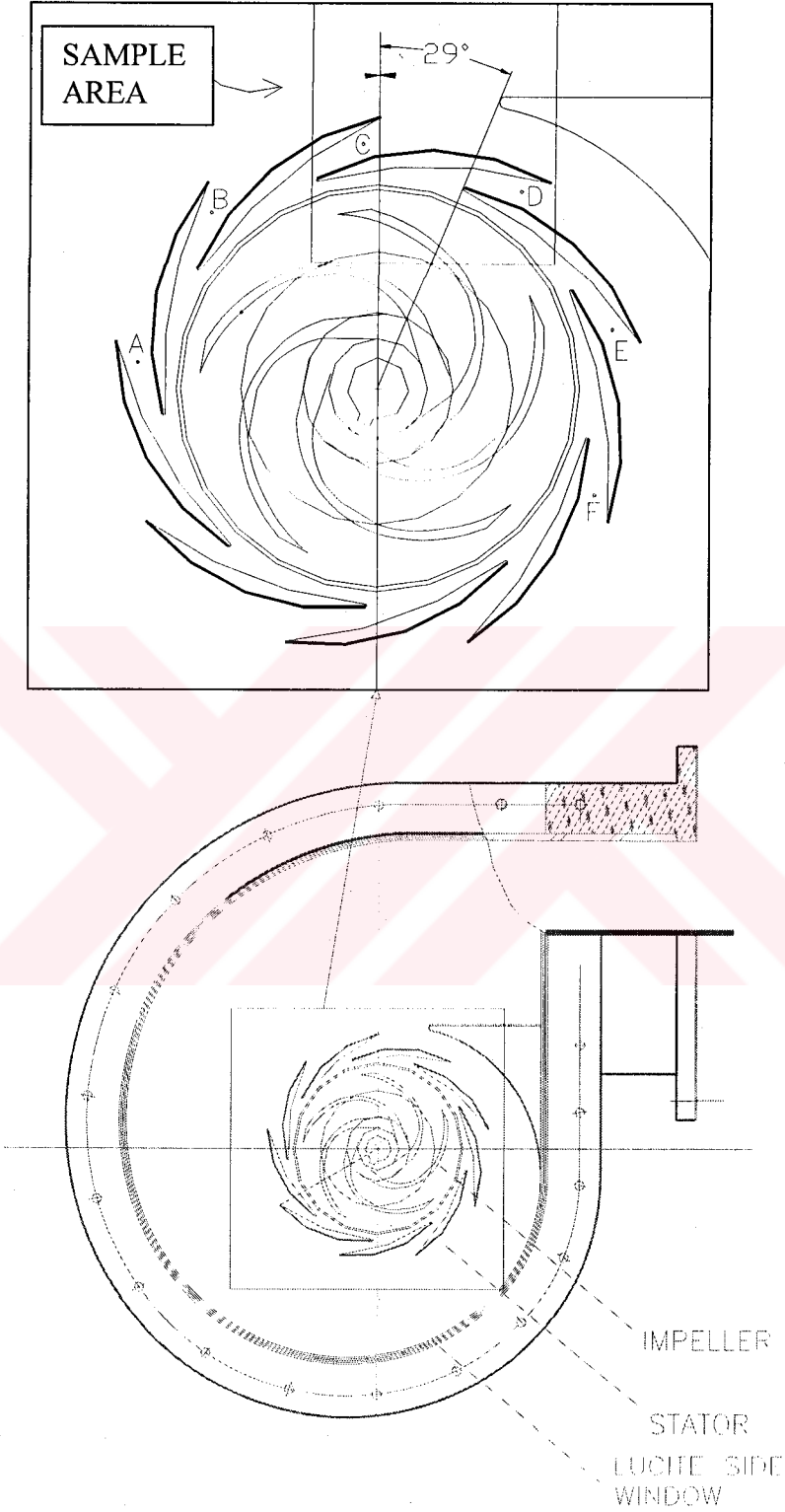


Figure 3.2 The pump geometry and location of the PIV measurements (top view) (Sinha, 1999)

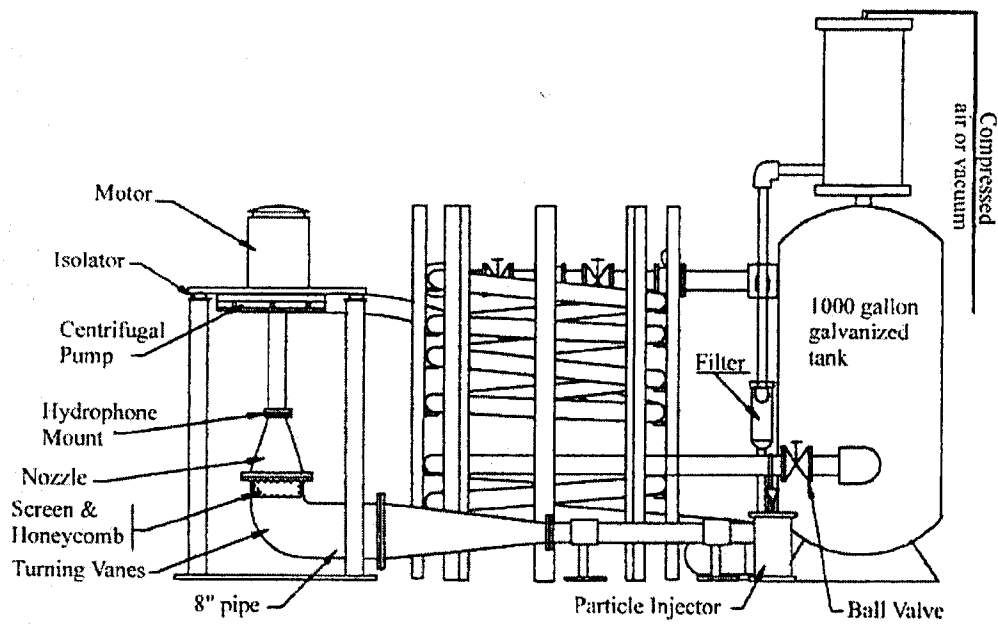


Figure 3.3 A schematic description of the pump loop (Sinha, 1999)

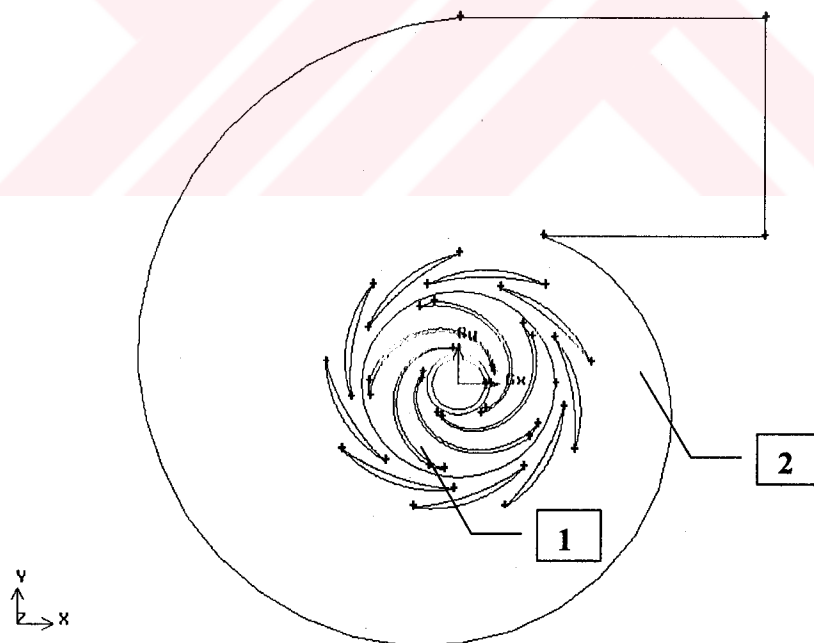


Figure 3.4 The pump model and fluid zones (2D) (1-Impeller, 2-Volute)

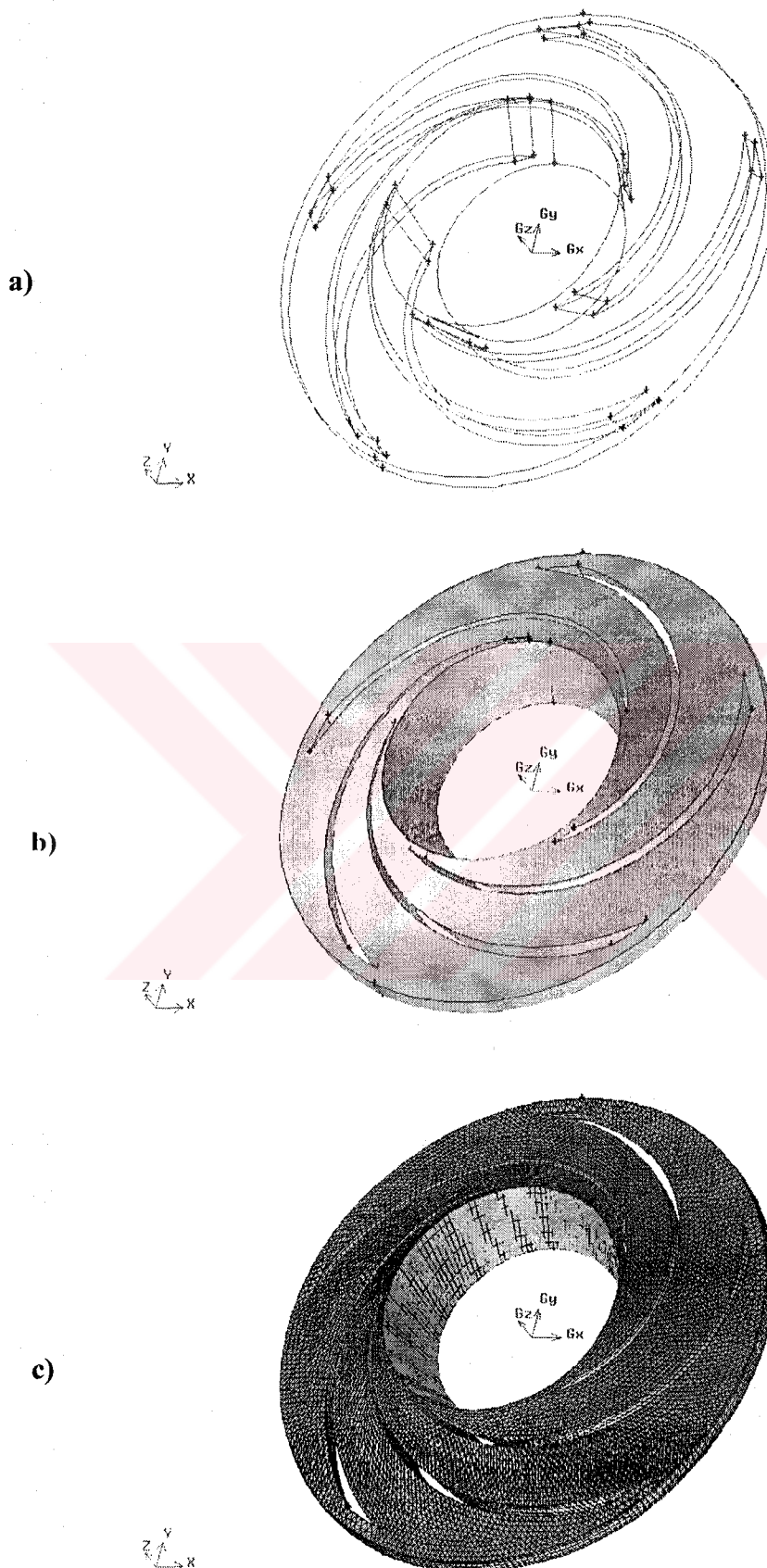


Figure 3.5 a) Wire model, b) Solid model, and c) Mesh model of the impeller (3D)

After computation of the pump modeling, firstly, the 2D volute, the diffuser vanes, the impeller and its blades were produced respectively. Equation 3.1 is used for modeling of volute. Dimensions, geometric arrangements of the pump are given in table 3.1. The fluid is divided into two parts the rotating fluid zone (1) and stationary fluid zone (2) as shown in Figure 3.4. This is necessity for defining of the fluid properties and meshing.

The half of the impeller blade is twisted, the other half continues radially. The leading edge and trailing edge of blade are 31 mm and 8 mm high, respectively (figure 3.5). The pump with this blade is known the mixed flow pump.

To produce 3D models, the 2D volute face is extrudes 12.7 mm along the z-axis and 3D volute is created (figure 3.6). While producing the pump models, diffuser vanes are removed from the volute and the blades are removed from the impeller. The hub, the shroud and other walls only surround the fluid and these parts are used to define boundary conditions. Because of that they are not into consideration in calculations. Since, the purpose is to investigate the fluid flow in the pump in fact in the models; only fluid regions are remained. Because, the purpose is investigate the fluid flow in the pump. Usually, in CFD studies this method is used. But, if thermal and mechanics interactions are taken into consideration, the solid region is also must be modeled. A geometrical discretization of the centrifugal pump is made for the numeric treatment. The fluid inside the pump is divided into four parts, because it is needed to be defined for fluid properties and grid generation. These fluid regions are the inlet (1), the transition region (inlet-impeller) (2), the impeller (3) and the volute (4) as shown in figure 3.6.

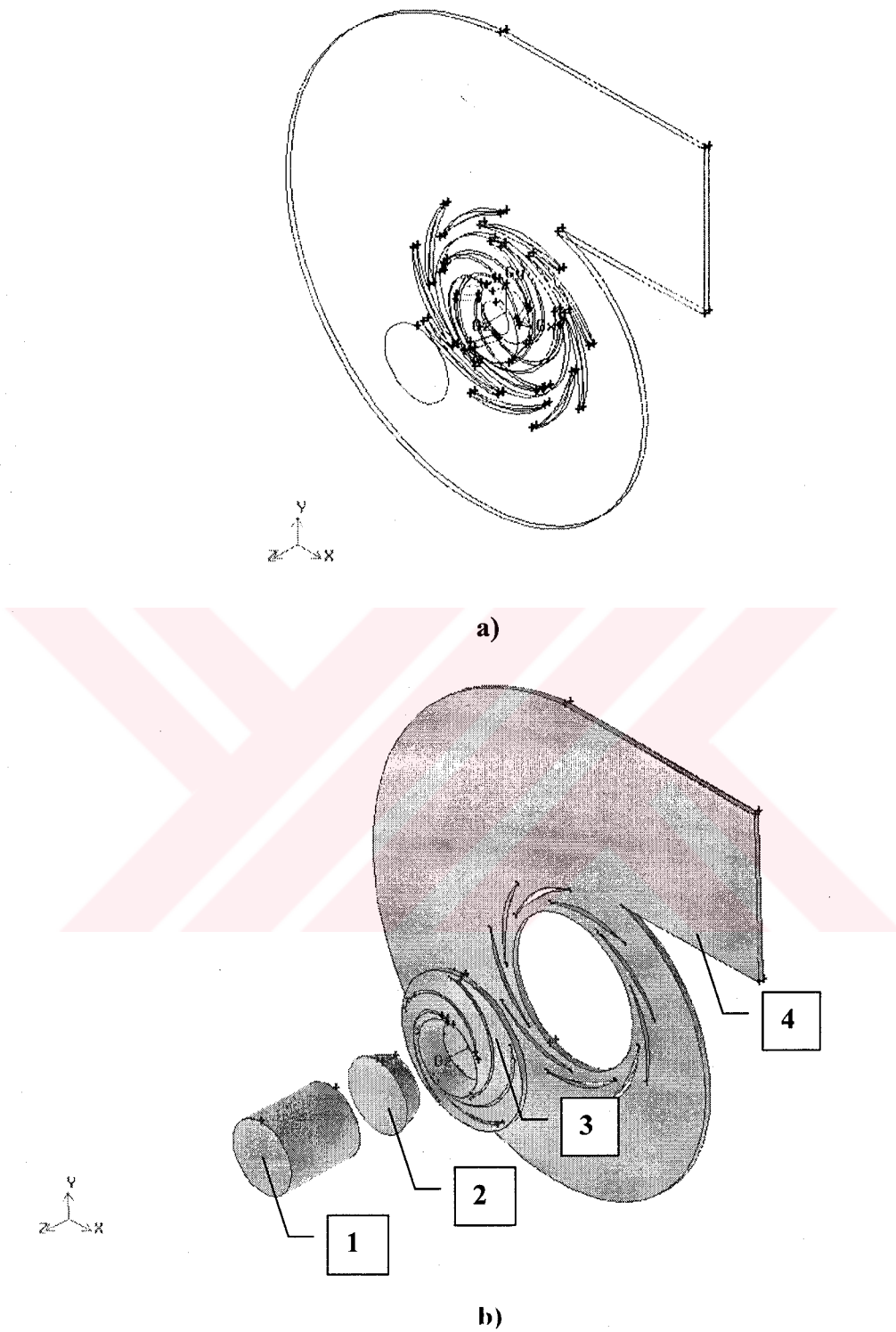


Figure 3.6 The 3D Pump Model, a) wire model b) solid model and fluid zones (fluid zones; 1-inlet, 2-transition region, 3-impeller, 4-volute)

3.2 Experimental Results Used in This Study

The flows within turbomachines are particularly challenging due to their complexity. They involve unsteady wakes, flow separation, high straining, interactions between blade rows etc. Consequently, modeling these flows remains a challenge. The Particle Image Velocimetry (PIV) performed in the diffuser vaned centrifugal pump provides quantitative information on the dynamics of wake interaction with downstream blades.

In the present study, experimental work has not been performed. For the purpose of comparison experimental results which were performed at Johns Hopkins University in United States by Sinha (1999) is used in the present study. For this reason it was decided to report experimental procedure of Pinarbasi (2000, 2001).

Two static pressure taps are located about 1m upstream of the inlet and 1m downstream from the exit of the pump to measure the pressure rise across the pump (figure 3.3). Piezoelectric transducers are used for the pressure measurements. These measurements, which are made at varying flow rates, are used to plot the performance curve for the centrifugal pump.

The technique of PIV is used to measure velocity vectors in a plane. This particle imaging method in fluid flows is now well established (Adrian (1991)). In this method, a section of the flow is illuminated with a thin laser light sheet and seeded with fluorescent particles that track the flow. The particles, mixed uniformly by particle injector installed far upstream of the pump inlet, emit yellow light in the green laser light. The laser sheet is pulsed twice and both sets of the images of particles are recorded either on a single frame or on separate photographic frames. Since the time delay between exposures is known, the distance between the particle pairs can be converted into velocity. The imaging methodology, equipment details and the application of a novel method called digital image shifting are described in the following sub-sections. The optical setup used in to generate the light sheet the pump facility is shown in figure 3.7. An Nd-Yag laser with 532 nm wavelength emissions is used for the application.

The pump facility has transparent window at the bottom and along the periphery and the laser sheet may be created in either the horizontal or vertical planes depending on the research interest.

In the experimental studies a horizontal light sheet at the mid-vane height (equidistant from the hub and shroud) at two different peripheral locations has been used. The camera sees the plane of laser sheet perpendicularly and records the motion of particles and therefore the particle displacements on a doubly exposed single photographic frame.

The experiments on rotor-stator interactions were carried out close to design conditions for the pump. The images were recorded near the mid-section of the volute, corresponding to the sample area shown in figure 3.2. In other words, the basic motive was to refrain from the unsteadiness generated in the pump caused by any other factor than the ones, which may be attributed to the interaction of the impeller blades, and its unsteady out flux with the stator vanes. However, to determine this operating point for the study on impeller-diffuser interaction and a different one for experiments on rotating stall, it was imperative first to characterize the performance of the pump.

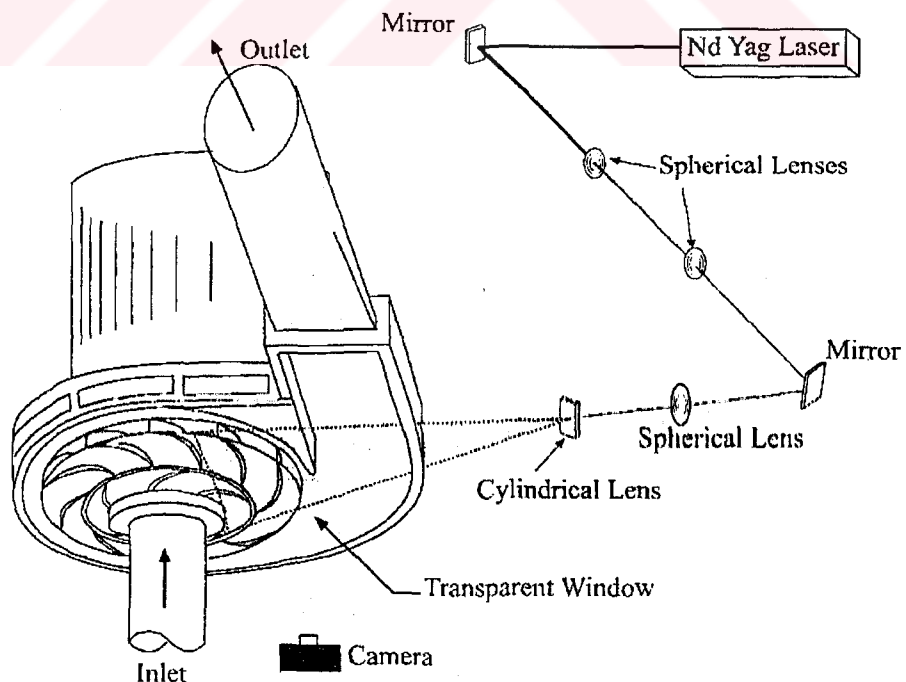


Figure 3.7 Schematic of the optical setup used to create a laser light sheet (Sinha, 1999)

To specify performance curve flow systems, the designer must know the pressure rise (or head), torque, power requirement and efficiency of a machine. For a given machine each of these characteristics for similar machines depends upon size and operating speed. To determine performance, a pump must be on an instrumented stand with the capability of measuring flow rate, speed, input torque, and pressure rise. Measurements are made as flow rate is varied from shutoff (zero flow) to maximum delivery. Power input to the machine is determined from a calibrated motor or calculated from measured speed and torque, and then efficiency is computed. Finally, the calculated characteristics are plotted. The difference in static pressures between the pump section and discharge was used to calculate the head rise produced by the pump. For pumps, dynamic pressure typically is a small fraction of the head rise developed by the pump, so it may be neglected compared with the head rise.

Typical characteristic curves for a centrifugal pump tested at constant speed were obtained; the head versus capacity curve is produced. For this typical machine, head is a maximum at shot off and decreases continuously as flow rate increases. Consequently, to minimize the starting load, it may be advisable to start the pump with the outlet valve closed. Pump efficiency increases with capacity until the BEP is reached, then decreases as flow rate is increased further. For minimum energy consumption, it is desirable to operate as close to BEP as possible.

This study has been made to provide a detailed flow structure between the impeller-diffuser and diffuser-volute in different gaps. The knowledge is needed to understand the nature of flow diffuser interactions in a centrifugal pump with a vaned diffuser that has the different gap geometries. The experimental study is made by Sinha (1999). The pump, used in experiments, has a gap of 20% impeller radius. This gap is larger than the most ones used in obtaining the available data for pumps. The effect of the gap on the flow is intended to show by changing the gap. In the present numerical study, the entire pump is modeled, but the region near the tongue, as shown in figure 3.2 is selected as working area. The PIV data's is presented for this region between 2.51 and 3.8 l/s for five different volume flow rates. At the beginning, the experimental results are compared with the numerical results in order

to show the validation of numerical study. Then, CFD studies were performed as 2D/3D flows in the different flow rates and the different gaps. This study is arranged for fifteen volume different flows to obtain the pump performance and efficiency. The produced 2D/3D models of the wire and solid models are shown in figures 3.4 and 3.6. Numerical results are compared with PIV results, but it is found that PIV results match better with the 3D numerical results comparing to 2D numerical results. Thus, the present study concentrates on 3D numerical results. Studies were performed as a steady and unsteady flow cases. The performance and efficiency curves are prepared using obtained results for three different gaps cases (see chapter 4). The design point values (or BEP) are found as 31.9 kPa - 4.74 l/s in experimental studies of Sinha (1999, 2001). This design point values are quite near the numerically determined design point values which are 36.9 kPa - 4.12 l/s (figure 3.8).

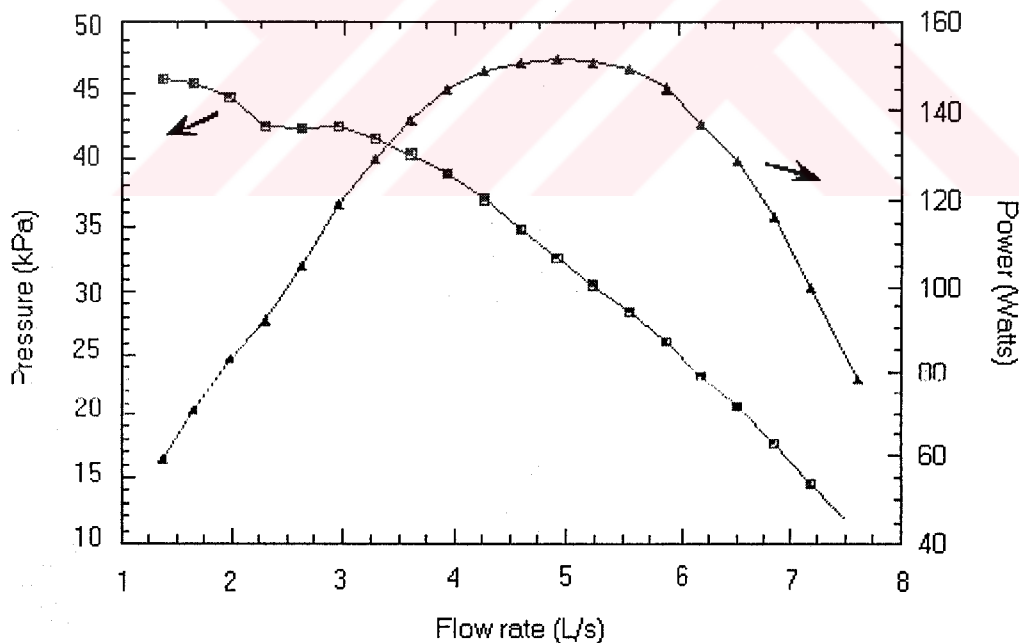


Figure 3.8 The performance and efficiency curve of the centrifugal pump with diffuser vanes (Sinha, 1999)

3.3 Pump Models and Grid Generation

Research interests in the field of rotating machines are whetted by its variety of applications in organizations and industries such as aerospace, automotive, space and marine research. Due to forbidding cost of experimental investigation and rapid development of computational power and relative economic viability of a computational simulation, CFD is emerging as a major tool for analysis and better understanding of the complex physics involved in flow regime of such applications. The wide and critical application interests have been a driving force for research in this area. For the past several years, computational experiments in this area have been a major thrust of research in the world, with the significant advancement.

Fluid mechanics is described by nonlinear equations, which cannot generally be solved analytically, but which have been solved using various approximate methods including expansion and perturbation methods, sundry particle and vortex tracing methods, collocation and integral methods, and Finite Difference Method (FDM), FVM and FEM. Generally these discretization methods have been the most successful, but to use them it is necessary to discretize the field using a grid (mesh).

CFD obtains solutions of the governing NSEs and, depending upon the particular application, solves additional equations involving multiphase, turbulence, heat transfer and other relevant processes. The partial differential NSEs and associated equations are converted into algebraic form (solvable by computer) on a mesh that defines the geometry and flow domain of interest. Appropriate boundary and initial conditions are applied on the mesh, and the distributions of quantities such as velocity, pressure, turbulence, temperature and concentration are determined iteratively at every point in space and time within the domain.

3.3.1 Grid Generation

A mesh is a set of points distributed over a calculation field for a numerical solution of a set of partial differential equations. Essential CFD involves the setting up partial differential equations and boundary conditions describing fluid flow over a

physical domain these equations are then solved numerically to generate the pressure and velocity fields in the flow domain.

In order to perform the discretisation of the governing equations the physical space must be subdivided into a number of cells by means of a structured grid. The grid can be constructed using a rectangular, a cylindrical or a body fitted coordinate system. In the first two cases, the grid lines are aligned with the coordinate axes (either cartesian or cylindrical) while in the latter the grid lines conform to the geometry. The node points, i.e. the points where the grid lines intersect, define the cells generated (figure 3.9). It is possible to enumerate the grid nodes, so that instead of working with the physical coordinates of the nodes, the integer numbers of the enumerated nodes are used. The space where the integer numbers are used is called the computational domain, and it is said to be mapped to the physical domain once a one-to-one relation has been established. The mapping process essentially performs a transformation between the integer indices and the physical coordinates of the nodes and additionally gives weighting factors to each of the faces of the cells in order to compute correctly the fluxes and gradients. A mesh is broken up into control volumes, or cells. Each cell is defined by a set of grid points (or nodes), a cell center, and the faces that bound the cell (Versteeg and Malalasekera (1995) and Anderson (1995)).

A CFD simulation has essentially two critical components, namely domain decomposition and solution of the governing equations on the discretized domain. The process of domain decomposition is called grid generation while the solution algorithm is a procedure for discretizing and solving the governing mathematical model of the physics involved. CFD codes produce physically realistic results with good accuracy in simulations with finite grids. The different types of grids have their advantages and disadvantages in terms of both solution accuracy and the complexity of the grid generation processes. The process of grid generation can be broadly classified in to two categories based on the characteristics of the data structure and the topology of the elements to fill the domain. These two basic categories are known as structured and unstructured grids. A structured grid is defined as set tetrahedral elements with an explicitly defined connectivity. The process of structured grid

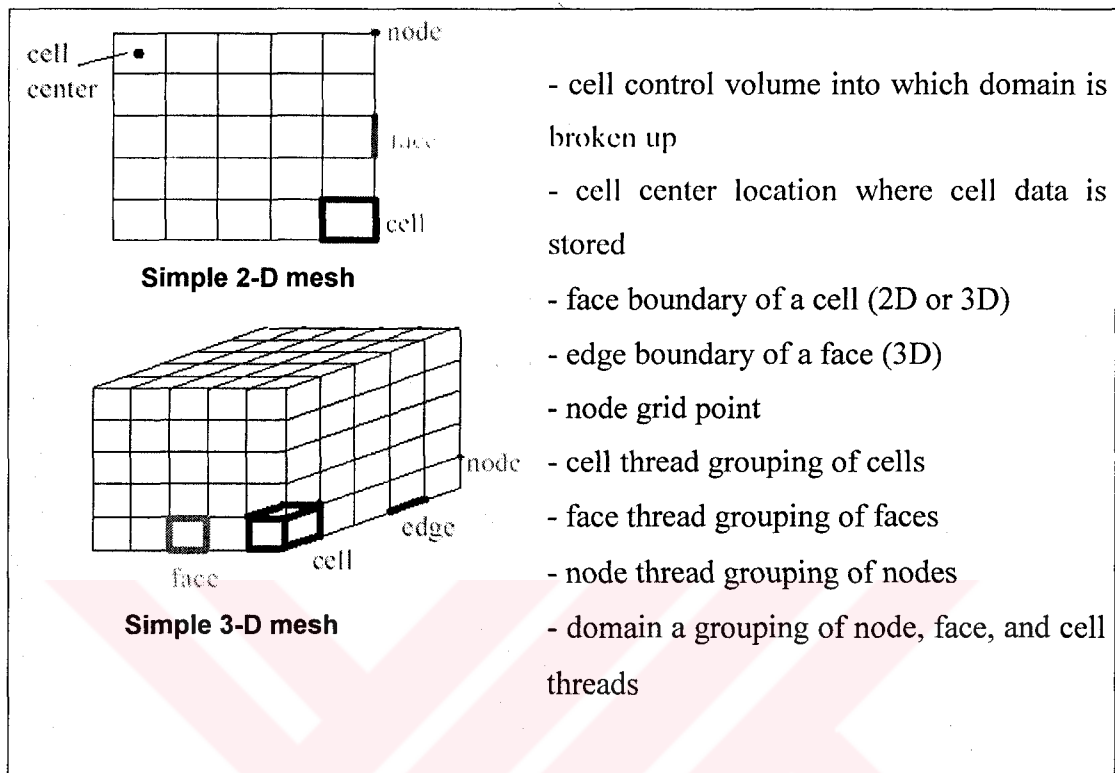


Figure 3.9 Grid components

generation involves breaking the domain in to smaller blocks in which structured grids is generated. An unstructured grid generation has two basic steps, point creation and point connection. The choice of location of point creation and point connection gives the flexibility that makes this type of grid a favored choice for complicated computational domains, while a structured grid may also give a good quality of domain discretization but the process is very time consuming as the grid may need to manually broken into several blocks depending on the nature of the geometry. The flexibility and the convenience make unstructured grid a favorable choice, but a cost, as the solution accuracy may be relatively poor due to presence of skewed elements in sensitive regions like boundary layers. Although structured grid may be more desirable attribute in terms of the solution accuracy, an unstructured grid may be an easier way out for complicated geometries. Due to the accuracy and efficiency concerns with an unstructured grid as it may take a large number of

elements to fill a region with this type of grid a compromise is made in the form of hybrid grid which uses the desirable features of both of the above mentioned generic grid types. In a hybrid grid, the viscous region can be filled with prismatic or hexahedral cells while the rest of the zone can be filled with tetrahedral cells. It has been observed that a hybrid grid in viscous regions creates much lesser number of elements than completely unstructured grid with similar resolution. This idea has been generalized further to give rise to what is called generalized grid. Since generalized grids have no restriction on the number of edges or faces on a cell. It makes this type of grid extremely flexible for topological adaptivity.

CFD package programs are used for modeling fluid flow (incompressible and compressible) and heat transfer in complex geometries. FLUENT can import many different types of grids and solid models from various sources. It can be modified the grid and geometry, reordering the cells in the domain, and merging or separating zones. FLUENT provides complete mesh flexibility, solving flow problems with unstructured meshes that can be generated about complex geometries with relative ease. Supported mesh types include 2D triangular/quadrilateral, 3D tetrahedral/hexahedral/pyramid/wedge, and mixed (hybrid) meshes. FLUENT also allows you to refine or coarsen grid based on the flow solution. FLUENT is ideally suited for incompressible and compressible fluid flow simulations in complex geometries. The structured, unstructured and hybrid grids have been extensively employed in simulation of rotating machines. FLUENT uses unstructured meshes in order to reduce the amount of time you spend generating meshes, simplify the geometry modeling and mesh generation process, model more-complex geometries than you can handle with conventional, multi-block structured meshes, and let you adapt the mesh to resolve the flow-field features. FLUENT is capable of handling triangular and quadrilateral elements (or a combination of the two) in 2D, and tetrahedral, hexahedral, pyramid, and wedge elements (or a combination of these) in 3D. All types of meshes can be adapted in FLUENT in order to resolve large gradients in the flow field, but it must be always generated the initial mesh (whatever the element types used) outside of the solver, using GAMBIT, TGrid, or one of the CAD (Computer Aided Drawing) systems for which mesh import filters exist.

Geometry and grid can be created using GAMBIT. Since FLUENT is an unstructured solver, it uses internal data structures to assign an order to the cells, faces, and grid points in a mesh and to maintain contact between adjacent cells.

Grid generation is an essential aspect of all numerical methods that employ FDM, FEM and FVM for the solution of partial differential equations. Shortly, it consists of dividing bounded or unbounded flow domains into elements (triangles, quadrilaterals, tetrahedrons, etc.) called cells. The topic of grid generation has become a field on its own in the increasingly vast field of CFD. The geometry of the flow domain in computational fluid analyses is usually complicated. To discretize the flow equations, two alternative approaches are commonly used: (i) to map the flow domain to a rectangular or parallel piped domain where a simple equispaced Cartesian grid is used, and (ii) to cover the domain with a unstructured grid.

NSEs are solved on the discretized grid (also known as the mesh), which defines elements in the control-volumes (also known as cell), in the FVM. Values of the solution variables are obtained at the cells, and there is, therefore, usually at least one equation for each variable at each cell. The variables would include velocity components for each “direction” in which the flow is solved, i.e. two components in a 2D flow and three components in a 3D flow, and pressure. At each node in the mesh, between 3 and 20 variables are associated: the pressure, the three velocity components, density, temperature, etc. Furthermore, capturing physically important phenomena such as turbulence requires extremely fine meshes in parts of the physical domain.

CFD programs can use grids comprised of triangular or quadrilateral cells (or a combination of the two) in 2D, and tetrahedral, hexahedral, pyramid, or wedge cells (or a combination of these) in 3D. The choice of which mesh type to use will depend on the application. When choosing the mesh type, it should be considered the following issues; setup time, computational expense, numerical diffusion. In 2D, quadrilateral and triangular cells are accepted, and in 3D, hexahedral, tetrahedral, pyramid, and wedge cells can be used (figure 3.10). Many flow problems solved in engineering practice involve complex geometries. For such problems can be extremely time-consuming, if not impossible. Setup time for complex geometries is,

therefore, the major motivation for using unstructured grids employing triangular or tetrahedral cells. If the geometry is relatively simple, however, there may be no clear saving in setup time with either approach.

When geometries are complex or the range of length scales of the flow is large, a triangular/tetrahedral mesh can often be created with far fewer cells than the equivalent mesh consisting of quadrilateral/hexahedral elements. This is because a triangular/tetrahedral mesh allows cells to be clustered in selected regions of the flow domain, whereas structured quadrilateral/hexahedral meshes will generally force cells to be placed in regions where they are not needed. Unstructured quadrilateral / hexahedral meshes offer many of the advantages of triangular/tetrahedral meshes for moderately complex geometries. One characteristic of quadrilateral/hexahedral elements that might make them more economical in some situations is that they permit a much larger aspect ratio than triangular/tetrahedral cells. A large aspect ratio in a triangular/tetrahedral cell will invariably affect the skewness of the cell, which is undesirable as it may impede accuracy and convergence. Therefore, if you have a relatively simple geometry in which the flow conforms well to the shape of the geometry, such as a long thin duct, you can use a mesh of high aspect-ratio quadrilateral/hexahedral cells. The mesh is likely to have far fewer cells than if you use triangular/tetrahedral cells.

CFD provides a good example of the many areas that a scientific computing project can touch on, and its relationship to Computer Science. Fluid flows are modeled by a set of partial differential equations, the NSEs. Solving a particular problem generally involves first discretizing the physical domain that the flow occurs in, such as the interior of a pump. This discretization is straightforward for very simple geometries such as rectangles or circles, but is a difficult problem in CFD for more complicated objects. Currently automatic “mesh generators” are simply not adequate, requiring extensive investment of time on the part of the scientist or engineer.

Domain decomposition is often expressed as a graph-partitioning problem, namely finding a minimum edge cut partitioning of the discrete mesh, with roughly the same number of nodes in each partition set. Surfaces between inlet-impeller and

impeller volute correspond to grid interfaces. The mesh technique provided by FLUENT allows the relative motion of the impeller grid with respect to the inlet and the volute during unsteady simulation. The grid faces do not need to be aligned on both sides. The topological information to be verified begins with the number of faces and nodes per cell. A triangular cell (2D) should have 3 faces and 3 nodes, a tetrahedral cell (3D) should have 4 faces and 4 nodes, a quadrilateral cell (2D) should have 4 faces and 4 nodes, and a hexahedral cell (3D) should have 6 faces and 8 nodes.

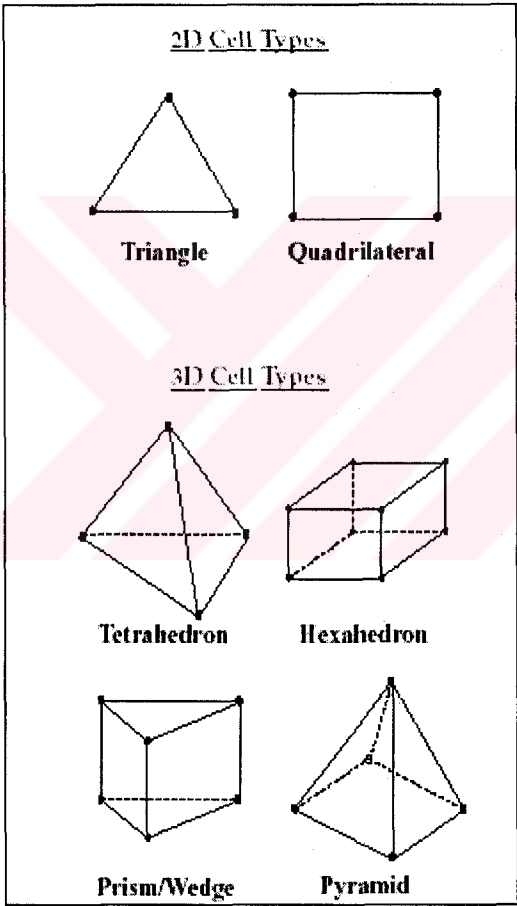


Figure 3.10 Cell types

According to CFD practice, a unstructured mesh allows greater flexibility in adapting a grid to domain boundaries in case of complex shapes, since it allows a more detailed mesh design. In order to approximate the volumes outlined by the complex impeller shape involved, in this study it is used unstructured triangular

elements and the unstructured grid composed of hexahedron and wedge elements in 3D models. GAMBIT v. 2.0 was used to construct the mesh.

3.3.2. Grid Refinement Study

To validate the numerical results, it is of great importance to check whether the grid that has been used to discretize the computational domain complies with the requirements of the turbulence model used.

To check the grid used for the whole pump model, simulations were performed for Case I with three different grids. The used first, second and third grids have about 200.000, 300.000 and 400.000 elements, respectively. The grids were compressed near the solid surfaces, especially rotating surfaces of the blade to ensure adequate resolution. For turbulent simulations, the grids were adapted in order to further resolve the flow near the wall regions. All grids are shown in Figure 3.11. The simulations were made with the same boundary conditions at five different volume flow rates (1.96, 2.86, 3.48, 4.12, and 4.74 l/s). A comparison of the performance curves from different simulations was made at different flow rates. Figure 3.12 shows the performance curves at Case I of impeller radius for three different grids. The results for the three grids are similar, only deviating by about 4% for head. One can therefore conclude that the coarseness of the grid in the pump does not have a strong effect on these results. Thus, it was used the second grid (300.000 elements) to better visualize the flow in pump.

For a given geometry if the solutions of working grid and a fine grid are essentially the same, then the working grid solution is considered to be grid independent. To verify the numerical model, several simulations were performed with different orders of discretizations.

The pressure difference changed in the number of grid elements from about 200.000 to 400.000 for the same gap (Case I) about 4%. So that, total number of 300.000 grid elements were decided as a working grid. The computational grid layout is shown in Figure 3.11(b). The computational domain contains approximately 300.000 grid elements. The pump was discretized using four regions (see figure 3.6).

The generated mesh must be sufficiently dense that the numerical approximation is an accurate one, but it cannot be so dense that the solution is impractical to obtain. Generally, the grid spacing should smoothly and sufficiently be refined to resolve changes in the gradients of the solution. The grid should also be constructed with the computational efficiency in mind. The accuracy of a numerical approximation can also be impaired, if a grid changes discontinuously or is too skewed. Creating a good quality mesh is essential for obtaining accurate, robust, and reliable CFD solutions. Good mesh quality means:

- Minimizing element distortion (also called skewness), aspect ratio, and abrupt size variations
- Having required resolution in high gradient regions of the solution (e.g. wall boundary layers and shear layers)

The quality of the mesh has a significant impact on rate of convergence (or even lack of convergence), solution accuracy, Central Processing Unit (CPU) time required.

Good Quality = cells/elements

should not be highly distorted (also known as skewness). Skewness is determined according to cell size or angle. Quality classification based on skewness values is shown in table 3.2.

Table 3.2 Quality classification based on skewness

<i>Value of Skewness</i>	<i>Excellent</i>	<i>Good</i>	<i>Acceptable</i>	<i>Poor</i>	<i>Sliver</i>	<i>Degenerate</i>
<i>Cell Quality</i>	0-0.25	0.25-0.50	0.50-0.80	0.80-0.95	0.95-0.99	0.99-1.00

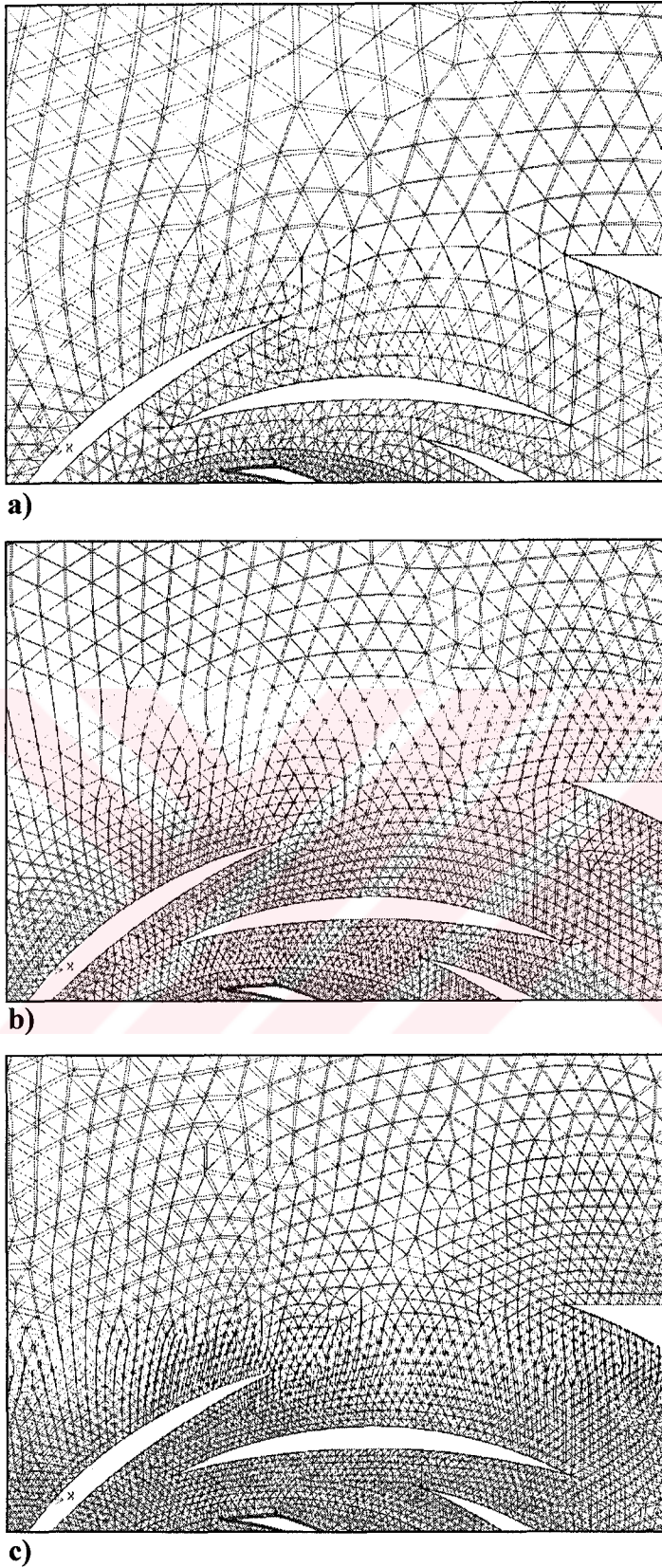


Figure 3.11 A view of used grids for grid independency a) 200.000 elements
b) 300.000 elements c) 400.000 elements

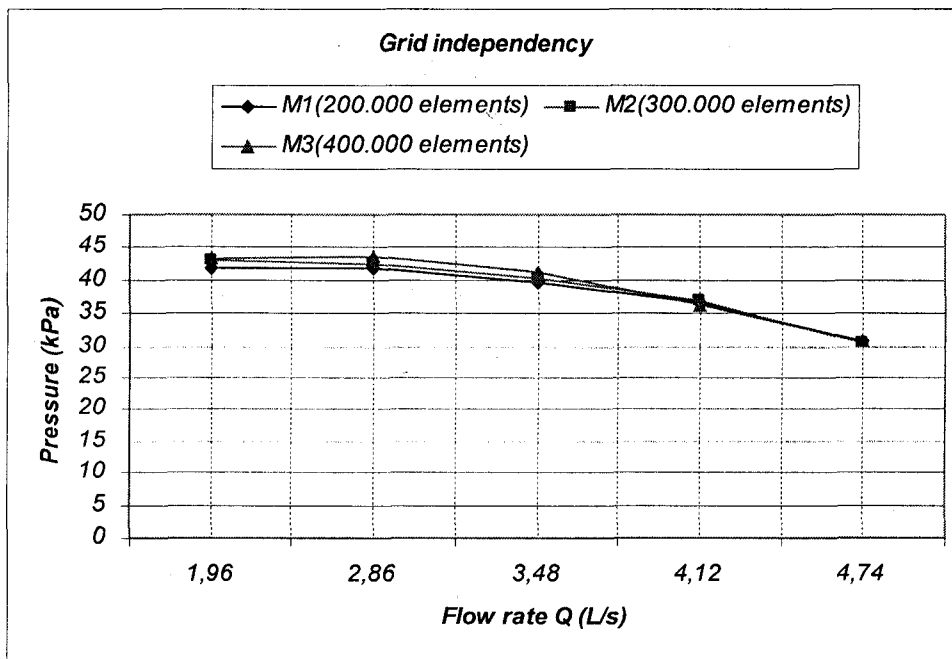


Figure 3.12 The performance curves for different grids

The quality of the mesh plays a significant role in the accuracy and stability of the numerical computation. The attributes associated with mesh quality are node point distribution, smoothness, and skewness. Since, a continuous domain is discretely defining; the degree to which the salient features of the flow (such as shear layers, separated regions, shock waves, boundary layers, and mixing zones) are resolved depends on the density and distribution of nodes in the mesh. In many cases, poor resolution in critical regions can dramatically alter the flow characteristics. But, the grid resolution in most complicated 3D flow fields will be constrained by CPU time and computer resource limitations (i.e., memory and disk space). Although accuracy increases with larger grids, the CPU and memory requirements to compute the solution and post-process the results also increase. Solution-adaptive grid refinement can be used to increase and/or decrease grid density based on the evolving flow field, and thus provides the potential for more economical use of grid points (and, hence, reduced time and resource requirements). Smoothness Rapid changes in cell volume between adjacent cells translate into larger truncation errors. Truncation error is the difference between the partial derivatives in the governing equations and their discrete approximations.

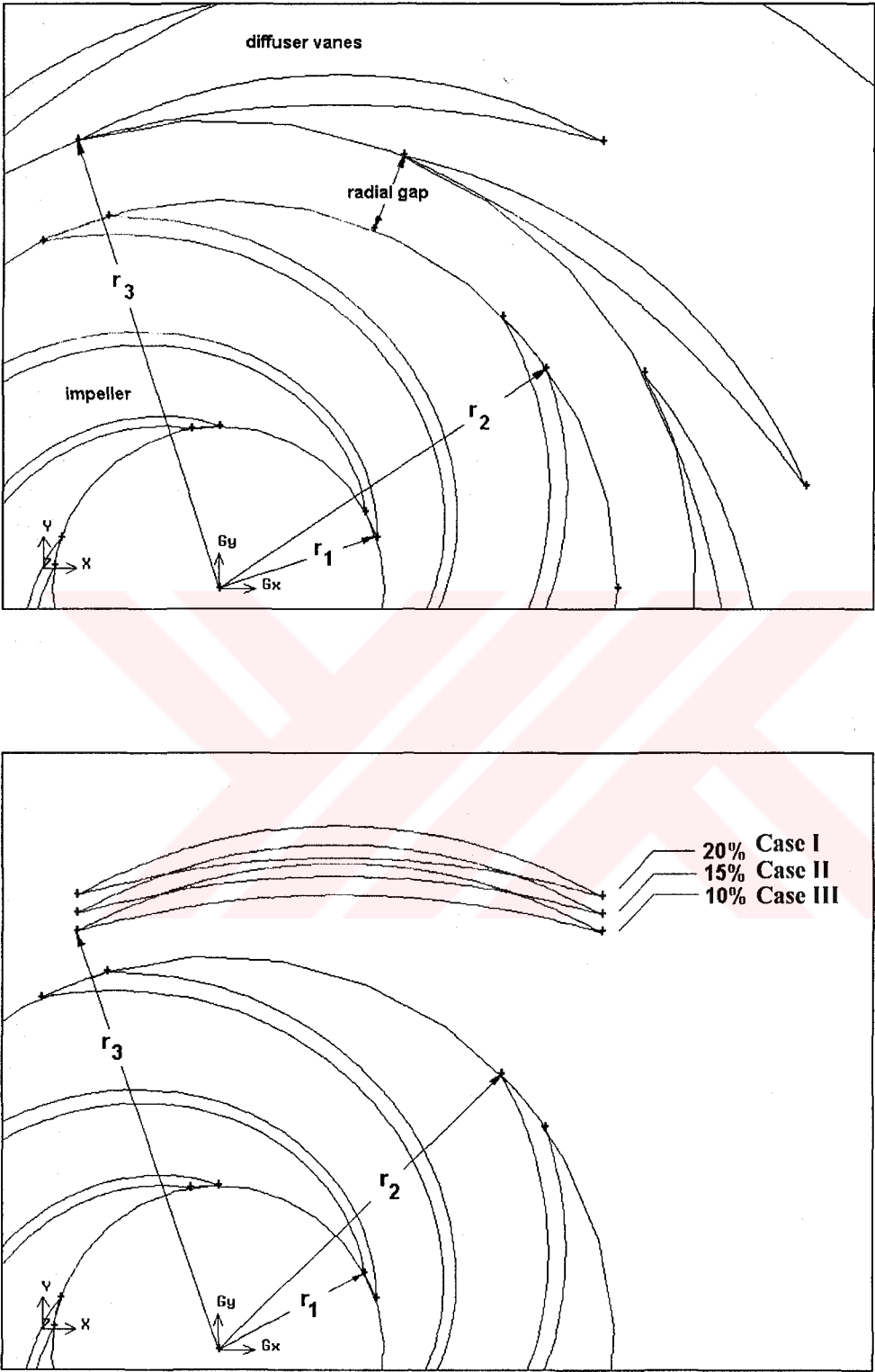


Figure 3.13 The positions of impeller-diffuser and gaps

CFD programs provide the capability to improve the smoothness by refining the mesh based on the change in cell volume or the gradient of cell volume. The shape of the cell (including its skewness and aspect ratio) also has a significant impact on the accuracy of the numerical solution. It is well known that the grid structure must be selected carefully in order to achieve accurate resolution of the complex flow fields. The minimization of the grid skewness generally results in a faster convergence as well as in less dependence of the solution on the grid density; therefore reducing computational cost both in terms of memory and CPU time. As a consequence, the grid generation procedure should be considered an integral part of the numerical method, especially when approaching advanced applications where the computational overhead can be severe, or when dealing with routine design. Skewness can be defined as the difference between the cell's shape and the shape of an equilateral cell of equivalent volume. Highly skewed cells can decrease accuracy and destabilize the solution (for example, optimal quadrilateral meshes will have vertex angles close to 90° , while triangular meshes should preferably have angles of close to 60° and have all angles less than 90°). Aspect ratio is a measure of the stretching of the cell. However, a general rule of thumb is to avoid aspect ratios in excess of 5:1. The effect of resolution, smoothness and cell shape on the accuracy and stability of the solution process is strongly dependent on the flow field being simulated. For example, much skewed cells can be tolerated in flow regions, but can be very damaging in regions with strong flow gradients. Since the locations of strong flow gradients generally cannot be determined a priori, you should strive to achieve a high-quality mesh over the entire flow domain.

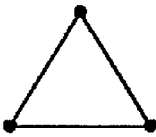
3.3.3 The Pump Models Grid

Two and three dimensional pump models and grid generations of three states of gap (20%, 15% and 10% of impeller radius) were produced with the pre-processor GAMBIT, which offers some CAD tools as well as meshing tools for CFD analysis. The positions of impeller-diffuser and the gaps used in models were shown in figure 3.13. Radial gap between impeller blades trailing edge and diffuser vanes leading

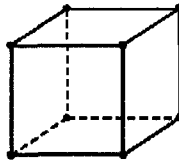
edge is an important parameter influencing the flow in a centrifugal pump. In this study, it is defined as

$$\delta = \frac{(r_3 - r_2)}{r_2} = \frac{(122.25 - 101.6)}{101.6} \cong 0.20$$

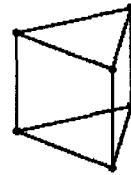
Where, δ was named as the radial gap ratio. Computational study of radial gap effect was completed on the geometry of a centrifugal pump. The pump has an impeller of 101.6 mm discharge radius with 5 blades and diffuser of 122.5 mm inside radius with 9 vanes. Three radial gap ratios were studied with radial gap 20%, (actual pump geometry), 15% and 10%. These radial gaps were named case I for 20%, case II and case III 10%. The radial gap change was made without spoiling pump geometry. All geometry parameters of the impeller, diffuser and volute were kept unchanged. So the result presents a “pure” effect of the radial gap on the flow. Three types of radial gap geometries are shown in figure 3.13. One of the most important and time-consuming tasks in the processes of a CFD simulation is the generation of the computational grid. In the present study, in the 2D models meshes triangular elements and in the 3D models meshes were used hexahedron and wedge elements (figure 3.14).



a) Triangular element
(2D / 3 Node)



b) Hexahedron element
(3D / 8 Node)



c) Wedge element
(3D / 6 Node)

Figure 3.14 Mesh elements

Views of 2D and 3D model mesh are shown in figure 3.15. Triangular elements in the grid structure of 2D models are preferred to be more suitable to this structure.

In this study, the structured and unstructured meshes are used. The structured hexahedron elements are generated to define the inlet, while unstructured wedge elements are used to define impeller and volute in 3D pump models. In the volute, a mesh refinement zone is defined for a region near the diffuser vanes and the walls. The mesh points were clustered toward volute wall surface, blade surface and vane surface, with a typical ratio of 1.2 between adjacent mesh. Once the model is defined, the geometry is to be ready to be simulated, and then the mesh is created. A view of generated mesh elements distribution can be seen for 3D geometry and a detail of the impeller grid is shown in figure 3.16.

The used elements and nodes numbers, properties of the elements for each 2D/3D pump models is presented in table 3.3 and table 3.4 since more detailed flow structure was desired relatively fine grids were used near diffuser, impeller blades, and wall surface, whereas the grids in other regions were coarse. The volute along z axis is divided by six parts to use fine grids at the neighborhood of the volute walls.

The size of resulting cells is not enough for a full boundary layer simulation but it gives correct values for the pump performance and allows the analysis the details of the main phenomena involved.

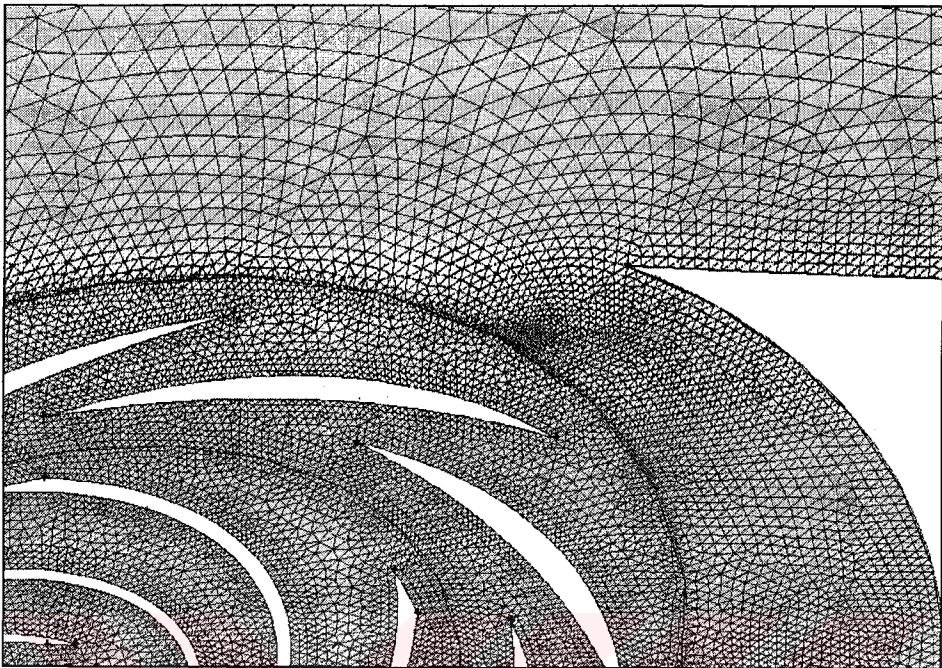
A special technique called sliding mesh was used in the present solution to accommodate the relative motion between the impeller and stationary parts of the diffuser vanes and volute casing. This allowed portions of mesh in the rotary part of the impeller to slide in the small incremental movements relative to those of the stationary part during simulations.

Surfaces between inlet-impeller and impeller-diffuser define the grid interfaces needed for the relative rotation of the impeller. The sliding mesh technique provided by FLUENT allows that the relative motion of the impeller grid with respect to the inlet and the volute during unsteady simulations.

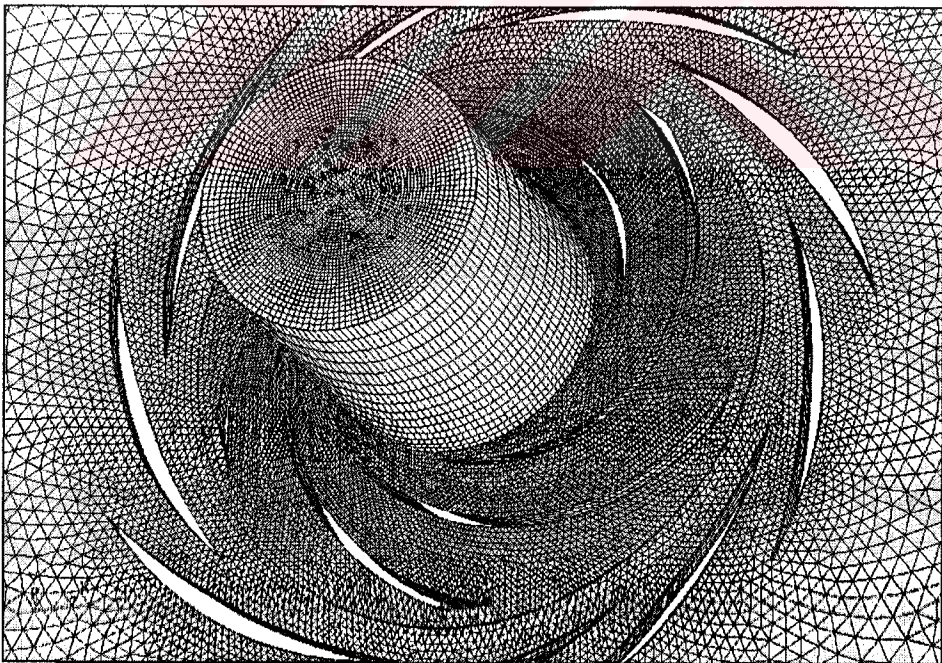
In this study, there are the interface between impeller fluid and volute fluid in 2D models, in the same way; two interfaces are created between stationary fluids and moving fluids in 3D models.

For a given geometry if the solutions of working grid and a fine grid are essentially the same, then the working grid solution is considered to be grid independent. To verify numerical model, two simulations were performed with different orders of discretization. Tests with different mesh configurations were carried out in order to obtain grid independent solution (FLUENT User's Guide (2000)).





a)



b)

Figure 3.15 A view of model mesh a) 2D mesh, b) 3D mesh

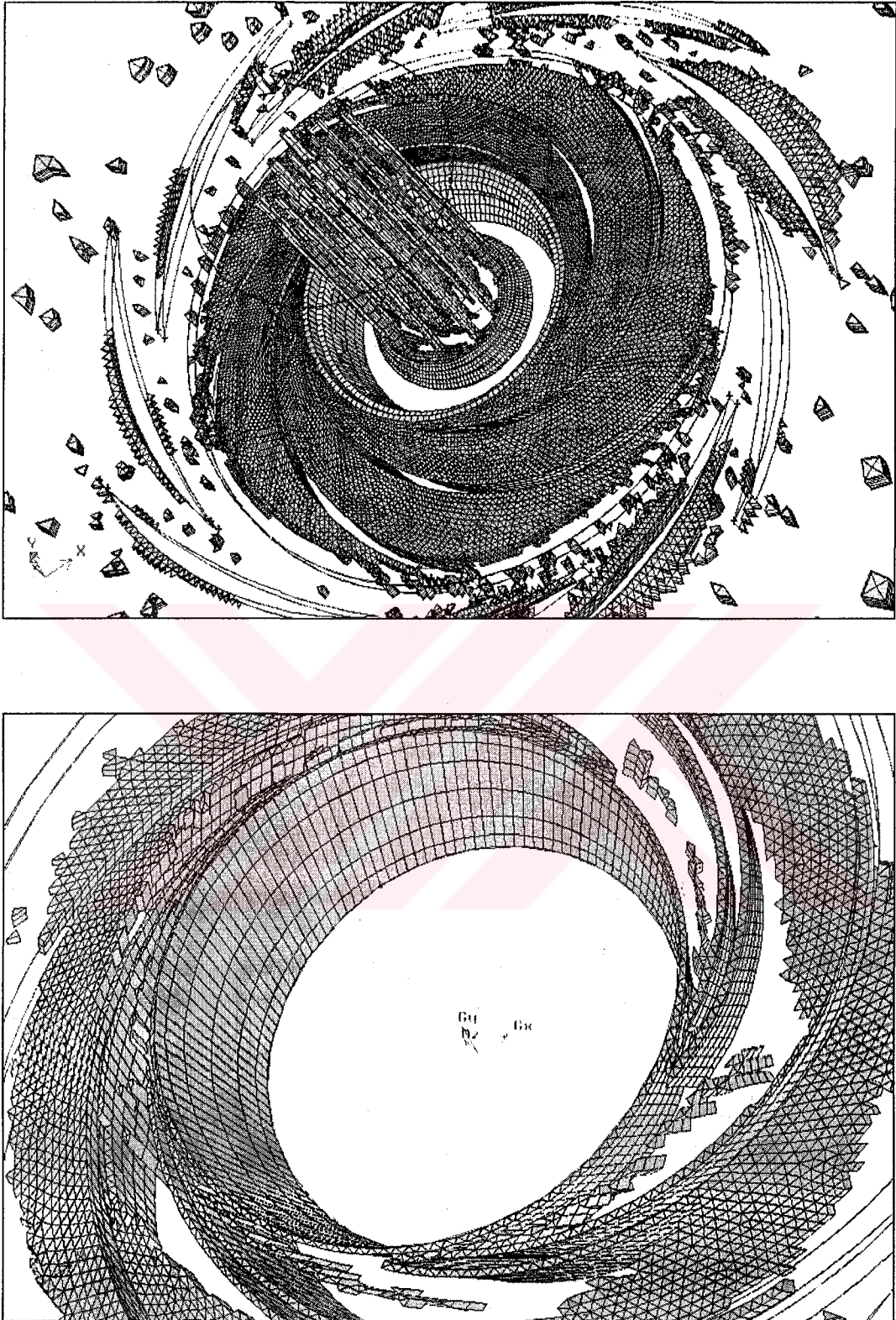


Figure 3.16 The views of the distribution of the mesh elements in the 3D pump model

Table 3.3 The 2D Grid size of all fluid zones and complete domain

<i>Cell quality</i>		<i>Count in range</i>			<i>% Of total count</i>		
<i>From value</i>	<i>To value</i>	<i>20%</i>	<i>15%</i>	<i>10%</i>	<i>20%</i>	<i>15%</i>	<i>10%</i>
0	0.1	27880	29595	29865	48.14	51.86	52.54
0.1	0.2	23322	21275	20725	40.27	37.28	36.46
0.2	0.3	4973	5016	5125	8.59	8.79	9.016
0.3	0.4	1380	1103	1057	2.38	1.93	1.86
0.4	0.5	297	67	62	0.51	0.12	0.11
0.5	0.6	49	10	8	0.08	0.02	0.014
0.6	0.7	16	0	0	0.03	0	0
0.7	0.8	0	0	0	0	0	0
0.8	0.9	0	0	0	0	0	0
0.9	1.0	0	0	0	0	0	0
Total		57918	57066	56842	100	100	100
<i>Zone name</i>	<i>El. type</i>	<i>Total elements</i>			<i>Total nodes</i>		
		<i>20%</i>	<i>15%</i>	<i>10%</i>	<i>20%</i>	<i>15%</i>	<i>10%</i>
<i>Imp</i>	<i>Triang.</i>	11958	12056	12964	6688	6738	7045
<i>Diff.</i>	"	17190	16618	15448	9615	9245	8665
<i>Volute</i>	"	28770	28392	28392	15165	14891	14210
Total		57918	57066	56842	31468	30874	29920

Table 3.4 The 3D Grid size of all fluid zones and complete domain

<i>Cell quality</i>		<i>Count in range</i>			<i>% Of total count</i>		
<i>From value</i>	<i>To value</i>	<i>20%</i>	<i>15%</i>	<i>10%</i>	<i>20%</i>	<i>15%</i>	<i>10%</i>
0	0.1	97437	93972	101236	32.96	31.21	35.54
0.1	0.2	95260	94852	85162	32.22	31.5	29.89
0.2	0.3	52763	58019	48974	17.85	19.27	17.19
0.3	0.4	34685	38792	34256	11.73	12.88	12.02
0.4	0.5	10651	10747	10189	3.60	3.57	3.58
0.5	0.6	3146	3043	3398	1.06	1.01	1.19
0.6	0.7	1576	1576	1559	0.53	0.52	0.55
0.7	0.8	114	103	106	0.04	0.03	0.04
0.8	0.9	0	0	0	0	0	0
0.9	1.0	0	0	0	0	0	0
Total		295632	301104	284880	100	100	100
<i>Zone name</i>	<i>El. type</i>	<i>Total elements</i>			<i>Total nodes</i>		
		<i>20%</i>	<i>15%</i>	<i>10%</i>	<i>20%</i>	<i>15%</i>	<i>10%</i>
<i>Inlet</i>	<i>Hex.</i>	52260	52260	52260	56832	56832	56832
<i>Imp-in</i>	"	20904	20904	20904	24864	24864	24864
<i>Imp.</i>	<i>Wedge</i>	71580	71580	71580	46060	46060	46060
<i>Volute</i>	"	150888	156360	140136	94220	97734	87906
Total		295632	301104	284880	221976	225490	215662

CFD is the art of replacing the differential equation governing the Fluid Flow, with a set of algebraic equations, which in turn can be solved with the aid of a digital computer to get an approximate solution. CFD modeling is the process of representing a fluid flow problem by mathematical equations based on the fundamental laws of physics and solving those equations to predict the variation of the relevant parameters within the flow field domain. Usually, these will be velocity, pressure and temperature, and other variables such as turbulence parameters and concentrations of chemical species. The equations will include those for the conservation of momentum, which are sometimes referred to as the NSEs, and the conservation of mass and the conservation of energy. In solving these equations the velocity, pressure, and temperature are predicted throughout the domain field. The solution of an additional equation must be obtained for each new variable to be calculated. These equations, for the conservation of momentum, energy, mass, etc., are known as field equations; they represent the variation of solution variables in space and time. They are of partial differential form and, in the mathematical sense, are highly non-linear and cannot be solved analytically except in very simplified cases that are of limited practical use. The approach within CFD is to represent the equations in numerical form, linearize the equations and then solve them using numerical analysis techniques. The act of linearizing the equations means that an iterative calculation procedure must be adopted, whereby the equations are successively re-linearized and solved until the solution to the original numerical form of the equations is attained.

The CFD simulation of real turbomachinery flows can be realized using time-dependent numerical methods, i.e. the rotation of the impeller relative to non-moving parts of the machine is modeled. Although enormous computer resources are needed for this modeling, it simulates real flow physics best of all, while steady or quasi-steady numerical approaches only approximate the real flow, because they neglect important effects of real physics. Nevertheless the steady models are quite popular in industrial use, mainly due to relative short time compared to transient calculations needed to obtain numerical results. In this study, MRF and SMM are

used for steady and unsteady flow, respectively. Then, the results were compared with detailed investigations in 2D and 3D models.

The numerical code used (FLUENT) solves the fully 2D/3D incompressible NSEs, including the centrifugal force in the impeller and unsteady terms. A turbulence model study and grid sensitivity study performed. Turbulence is modeled using statistical approaches for the unknown Reynolds-Stress-Tensor in the NSEs. Models using two additional transport equations for turbulence quantities are the most common ones because they have been tested for wide range of applications and they are known to be good compromise between precision economies. Turbulence is simulated with the RNG $k-\varepsilon$ model. Since there are no significant differences in the results, only the RNG, $k-\varepsilon$ results are shown (Meneveau and Katz (2002)). Although grid size is not adequate to investigate local boundary layer variables, global ones are well captured. For such calculations, wall functions, based on the logarithmic law, have been used. The time dependent term scheme is second order, implicit. The pressure velocity couplings are calculated through the SIMPLEC algorithm. Second order, upwind discretizations have been used for convection terms and central difference schemes for diffusion terms.

3.4 Governing Equations

For all flows, FLUENT solves conservation equations for mass and momentum. For flows involving heat transfer or compressibility, an additional equation for energy conservation is solved. For flows involving species mixing or reactions, a species conservation equation is solved or, if the non-premixed combustion model is used, conservation equations for the mixture fraction and its variance are solved. Additional transport equations are also solved when the flow is turbulent.

3.4.1 The Mass Conservation Equation

The equation for conservation of mass, or continuity equation, can be written as follows:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = S_m \quad (3.2)$$

Equation 3.2 is the general form of the mass conservation equation and is valid for incompressible as well as compressible flows. The source S_m is the mass added to the continuous phase from the dispersed second phase (e.g., due to vaporization of liquid droplets) and any user-defined sources.

3.4.2 Momentum Conservation Equations

Conservation of momentum in an inertial (non-accelerating) reference frame is described by

$$\frac{\partial}{\partial t}(\rho \vec{v}) + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p + \nabla \cdot (\vec{\tau}) + \rho \vec{g} + \vec{F} \quad (3.3)$$

where p is the static pressure, $\vec{\tau}$ is the stress tensor (described below), and $\rho \vec{g}$ and $\vec{F}$ are the gravitational body force and external body forces (e.g., that arise from interaction with the dispersed phase), respectively. $\vec{F}$ also contains other model-dependent source terms such as porous-media and user-defined sources. The stress tensor $\vec{\tau}$ is given by

$$\vec{\tau} = \mu \left[(\nabla \vec{v} + \vec{v} \nabla) - \frac{2}{3} \nabla \cdot \vec{v} I \right] \quad (3.4)$$

where μ is the molecular viscosity, I is the unit tensor, and the second term on the right hand side is the effect of volume dilation.

3.4.3 Transport Equations

Recall that the FLUENT solver is based on the FVM, where the domain is discretized into a finite set of control volumes or cells. The cells of a mesh are the basic computational elements in FLUENT in which property conservation is enforced. That is, the integral form of the general transport equation for mass, momentum, energy, etc. can be applied to each cell:

$$\underbrace{\frac{\partial}{\partial t} \int_V \rho \phi dV}_{\text{Unsteady}} + \underbrace{\oint_A \rho \phi V \cdot dA}_{\text{Convection}} = \underbrace{\oint_A \Gamma \nabla \phi \cdot dA}_{\text{Diffusion}} + \underbrace{\int_V S_\phi dV}_{\text{Generation}} \quad (3.5)$$

where ϕ is a variable that is used to describe a general transportable quantity and takes on different values, depending on the transport equation that is being solved. There is a subset of the variables for ϕ that are solved in transport equations.

Transport Equation Variable for ϕ : continuity 1, x momentum velocity (u), y momentum velocity (v), z momentum velocity (w), energy enthalpy (h), turbulent kinetic energy k , turbulent dissipation rate ε , and species transport mass fraction of species (Y_i). Evaluation of the conservation laws requires that fluxes across the cell boundaries be known. Consequently, properties need to be evaluated at the cells and faces.

3.5 Modeling of Flow

Most of CFD programs can model flow involving moving reference frames and moving cell zones, using several different approaches, and flow in moving and deforming domains. Solving flows in moving reference frames requires the use of moving cell zones. This cell zone motion is interpreted as the motion of a reference frame to which the cell zone is attached. With this capability, a wide variety of problems that involve moving parts can be set up and solved using FLUENT.

Depending on the level of complexity of the motion, and on the flow physics involved, one of the moving cell zone models may be the most suited to the application. The solution of flows in moving reference frames requires the use of moving cell zones. This cell zone motion is interpreted as the motion of a reference frame to which the cell zone is attached. This moving cell zone capability provides a powerful set of features for solving problems in which the domain on parts of the domain are in the motion. The computational domain can consist only one of the moving cell zone a combination of moving and stationary cell zones: *i*) Flow in a (single) rotating frame, *ii*) Flow in multiple rotating and/or translating reference frames.

The single rotating frame option can be used to model flows in turbomachinery, mixing tanks, and related devices. In each of these cases, the flow is unsteady in an inertial frame (i.e., a domain fixed in the laboratory frame) because the impeller/impeller blades sweep the domain periodically. However, in the absence of diffusers or baffles, it is possible to perform calculations in a domain that moves with the rotating part. In this case, the flow is steady relative to the rotating (non-inertial) frame, which simplifies the analysis.

If diffusers or baffles are present in addition to a impeller or impeller, then it is not possible to render the computational problem steady by choosing a calculation domain that rotates with the impeller or impeller. This situation occurs, for example, in turbomachinery applications where impeller and diffuser blades are in close proximity (and hence impeller-diffuser interaction is important). The CFD programs offer different approaches for treating this problem, all of them using multiple frames of reference:

- The multiple reference frame (MRF) model
- The mixing plane model
- The sliding mesh (SM) model

Both the MRF model assumes that the flow field is steady, with the impeller-diffuser or impeller-diffuser effects being accounted for by approximate means. These can be acceptable models in cases where the impeller-diffuser interaction is weak or an approximate solution for the system is desired. On the other

hand, SM model assumes that the flow field is unsteady, and hence models the interaction with complete fidelity. This is the model of choice if impeller-diffuser interaction is strong and a more accurate simulation of the system is desired. Note, however, that because SM model requires an unsteady numerical solution, it is computationally more demanding than the MRF model.

The MRF model is a steady-state approximation in which individual cell zones move at different rotational/translational speeds. This approach is appropriate when the flow at the boundary between these zones is nearly uniform. While the multiple reference frame approach is clearly an approximation, it can provide a reasonable model of the time-averaged flow for many applications. For example, the MRF model can be used for a turbomachinery application. Another potential use of the MRF model is to compute a flow field that can be used as an initial condition for a transient sliding mesh calculation. This eliminates the need for a startup calculation. The multiple reference frame models should not be used, however, if it is necessary to actually simulate the transients that may occur in strong impeller-diffuser interactions. For such cases, the sliding mesh model should be used. Steady state flow conditions are assumed at the interface between the two reference frames. That is, the velocity at the interface must be the same (in absolute terms) for each reference frame. The grid does not move. It can also be modeled a problem that includes more than one rotating reference frame.

The central issues in CFD into obtain numerical solution to the complete NSEs. The basic step in obtaining a solution for any find of problem requires the equations to be written with reference to a coordinate system. The typical coordinate systems used are global reference frame system and body reference frame system.

3.5.1 The MRF Model

The MRF formulation will depend on the velocity formulation being used. Relative Velocity Formulation; in FLUENT's implementation of the multiple reference frame feature, the calculation domain is divided into subdomains, each of which may be rotating/translating with respect to the laboratory (inertial) frame. The

governing equations in each subdomain are written with respect to that subdomain's reference frame. Thus, the flow in stationary or translating subdomains and rotating subdomains are governed by different equations. At the boundary between two subdomains, the diffusion and other terms in the governing equations in one subdomain require values for the velocities in the adjacent subdomain.

FLUENT enforces the continuity of the absolute velocity, $\vec{v}$, to provide the correct neighbor values of velocity for the subdomain under consideration. When the relative velocity formulation is used, velocities in each subdomain are computed relative to the motion of the subdomain. Velocities and velocity gradients are converted from a moving reference frame to the absolute inertial frame as described below. The position vector relative to the origin of the zone rotation axis is defined as

$$\vec{r} = \vec{x} - \vec{x}_0 \quad (3.6)$$

where $\vec{x}$ is the position in absolute Cartesian coordinates and $\vec{x}_0$ is the origin of the zone rotation axis, as shown in Figure 3.17. The relative velocity in the moving reference frame can be converted to the absolute (stationary) frame of reference using the following equation:

$$\vec{v} = \vec{v} + (\vec{\omega} \times \vec{r}) + \vec{v}_t \quad (3.7)$$

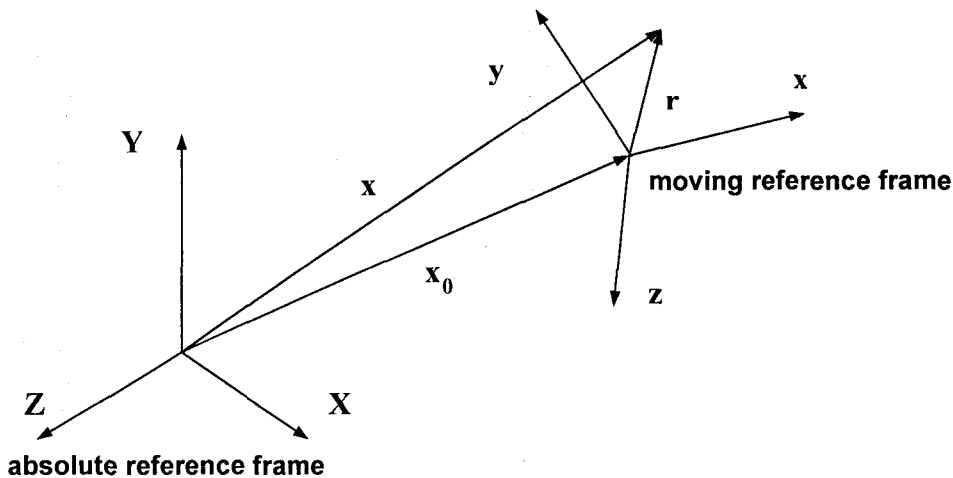


Figure 3.17 Coordinate system for relative velocity

where $\vec{v}$ is the velocity in the absolute inertial reference frame, $\vec{v}_r$ is the velocity in the relative non-inertial reference frame, and $\vec{v}_t$ is the translational velocity of the noninertial reference frame. Using this definition of absolute velocity, the gradient of the absolute velocity vector is given by

$$\nabla \vec{v} = \nabla \vec{v}_r + \nabla (\vec{\omega} \times \vec{r}) \quad (3.8)$$

3.5.2 The SM Model

When a time-accurate solution for rotor-stator interaction (rather than a time-averaged solution) is desired, you must use the sliding mesh model to compute the unsteady flow field. The sliding mesh model is the most accurate method for simulating flows in multiple moving reference frames, but also the most computationally demanding. Most often, the unsteady solution that is sought in a sliding mesh simulation is time periodic. That is, the unsteady solution repeats with a period related to the speeds of the moving domains. However, you can model other types of transients, including translating sliding mesh zones. Note that for flow situations where there is no interaction between stationary and moving parts (i.e., when there is only a rotor), the computational domain can be made stationary by using a rotating reference frame. When transient rotors-stator interaction is desired, you must use sliding meshes. If you are interested in a steady approximation of the interaction, you may use the multiple reference frame model.

In the sliding mesh technique two or more cell zones are used (if you generate the mesh in each zone independently, you will need to merge the mesh files prior to starting the calculation). Each cell zone is bounded by at least one “interface zone” where it meets the opposing cell zone. The interface zones of adjacent cell zones are associated with one another to form a “grid interface”. The two cell zones will move relative to each other along the grid interface. Note that the grid interface must be positioned so that it has fluid cells on both sides. During the calculation, the

cell zones slide (i.e., rotate or translate) relative to one another along the grid interface in discrete steps. Initial position of two grids and their positions after some translation have occurred. As the rotation or translation takes place, node alignment along the grid interface is not required. Since the flow is inherently unsteady, a time-dependent solution procedure is required. The grid interface and the associated interface zones can be any shape, provided that the two interface boundaries are based on the same geometry.

The sliding mesh model allows adjacent grids to slide relative to one another. In doing so, the grid faces do not need to be aligned on the grid interface. This situation requires a means of computing the flux across the two non-conformal interface zones of each grid interface. To compute the interface flux, the intersection between the interface zones is determined at each new time step. The resulting intersection produces one interior zone (a zone with fluid cells on both sides) and one or more periodic zones. The resultant interior zone corresponds to where the two interface zones overlap; the resultant periodic zone corresponds to where they do not. The number of faces in these intersection zones will vary as the interface zones move relative to one another. Principally, fluxes across the grid interface are computed using the faces resulting from the intersection of the two interface zones, rather than from the interface zone faces themselves.

In the grid interface, between a stationary and moving mesh, impeller-volute, the overlapping faces in the interfaces zones are determined at the each new time step. Fluxes across each grid interface are calculated proportionally to the areas of the superposed faces. The pressure and velocity fluctuations at the interface have been numerically tested and the sliding technique has proven its reliability, keeping the same accuracy levels observed in any other mesh position.

3.5.3 Rotating Reference Frame

When a model is created, it is typically modeling the flow in an inertial reference frame (i.e., in a non-accelerating coordinate system). However, it also has an accelerating reference frame. In this situation the accelerating of the coordinate

system is include in the equations of motion describing the flow. A common example of an accelerating reference frame in engineering applications is flow in rotating equipment. Many such flows can be modeling in rotating coordinate system that is moving with the rotating equipment and thus experiences a constant acceleration in radial direction. This class of rotating flows can be treated using the rotating reference frame capability in the programs. When such a rotating reference frame problems are defined, the rotating boundaries become stationary relative to the rotating frame, since they are moving at the same speed as the reference frame. Rotor-stator interaction problems cannot be modeled by a simple coordinate transformation to a rotating reference frame. In FLUENT, impeller-diffuser interaction must be treated by applying the MRF, mixing plane, or SM approaches.

When such problems are defined in a rotating reference frame, the rotating boundaries become stationary relative to the rotating frame, since they are moving at the same speed as the reference frame. For all flows, CFD solvers solve conservation equations for mass and momentum. The equations for conservation of mass, or continuity equation in laminar flow (in an inertial reference frame), can be written as equation 3.2.

Many important engineering flows involve swirl or rotation and FLUENT is well-equipped to model such flows. Swirling flows are common in combustion, with swirl introduced in burners and combustors in order to increase residence time and stabilize the flow pattern. Rotating flows are also encountered in turbomachinery, mixing tanks, and a variety of other applications. If the flow involves a rotating boundary, which moves through the fluid (e.g., an impeller blade or a grooved or notched surface), it will be needed to use a rotating reference frame to model the problem. If there is more than one rotating boundary, it can be used multiple reference frames or mixing planes.

When a impeller blade passes by an upstream diffuser blade, it experiences a fluctuation in the fluid forces that act upon it. Considerable effort has already been invested in studying these interactions between the flow within the impeller and the flow between the diffuser vanes of centrifugal pump.

Impeller-diffuser interaction problems (as illustrated in figure 3.18) cannot be modeled by a simple coordinate transformation to a rotating reference frame. Applying the Multiple Reference Frame (MRF) and Sliding Mesh approach must treat impeller-diffuser interaction.

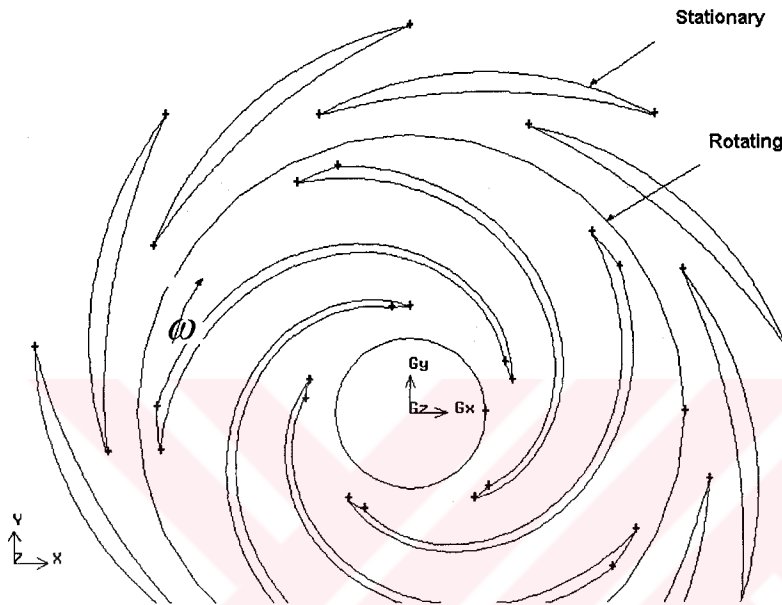


Figure 3.18 The rotating and stationary parts in the pump

When the equations of motion are solved in a rotating frame of reference, the acceleration of the fluid is augmented by additional terms that appear in the momentum equations. FLUENT allows you to solve rotating frame problems using the absolute velocity, $\vec{v}$, or the relative velocity, " $\vec{v}_r$ ", as the dependent variable. The two velocities are related by the following equation:

$$\vec{v}_r = \vec{v} - (\vec{\Omega} \times \vec{r}) \quad (3.9)$$

Here, $\vec{\Omega}$ is the angular velocity vector (that is, the angular velocity of the rotating frame) and $\vec{r}$ is the position vector in the rotating frame.

The left hand side of the momentum equations appears as follows for an inertial reference frame:

$$\frac{\partial}{\partial t}(\rho \vec{v}) + \nabla \cdot (\rho \vec{v} \vec{v}) \quad (3.10)$$

For a rotating reference frame, the left hand side written in terms of absolute velocities becomes

$$\frac{\partial}{\partial t}(\rho \vec{v}) + \nabla \cdot (\rho \vec{v}_r \vec{v}) + (\vec{\Omega} \times \vec{v}) \quad (3.11)$$

In terms of relative velocities the left-hand side is given by

$$\frac{\partial}{\partial t}(\rho \vec{v}_r) + \nabla \cdot (\rho \vec{v}_r \vec{v}_r) + \rho(2 \vec{\Omega} \times \vec{v}_r + \vec{\Omega} \times \vec{\Omega} \times \vec{r}) + \rho(\frac{\partial \vec{\Omega}}{\partial t} \times \vec{r}) \quad (3.12)$$

where $2 \vec{\Omega} \times \vec{v}_r + \vec{\Omega} \times \vec{\Omega} \times \vec{r}$ is the Coriolis force. The using solvers (FLUENT) neglect the $\rho \frac{\partial \vec{\Omega}}{\partial t} \times \vec{r}$ term, so it cannot accurately model a time-varying angular velocity using the relative velocity formulation.

For flows in rotating domains, the equation for conservation of mass, or continuity equation, can be written as in Equation 3.2 for both the absolute and the relative velocity formulations.

3.5.4 Turbulent Modeling and RNG $k-\varepsilon$ Model

Turbulent flows are characterized by fluctuating velocity fields. These fluctuations mix transported quantities such as momentum, energy, and species concentration, and cause the transported quantities to fluctuate as well. Since these

fluctuations can be of small scale and high frequency, they are too computationally expensive to simulate directly in practical engineering calculations. Instead, the instantaneous (exact) governing equations can be time-averaged, ensemble-averaged, or otherwise manipulated to remove the small scales, resulting in a modified set of equations that are computationally less expensive to solve. However, the modified equations contain additional unknown variables, and turbulence models are needed to determine these variables in terms of known quantities. CFD programs provide the choices of turbulence models; Spalart-Allmaras Model, $k-\varepsilon$ models, $k-\omega$ models, Reynolds Stress Model (RSM), LES model etc.

It is an unfortunate fact that no single turbulence model is universally accepted as being superior for all classes of problems. The choice of turbulence model will depend on considerations such as the physics encompassed in the flow, the established practice for a specific class of problem, the level of accuracy required, the available computational resources, and the amount of time available for the simulation.

Successful computations of turbulent flows require some consideration during the mesh generation. Since turbulence (through the spatially-varying effective viscosity) plays a dominant role in the transport of mean momentum and other scalars for the majority of complex turbulent flows, it must be as certain that turbulence quantities are properly resolved, if high accuracy is required. Due to the strong interaction of the mean flow and turbulence, the numerical results for turbulent flows tend to be more susceptible to grid dependency than those for laminar flows. It is therefore recommended that the problem is resolved; with sufficiently fine meshes, the regions where the mean flow changes rapidly and there are shear layers with a large mean rate of strain.

The $k-\varepsilon$ models have similar forms, with transport equations for $k-\varepsilon$. The major differences in the models are as follows: *i)* the method of calculating turbulent viscosity, *ii)* the turbulent Prandtl numbers governing the turbulent diffusion of $k-\varepsilon$, and *iii)* the generation and destruction terms in the ε equation. The transport equations, methods of calculating turbulent viscosity, and model constants are presented separately for each model. The features that are essentially

common to all models follow, including turbulent production, generation due to buoyancy, accounting for the effects of compressibility, and modeling heat and mass transfer.

The standard $k-\varepsilon$ model is a semi-empirical model based on model transport equations for the turbulence kinetic energy (k) and its dissipation rate (ε). The standard $k-\varepsilon$ model in FLUENT falls within this class of turbulence model and has become the workhorse of practical engineering flow calculations in the time since it was proposed by Launder and Spalding. As the strengths and weaknesses of the standard $k-\varepsilon$ model have become known, improvements have been made to the model to improve its performance. Two of these variants are available in FLUENT; the RNG $k-\varepsilon$ model and the realizable $k-\varepsilon$ model. In this study, the RNG $k-\varepsilon$ model was used.

The RNG $k-\varepsilon$ model was derived using a rigorous statistical technique (called renormalization group theory). It is similar in form to the standard $k-\varepsilon$ model, but includes the following refinements:

- The RNG model has an additional term in its ε equation that significantly improves the accuracy for rapidly strained flows.
- The effect of swirl on turbulence is included in the RNG model, enhancing accuracy for swirling flows.
- The RNG theory provides an analytical formula for turbulent Prandtl numbers, while the standard $k-\varepsilon$ model uses user-specified, constant values.
- While the standard $k-\varepsilon$ model is a high-Reynolds-number model, the RNG theory provides an analytically derived differential formula for effective viscosity that accounts for low-Reynolds-number effects. Effective use of this feature does, however, depend on an appropriate treatment of the near-wall region.

These features make the RNG $k-\varepsilon$ model more accurate and reliable for a wider class of flows than the standard $k-\varepsilon$ model.

The RNG-based $k-\varepsilon$ turbulence model is derived from the instantaneous NSEs, using a mathematical technique called RNG methods. The analytical

derivation results in a model with constants different from those in the standard $k - \varepsilon$ model, and additional terms and functions in the transport equations for k and ε .

Transport equations in the RNG $k - \varepsilon$ model has a form of

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}(\alpha k \mu_{eff} \frac{\partial k}{\partial x_j}) + G_k + G_b - \rho \varepsilon - Y_M + S_k \quad (3.13)$$

and

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_j}(\alpha \varepsilon \mu_{eff} \frac{\partial \varepsilon}{\partial x_j}) + C_{1\varepsilon} \frac{\varepsilon}{k} + (G_k + C_{3\varepsilon} G_b) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} - R_\varepsilon - S_\varepsilon \quad (3.14)$$

In these equations, G_k represents the generation of turbulence kinetic energy due to the mean velocity gradients, G_b is the generation of turbulence kinetic energy due to buoyancy. Y_M represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate. The quantities α_k and α_ε are the inverse effective Prandtl numbers for k and ε , respectively. S_k and S_ε are user-defined source terms.

The model constants $C_{1\varepsilon}$ and $C_{2\varepsilon}$ have values derived analytically by the RNG theory and these values used $C_{1\varepsilon} = 1.42$ $C_{2\varepsilon} = 1.68$ in these studies.

3.6 Boundary Conditions

Boundary conditions specify the flow and thermal variables on the boundaries of the physical model. They are, therefore, a critical component of FLUENT simulations and it is important that they are specified appropriately. The modeled boundary conditions are the ones considered with more physical meaning for turbomachinery flow simulations. For all the definitions of the cylindrical coordinate system, positive radial velocities point radial out from the rotation axis, positive axial velocities are in the direction of the rotation axis vector, and positive tangential

velocities are based on the right-hand rule using the positive rotation axis. The flow rate is changed by modifying the static pressure to kinetic energy ratio at the outlet condition, which simulates different closing positions of a valve. Since the computational domain incorporated the entire stage of the diffuser pump, the boundary conditions were specified at the inlet and outlet of the pump. The boundary conditions were specified as follows:

An important step in the setup of the model is the definition of the physical properties of the material. For solid materials, only density, thermal conductivity, and heat capacity are defined. It can be specified a constant value, a temperature-dependent function, or a user-defined function for thermal conductivity; a constant value or temperature dependent function for heat capacity; and a constant value for density. In FLUENT, physical properties of fluids and solids are associated with named “materials”, and these materials are then assigned as boundary conditions for zones. In this study, it is assumed that water has constant density, temperature and viscosity.

3.6.1 Solid and Fluid Conditions

A “solid” zone is a group of cells for which only a heat conduction problem is solved; no flow equations are solved. The material being treated as a solid may actually be a fluid, but it is assumed that no convection is taking place. The only required input for a solid zone is the type of solid material. It must be indicated which material the solid zone contains so that the appropriate material properties will be used. For the solid zone can also be defined motion. If there are rotationally periodic boundaries adjacent to the solid zone, you will need to specify the rotation axis.

A fluid zone is a group of cells for which all active equations are solved. The only required input for a fluid zone is the type of fluid zone is the type of fluid material. For working fluid, it was accepted the properties of water 20°C. It must be indicate which material the fluid zone contains so that the appropriate material properties will be used. If there are rotationally periodic boundaries adjacent to the

fluid zone or if the zone is rotating, it must be specify the rotation axis. For rotational parts angular velocity was taken as constant and 890 rpm clockwise in this study.

3.6.2 Flow Inlets and Exits

FLUENT has a wide range of boundary conditions that permit flow to enter and exit the solution domain. Fluent provides different types of boundary zone types for the specifications of flow inlets and exits. This velocity specify flow rate.

3.6.2.1 Constant velocity inlet boundary conditions

Velocity inlet boundary conditions are used to define the flow velocity, along with all relevant scalar properties of the flow, at flow inlets. At the inlet, a steady, uniform, axial velocity distribution is imposed. The total (or stagnation) properties of the flow are not fixed, so they will rise to whatever value is necessary to provide the prescribed velocity distribution.

This boundary condition is intended for incompressible flows. Flows will lead to a nonphysical result because it allows stagnation conditions to oat to any level. It should also be careful not to place a velocity inlet too close to a solid obstruction, since this could cause the inflow stagnation properties to become highly non-uniform.

The inflow velocity can be defined by specifying the velocity magnitude and direction, the velocity components, or the velocity magnitude normal to the boundary. If the cell zone adjacent to the velocity inlet is moving (i.e., if a rotation reference frame is used, multiple reference frames, or sliding meshes), it can be specified either relative or absolute velocities. For axisymmetric problems with swirl, it will also be specify the swirl velocity.

FLUENT uses the boundary condition inputs at velocity inlets to compute the mass flow into the domain through the inlet and to compute the fluxes of momentum, energy, and species through the inlet. This section describes these calculations for the case of flow entering the domain through the velocity inlet

boundary and for the less common case of flow exiting the domain through the velocity inlet boundary. When the velocity inlet boundary condition defines flow entering the physical domain of the model, FLUENT uses both the velocity components and the scalar quantities that it is defined as boundary conditions to compute the inlet mass flow rate, momentum fluxes, and fluxes of energy and chemical species. The mass flow rate entering a fluid cell adjacent to a velocity inlet boundary is computed as

$$\dot{m} = \int \rho \vec{v} \cdot d\vec{A} \quad (3.15)$$

Note that only the velocity component normal to the control volume face contributes to the inlet mass flow rate.

3.6.2.2 Transient velocity inlet boundary conditions

Most user-defined functions access data from the FLUENT solver. Since solver data is defined in terms of grid components. Most conditions at each type of boundary zone can be defined as profile functions instead of constant values. It can be used a profile contained in an externally generated boundary profile file, or a function that you create using a User-Defined Function (UDF).

Cells and cell faces are grouped into zones that generally define physical components of the domain (e.g., inlets, outlets, walls). When using a UDF, the UDF can access solution variables at individual cells or cell faces in the fluid and boundary zones. UDFs need to be passed appropriate arguments such as a zone reference and cell ID in order to allow the individual cells to be identified. A zone is a group of cells or cell faces that can be identified using physical features of the model and domain (e.g., inlets, outlets, walls, fluid regions). For example, a set of cell faces designated as a face zone can be assigned the type velocity-inlet whereby velocity data can be assigned. A thread is the internal name of the data structure that is used to represent a zone. The Thread structure acts as a container for data that is common to the group of cells or faces that it represents.

A UDF, is a function that the program can be dynamically loaded with the FLUENT solver to enhance the standard features of the code. UDFs are written in the C programming language. UDFs are executed as either interpreted or compiled functions. Values that are passed to the solver by a UDF or returned by the solver to a UDF must be specified in SI units. User-defined functions can perform a variety of tasks.

A different inlet boundary condition needs for unsteady flow state. This condition can be obtained using UDF. A temporarily periodic velocity boundary condition is been applied to the inlet using the UDF. The UDF is a function programmed that dynamically linked with the solver to enhance the standard features of the code. UDFs can be implemented as interpreted or compiled functions. In this study used code is produced as interpreted. Interpreted UDFs are read in and interpreted at run time. The velocity used in UDF has the form

$$v_x = v_0 + A \sin (\omega t) \quad (3.16)$$

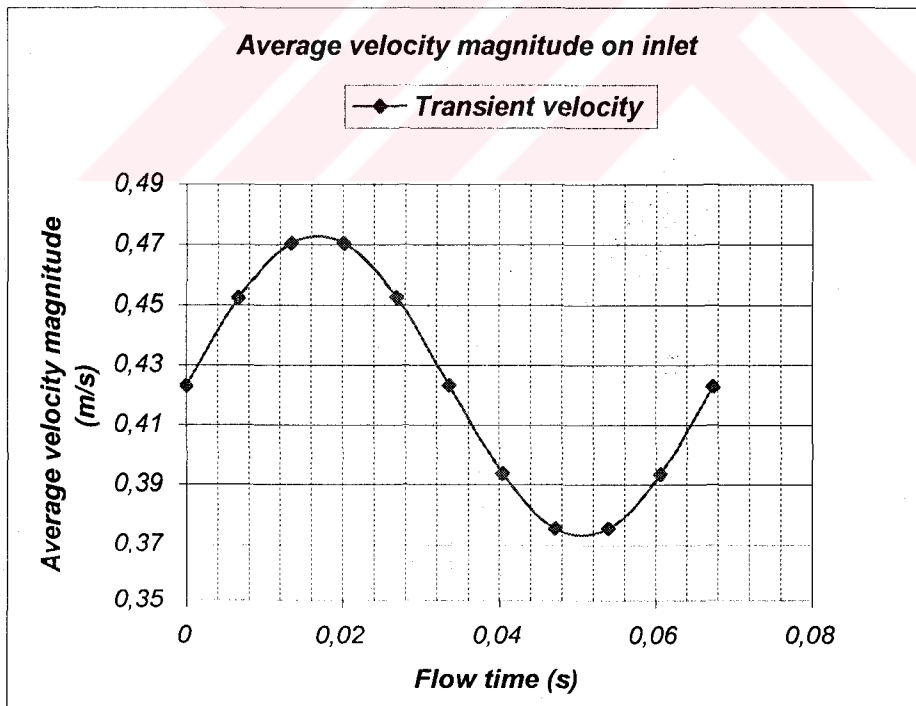
where ω is angular velocity, v_x is velocity at anytime t , v_0 is the calculated velocity according to different mass flow rate, and A constant which ensure the change of velocity according to time is accepted as 0.05 (figure 3.19). The source code for velocity profile UDF is written in the C programming language. The code written for UDF in this study was given in figure 3.19a, and the average velocity magnitude as a function of flow time for 3.8 l/s mass flow rate is given in figure 3.19b. Figure 3.19b nicely illustrates that the velocity oscillates around the equilibrium value, 0.423 m/s for 3D flow, with amplitude of 0.05 m/s, as accepted.

```

/*unsteady.c UDF for specifying a transient velocity profile b.c.*/
#include "udf.h"
DEFINE_PROFILE(unsteady_velocity, thread, position)
{
    face_t f;
    begin_f_loop(f, thread)
    {
        real t = RP_Get_Real("flow-time");
        F_PROFILE(f, thread, position) = (0.423+0.05*sin(93.15*t));;
    }
    end_f_loop(f, thread)
}

```

a)



b)

Figure 3.19 a) The C code of velocity profile b) Average velocity magnitude on inlet (for one cycle)

3.6.3 Pressure Outlet Boundary Conditions

Pressure outlet boundary conditions require the specification of a static (gauge) pressure at the outlet boundary. At the outlet constant static pressure is maintained. The value of static pressure specified is used only while the flow is subsonic. Should the flow become locally supersonic, the specified pressure is no longer used; pressure will be extrapolated from the flow in the interior. All other flow quantities are extrapolated from the interior. A set of “backflow” conditions is also specified to be used if the flow reverses direction at the pressure outlet boundary during the solution process. Convergence difficulties will be minimized if you specify realistic values for the backflow quantities. In this study, outlet pressure was selected as boundary condition at outlet and set to the 60 kPa. Outlet pressure was taken as constant, flow rate was changed, as a results inlet pressure was calculated.

3.6.4 Wall Boundary Conditions

Wall boundary conditions are used to bound fluid and solid regions. In viscous flows, the no-slip boundary condition is enforced at walls by default, but you can specify a tangential velocity component in terms of the translational or rotational motion of the wall boundary, or model a “slip” wall by specifying shear. Wall boundaries can be either stationary or moving. The stationary boundary condition specifies a fixed wall, whereas the moving boundary condition can be used to specify the translational or rotational velocity of the wall, or the velocity components. Note that all moving walls are no-slip; the other shear conditions are relevant only for stationary walls. The no-slip condition is the default, and it indicates that the fluid sticks to the wall and moves with the same velocity as the wall, if it is moving.

Non-slip boundary conditions have been imposed over the impeller blades, diffuser vanes and walls, the volute casing and the inlet pipe wall. In the grid interface, inlet-impeller and impeller-volute the intersection between the interface zones is determined at each new time step. Fluxes across the grid interface are computed using the faces resulting from the intersection of the two interfaces zones.

In this study, it is supposed that angular velocity of impeller is 890 rpm and other solid parts are stationary.

3.7 Structure of a CFD Code

The Physical aspects of any fluid flow are governed by three fundamental principles: conservation of mass; Newton's second law and conservation of energy. These fundamental principles can be expressed in terms of mathematical equations, which in their most general form are usually partial differential equations. CFD is the science of determining a numerical solution to the governing equations of fluid flow whilst advancing the solution through space or time to obtain a numerical description of the complete flow field of interest.

The governing equations for Newtonian fluid dynamics, the unsteady NSEs, have been known for over a century. However, the analytical investigation of reduced forms of these equations is still an active area of research as is the problem of turbulent closure for the Reynolds averaged form of the equations.

CFD is very powerful and spans a wide range of industrial and non-industrial application areas (aerodynamics of aircraft and vehicles, turbomachinery, hydrodynamics of ships, power plant, hydrology and oceanography, biomedical engineering etc.). Increasingly CFD is becoming a vital component in the design of industrial products and processes.

CFD codes are structured around the numerical algorithms that can tackle fluid flow problems. In order to provide easy access to their solving power all commercial CFD packages include sophisticated user interfaces to input problem parameters and to examine the results. A CFD code has three basic components: (i) a pre-processor, (ii) a solver and (iii) a post-processor.

The solver is the heart of the code, carrying out the major computations and providing the numerical solutions. Once a grid has been read into the program, all remaining operations are performed within the solver. This includes setting boundary conditions, defining the fluid properties, executing the solution refining the grid. The solver consists essentially of two components that provide a discretization for the

defining equations and subsequent solution. The first component uses a discretizing scheme to express the governing equations in a discretized form for all the mesh/grid elements over the whole computational domain and discretizes the boundary equations appropriate at the boundary elements. In summary, this section converts the partial differential equations and boundary condition formulae into a group of algebraic equations. The second part uses an iterative procedure to find solutions that satisfy these boundary conditions for the algebraic equations defining the flow domain. A solver in any CFD code is based on one of the available discretization methods. The detailed information will be given in 3.4.8 part. As FVM has been very well established and thoroughly validated, it is applied in most commercial CFD packages used worldwide. FLUENT solver uses finite volume method. The finite volume method was originally developed as a special finite difference formulation.

The pre-processor and post-processor are the front and end of the code, providing the user/machine interface that allows a CFD operator to communicate with the solver: inputting data to define the problem to be simulated and commanding the solver to use certain models and schemes to carry out the simulation and, finally, presenting for study the computed results. The pre-processor has a number of functions that allow users to define a fluid domain, known as the computational domain, and build up the physical geometry of the zone considered, creating a mesh/grid system throughout the domain and tuning this mesh to improve computation quality, including accuracy and speed. The user may also specify the properties of the fluid and other materials in contact with the fluid at this stage. In this study, the meshes and geometries required for the calculations were generated using the CFD pre-processing package, GAMBIT 2.0 and some CAD programs.

The post-processor allows the CFD operator to construct a picture of the simulated flow problem by displaying the geometry of the problem, the computational domain and the grid system. The post-processor may then display contour or iso-surface plots for the flow variables; including contours plotted over specified surfaces, such as an a solid/fluid interface or a iso-surface of a second flow variable. In many cases velocity vector plotting is important, as is streamline presentation or particle tracking. In some cases animation of the fluid flow or a flow

process may be appropriate and, finally, it may be desirable to provide hardcopy printouts. In this study, FLUENT program was used as post-precessor.

Apart from these key elements, a commercial package aimed at multi-purpose modeling will have a suite of models for various flow problems, such as various turbulence models to cover a range of turbulence conditions and assumptions. The packages will also have a library of material properties for defining the fluid media and solid boundaries in the computational domain. Experience will guide the user in the choice of appropriate model and boundary condition (Versteeg and Malalasekera (1995) and Douglas et al. (2000)).

3.8 Numerical Schemes

A complete CFD simulation often requires repeating the procedure a number of times. It is not rare to run a large number of trials before reaching a set of reasonable solutions. The procedure includes tuning the mesh system, adjusting boundary conditions, selecting numerical parameters, finding the right physical model, monitoring iteration and, finally, viewing the results. This again highlights the need for experience within the application of CFD code.

A good understanding of the numerical solution algorithm is also crucial. Three mathematical concepts are useful in determining the success or otherwise of such algorithms: convergence, consistency, and stability. Convergence is the property of a numerical method to produce a solution, which approaches the exact solution as the grid spacing; control volume size or element size is reduced to zero. Consistent numerical schemes produce systems of algebraic equations, which can be demonstrated to be equivalent to the original governing equation, as the grid spacing tends to zero. Stability is associated with damping of errors as the numerical method proceeds. If a technique is not stable even round of errors as the initial data can cause wild oscillations or divergence.

Conservativeness, boundedness and transportiveness are designed into all finite volume schemes and have been widely shown to lead to successful CFD

simulations. Therefore, they are now commonly accepted as alternatives for the more mathematically rigorous concepts of convergence, consistency and stability.

FLUENT allows choosing either of two numerical methods: (i) segregated solver (ii) coupled (implicit and explicit) solver. All solvers formulations will provide accurate results for a broad range of flows, but in some cases one formulation may perform better (i.e., yield a solution more quickly) than the others. The segregated and coupled approaches differ in the way that the continuity, momentum, and (where appropriate) energy and species equations are solved: the segregated solver solves these equations sequentially (i.e., segregated from one another), while the coupled solver solves them simultaneously (i.e., coupled together). Both formulations solve the equations for additional scalars (e.g., turbulence or radiation quantities) sequentially. The implicit and explicit coupled solvers differ in the way that they linearize the coupled equations. The segregated solver traditionally has been used for incompressible and mildly compressible flows. The coupled approach, on the other hand, was originally designed for high-speed compressible flows. Both approaches are now applicable to a broad range of flows (from incompressible to highly compressible), but the origins of the coupled formulation may give it a performance advantage over the segregated solver for high-speed compressible flows.

In this study, the segregated solution approach was used. Using this approach, the governing equations are solved sequentially (i.e., segregated from one another). Because the governing equations are non-linear (and coupled), several iterations of the solution loop must be performed before a converged solution is obtained.

In the Segregated solution method, each iteration consists of the steps illustrated in figure 3.20 and outlined below:

1. Fluid properties are updated, based on the current solution (if the calculation has just began, the fluid properties will be updated based on the initialized solution).

2. The u , v , and w momentum equations are each solved in turn using current values for pressure and face mass fluxes, in order to update the velocity field.

3. Since the velocities obtained in step 2 may not satisfy the continuity equation locally, a “Poisson-type” equation for the pressure correction is derived from the continuity equation and the linearized momentum equations. This pressure correction equation is then solved to obtain the necessary corrections to the pressure and velocity fields and the face mass fluxes such that continuity is satisfied.

4. Where appropriate, equations for scalars such as turbulence, energy, species, and radiation are solved using the previously updated values of the other variables.

5. When interphase coupling is to be included, the source terms in the appropriate continuous phase equations may be updated with a discrete phase trajectory calculation.

6. A check for convergence of the equation set is made. These steps are continued until the convergence criteria are met.

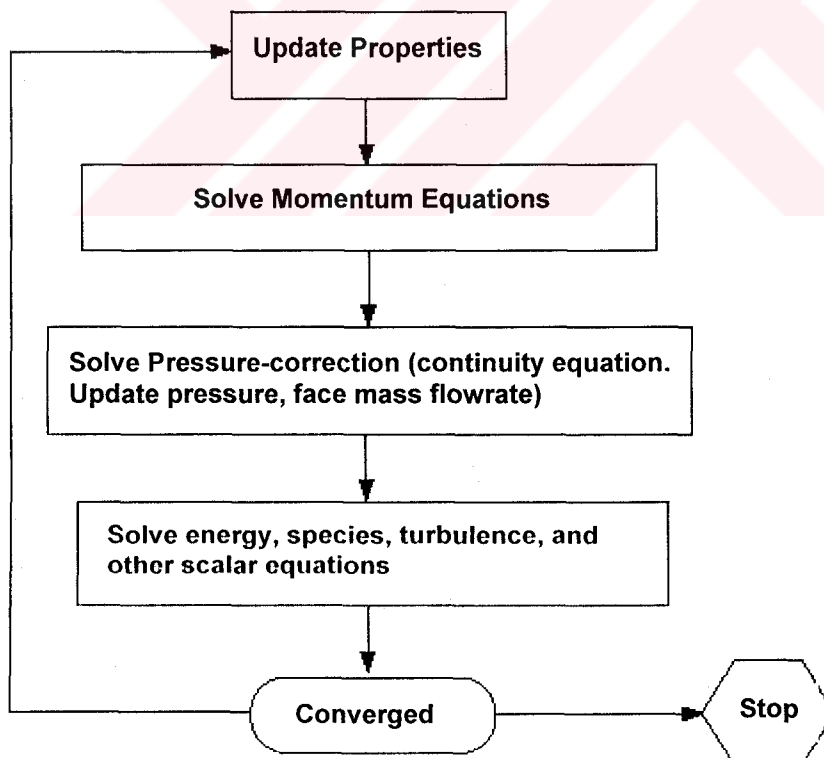


Figure 3.20 Overview of the Segregated Solution Method

In both the segregated and coupled solution methods the discrete, non-linear governing equations are linearized to produce a system of equations for the dependent variables in every computational cell. The resultant linear system is then solved to yield an updated flow-field solution.

The manner in which the governing equations are linearized may take an “implicit” or “explicit” form with respect to the dependent variable (or set of variables) of interest.

In the implicit form, for a given variable, the unknown value in each cell is computed using a relation that includes both existing and unknown values from neighboring cells. Therefore each unknown will appear in more than one equation in the system, and these equations must be solved simultaneously to give the unknown quantities.

Also, in the explicit form, for a given variable, the unknown value in each cell is computed using a relation that includes only existing values. Therefore each unknown will appear in only one equation in the system and the equations for the unknown value in each cell can be solved one at a time to give the unknown quantities.

3.9 Discretization Methods

FLUENT uses internal data structures to define the domain(s) of the mesh, assign an order to cells, cell faces, and grid points in a mesh, and establish connectivity between adjacent cells. A thread is the internal name of the data structure in FLUENT that is used to represent a (boundary or cell) zone. Cell threads are groupings of cells, and face threads are groupings of faces. A domain is the internal name of the data structure in FLUENT that is used to represent a grouping of node, face, and cell threads in a mesh. Cells and cell faces are grouped into zones that typically define the physical components of the model (e.g., inlets, outlets, walls, fluid regions).

The numerical solution of the differential equations governing a complex fluid dynamics problem requires the introduction of a discretization method. Several

methods have been developed and are currently in use. Discretization methods are FDM, FEM and FVM. FDM is the oldest method applied to the numerical solution of differential equations, and its development is based on the definition of the derivative and the properties of Taylor's series.

FEM, with their capability of handling intricate geometries of practical interest, has become a significant area of CFD. The basic FEM procedure can be outlined; *i)* Conversion to a weak formulation, *ii)* Selection of the function spaces, and *iii)* Solution of the discretized equations.

The FVM of solving the fluid dynamic equations is derived from the integral form of the conservation law equations. The net flux is composed generally of convective; shear stress and heat transfer terms. Using this integral approximation, numerical formulations can be determined by arbitrary divisions of the computational domain into discrete cells. The computational cell in a generalized coordinate transformation, or structured-grid, approach corresponds to a hexahedral (quadrilateral) cell in the three (two) dimensions defined by coordinates of the constant generalized coordinates. The FVM is equivalent to a FDM solution the strong conservation law of form the governing equations, subject to proper interpretation of the metric derivatives.

The discretization procedure consists of the integration of the conservation equations over each control volume assuming uniform values of the variables over the cell. This results in a set of algebraic equations, which relate the values of the variables at discrete points (assumed to be the centers of the cells) and at the faces of the cell, the fluxes across the faces of the cell and the dimensions of the cell. It is this set of algebraic equations which is actually solved by the code. The values of the variables at the centers of the cells become the unknowns. The values of the variables at the faces of the cells need to be interpolated between the values at the centers of two adjacent cells. This interpolation procedure is an important part of the solution scheme and is discussed in the following section.

FLUENT uses a control-volume-based technique to convert the governing equations to algebraic equations that can be solved numerically. This control volume technique consists of integrating the governing equations about each control volume,

yielding discrete equations that conserve each quantity on a control-volume basis. Discretization of the governing equations can be illustrated most easily by considering the steady-state conservation equation for transport of a scalar quantity. This is demonstrated by the following equation written in integral form for an arbitrary control volume V as follows:

$$\oint \rho \phi \vec{v} \cdot d\vec{A} = \oint \Gamma_\phi \nabla \phi \cdot d\vec{A} + \int_V S_\phi dV \quad (3.17)$$

where ρ = density, $\vec{v}$ = velocity vector, $d\vec{A}$ = surface area vector, Γ_ϕ = diffusion coefficient for ϕ , $\nabla \phi$ = gradient of ϕ , S_ϕ = source of ϕ per unit volume.

Equation 3.17 is applied to each control volume, or cell, in the computational domain. The 2-D, triangular cell shown in Figure 3.21 is an example of such a control volume. Discretization of equation (3.17) on a given cell yields,

$$\sum_f^{N_{faces}} \rho_f \vec{v}_f \cdot \vec{A}_f = \sum_f^{N_{faces}} \Gamma_\phi (\nabla \phi)_n \cdot \vec{A}_f + S_\phi V \quad (3.18)$$

where N_{faces} = number of faces enclosing cell, ϕ_f = value of ϕ convected through face f , $\rho_f \vec{v}_f \cdot \vec{A}_f$ = mass flux through the face, $\vec{A}_f$ = area of face f , V = cell volume, $(\nabla \phi)_n$ = magnitude of $\nabla \phi$ normal to face f .

The equations solved by FLUENT take the same general form as the one given above and apply readily to multi-dimensional, unstructured meshes composed of arbitrary polyhedra. A face will bound either one or two cells (identified by $c0$ and $c1$) depending on whether it is a boundary face or an interior face. The cells on either side of a face may or may not belong to the same cell thread. If a face is on the boundary of a domain, then only $c0$ exists. ($c1$ is undefined for an external face). If the face is in the interior of the domain, then both $c0$ and $c1$ exist.

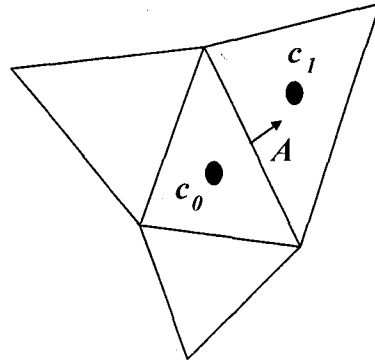


Figure 3.21 Control volume used to illustrate discretization of a scalar transport equation

By default, FLUENT stores discrete values of the scalar ϕ at the cell centers (c_0 and c_1 in figure 3.21). However, face values ϕ_f are required for the convection terms in equation 3.18 and must be interpolated from the cell center values. This is accomplished using an upwind scheme. Upwinding means that the face value ϕ_f is derived from quantities in the cell upstream, or “upwind”, relative to the direction of the normal velocity v_n in equation 3.18. FLUENT allow choosing from several upwind schemes: first-order upwind, second-order upwind, power law, and QUICK. When second-order accuracy is desired, quantities at cell faces are computed using a multidimensional linear reconstruction approach. In this approach, higher-order accuracy is achieved at cell faces through a Taylor series expansion of the cell-centered solution about the cell centroid.

The three momentum equations and the continuity equation provide four equations to determine the three components of the velocity vector and the pressure. The way in which the pressure appears in the equations after discretization, however, prevents a direct solution being obtained. This is because the pressure appears evaluated not at the nodes, but at the cell faces and simple interpolation can lead to the well-known checkerboard problem. SIMPLE or SIMPLEC algorithms are used in which an equation is derived for a pressure correction term to replace the continuity

equation. This equation is then used to update the (guessed) pressure field and this updated pressure is in turn used to update the velocity field when the momentum equations are solved. The difference between the algorithms just mentioned is the neglect of a term that relates the correction term at the node under consideration and the correction terms at the neighbor nodes. These algorithms improve convergence.

Pressure-velocity coupling is achieved to derive an equation for pressure from the discrete continuity equation. Solvers provide the option to choose among three pressure-velocity coupling algorithms: SIMPLE, SIMPLEC, and PISO. The SIMPLE algorithm uses a relationship between velocity and pressure corrections to enforce mass conservation and to obtain the pressure field. A number of variants of the basic SIMPLE algorithm are available in the literature. In addition to SIMPLE, FLUENT offers the SIMPLEC (SIMPLE-Consistent) algorithm. SIMPLE is the default, but many problems will benefit from the use of SIMPLEC. The SIMPLEC procedure is similar to the SIMPLE procedure. A finite volume numerical method is presented for simulating turbulent flows through a centrifugal pump impeller. The SIMPLEC algorithm is utilized for solving governing equations of incompressible viscous/turbulent flows through the pump impeller. The RNG $k-\varepsilon$ turbulence model is adopted to describe the turbulent flow process. In this study, second-order accuracy upwind scheme and SIMPLEC algorithm is used for discretization.

The actual iterative solution procedure consists of solving in turn each of the conservation equations, starting with the pressure correction equation, then the velocity equations followed by the scalar equations for enthalpy and species. Note that the equations are not solved simultaneously at all nodes. The line-by-line solution procedure is employed.

3.10 Numerical Solution Control

During the solution process, checking residuals, statistics, force values, surface integrals, and volume integrals monitored convergence. Each of these monitoring features is described below. At the end of each solver iteration, the

residual sum for each of the conserved variables is computed and stored, thus recording the convergence history. This history is also saved in the data file.

The number of iterations has been adjusted to reduce the residual below an acceptable value in each time step. In particular, the ratio between the sum of the residuals and the sum of the fluxes for a given variable in all the cells is reduced to the desired value. In this study, mass flow rate was especially inspected. Although the mass flow rate history the solution is converged. It should be also check the mass fluxes the domain to ensure that mass is being conserved. The net mass imbalance should be a small fraction of the total flux through the system. The-time step used in the unsteady calculation has been $6.74\text{e-}03$ seconds. The impeller grid movement is related with this time step and the rotational speed, so a complete revolution is performed each 10 time-steps. The iterations number has been adjusted to reduce the residual below an acceptable value in each time-step. Initializing the unsteady calculation with the solution, over 5 impeller revolutions are necessary to achieve the periodic unsteady solution converge. The numerical results dependence on the grid size has been tested with a lower number of cells. The overall performance of the pump is about the same with less than one-third cells. Flow main characteristics do not change. But, the more cells are selected to be observed the more details.

4. RESULTS AND DISCUSSION

In this study, the experimental results obtained by Sinha were compared with CFD results and then numerical analysis performed for different radial gap geometries. Particle Image Velocimetry measurements are used to identify the unsteady flow structures and turbulence in a centrifugal pump with a vaned diffuser. Sinha (1999) measured pressures at 20 volume flow rates in order to obtain the performance curves. Initially, the flow was assumed to be 2D for numerical calculations. But, the numerical results of 2D flow were not presented in the present work. Later, the flow was assumed to be 3D for the numerical studies. Consequently, Comparisons of performance curves indicate that the accuracy of 3D numerical calculations is higher than 2D numerical calculations. Thus, in the following phase the calculations are concentrated on 3D studies. For this reason, the results of 2D studies have not been given in this thesis. In this numerical study, performance and efficiency curves are obtained at 15 different volume flow rates.

Numerical studies were performed for the volume flow rate in the range of $1.31 \leq Q \leq 5.68$ l/s but, as an experimental study the range of volume flow rate was $1.31 \leq Q \leq 7.4$ l/s. It can be seen that experimental studies correspond a higher range of volume flow rate. Since numerical studies does not give an appropriate results above the value of 5.68 l/s volume flow rates numerical $H-Q$ and $\eta-Q$ curves were only compared with experimental results in a given ranges. It was seen that the experiment and numerical performance curves fit in the range of low flow rates for three radial gap ratios. On the other hand, in the case of higher level of volume flow rate differences between numerical and experimental results increase gradually when the level of flow rates increase. The calculated results was used to obtain the performance and efficiency curves, including that the higher level flow rates, but these results are not presented in this thesis since it will occupy much more space to give all these results. The solution gives a great number of data. Pressure, velocity, turbulent kinetic energy, etc. could be obtained at any point of geometry. The design point value was obtained as 4.12 l/s from numerical solution. In addition, the following investigations were performed at three different gap ratios. The pressure

contours, velocity contours, vorticity and turbulent kinetic energy are presented for design and off design condition, (below and above design points) corresponding to 4.12, 3.48 and 4.74 l/s respectively. When the experimental and numerical design point is taken into consideration, it is seen that the below and above design flow conditions are chosen for the purpose of doing suitable comparisons on off-design flow phenomena. On the other hand, these operating conditions are very close to the point of maximum efficiency.

Steady and unsteady performance curves obtained at three different gap ratios were compared with each other in order to demonstrate the validity of steady results. For steady and unsteady calculations were used geometries and meshes at three different radial gap ratios.

Sinha (1999) investigated impeller-diffuser interaction with respect to the blade passing frequency around the diffuser passage, but hasn't searched for the effect of different radial gap clearance; just apart from diffuser gap space has 20% of impeller radius. This pump is different from the traditional pump with respect to radial gap and volute thicknesses.

In this study, the case of different the radial gap ratios was investigated and also the effects of smaller gaps on pump performance and flow structure were studied. To demonstrate the validity of numerical study, primarily PIV results obtained at 5 different volume flow rates around tongue region by Sinha (1999) were compared with the numerical results obtained presently at the same flow rates for the radial gap ratio of Case I. In progressing phases, the effects of different radial gap ratios and volume flow rates over pressure fluctuations were investigated with respect to angle between impeller-diffuser, and after diffuser, as studied by Gonzalez et al. (2002b, 2003a and 2003b)

Firstly, two regions were considered as suitable at determining pressure fluctuations (Guleren and Pinarbasi (2004) and Gonzalez et al. (2002 a)). The pressure calculations were made along the circle which lies in the middle of impeller and diffuser per 10° steps on the circumferential position. On the other hand, for the pressure measurements after diffusers, it was accepted that 1.06 times of the diffuser outlet diameter is suitable (Gonzalez et al. (2002 a)). It is also made for 10° steps on

the circumference after the diffusers. These calculations are made on the plane which passes through the middle of volute thickness. The distance in between the plane and volute walls is 4 mm. Then, the effects of different radial gaps on pressure fluctuations was investigated within the diffuser (A, B, C, D, E, F, G, H and I points), as seen in figure 4.1.

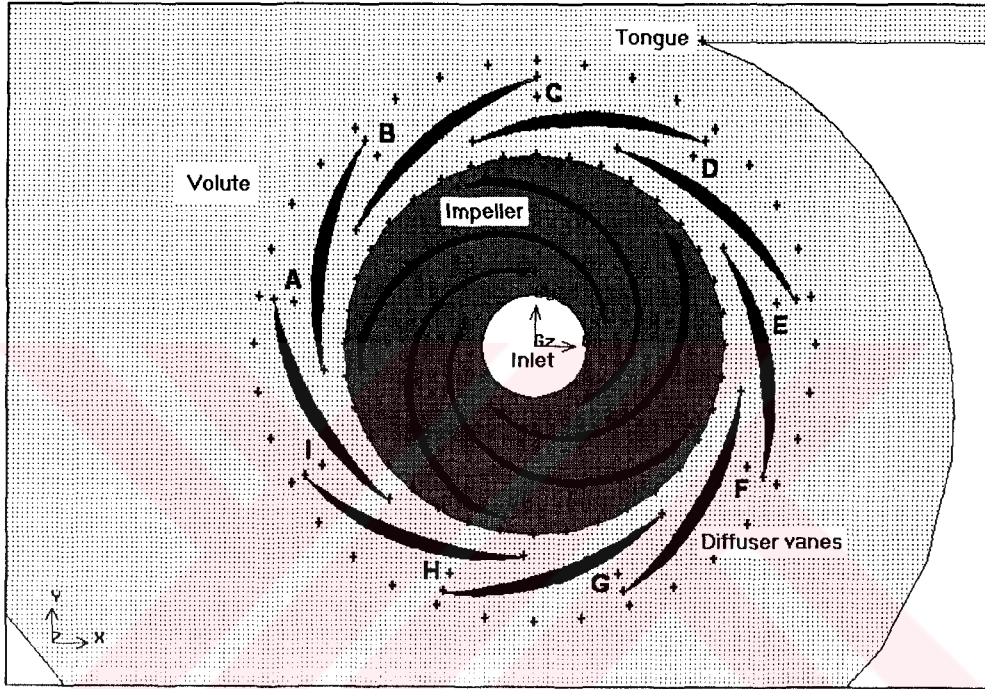


Figure 4.1 Static pressure calculation points; between impeller and diffuser, after diffuser, and between diffuser blades

It was accepted that the starting point of calculations is 0° with respect to x-y coordinate system which passes through the center of the impeller. The increment angle was 10° in counterclockwise direction, as seen from the geometry present in figure 3.2, the position of tongue is corresponding to 61° .

To obtain different gap results, the position and radius of impeller was taken as a fixed value and the diffuser vanes were moved as 5% of impeller radius along the radial direction to decrease the gap distance for each cases. Those cases are classified as $(r_3 - r_2)/r_2 = 0.2$ (Case I), 0.15 (Case II) and 0.1 (Case III) as explained in previous chapter. In these modifications, the geometry of volute and impeller was

not altered. If the radial gap is decreased by increasing the impeller radius, it is excessively changed the trailing edge geometry and flow conditions. Also, these changes can affect the volute geometry, the flow conditions and flow structures in the pump.

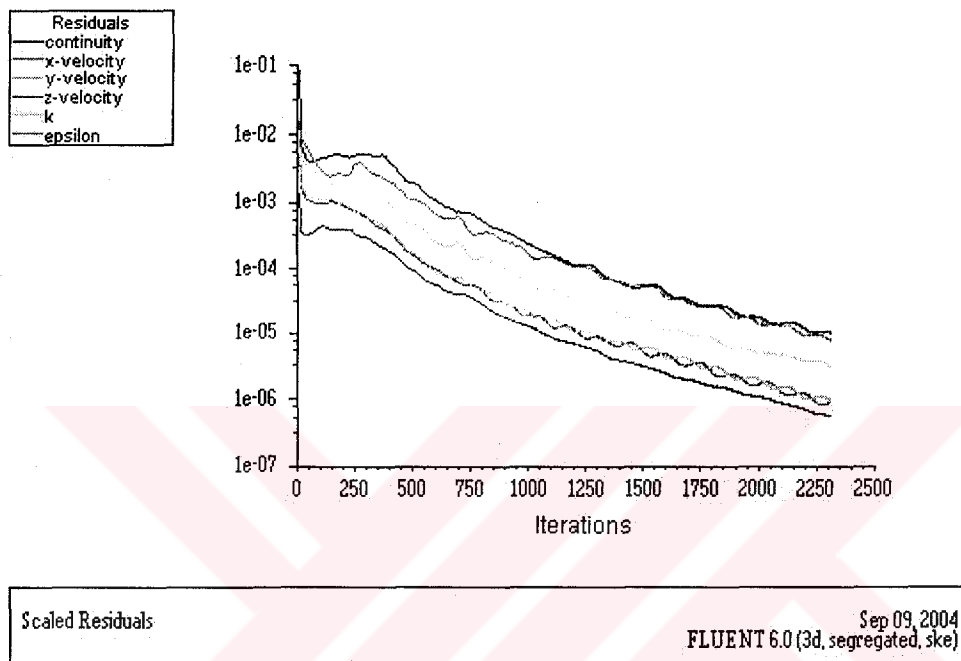


Figure 4.2 A sample of history of residuals

Convergence was monitored by checking the information from computed residuals of all equations and also by keeping track of surface monitors. Scaled residuals were primarily used. The convergence criteria for each run were set to be varying values between $1.03 \cdot 10^{-5}$ and $1.0 \cdot 10^{-5}$ for RMS (root-mean-square) residuals of mass/momentum equations and $k - \epsilon$ equations related to volume flow rate and radial gap ratios. It was clearly that after several hundred times each run, the above criteria could be satisfied, and the convergence was reached gradually. Computations were continued until the values of residuals were progressively reducing to be desired to magnitude, although in a few cases this could not be achieved. The monitoring the representative flow variables through iterations may show that the residuals have stagnated and do not change with further iterations. This could also be considered as convergence. The main convergence monitoring tools employed here were surface

monitors. In this case, it was continued iterating until the flow solution is clear for stability (around 2000~2500 iterations). Three surface monitors were used, one for static pressure and the others for the mass flow rates on the inlet and outlet sections. Convergence was considered to have been achieved to be desired. Such convergence criteria are very stringent and likely to ensure that the results were fully converged, and figure 4.2 shows the sample of histories residuals.

4.1 Performance Characteristics

To specify fluid machines for flow systems, it must be known the pressure rise (or head), torque, power requirement and efficiency of a machine. For a given machine each of these characteristics for similar machines depends upon size and operating speed.

To determine the performance curves, the volume flow rate measurements are made varying from shut off (zero flow) to maximum delivery. The difference in static pressures between the pump suction and discharge cross-sections was used to calculate the head rise which is produced by the pump. Typical characteristic curves for a centrifugal pump tested at constant speed are shown in figure 4.2. For this typical machine, head is a maximum at shut off and decreases continuously as flow rate increases. Consequently, to minimize the starting load, it may be advisable to start the pump with the outlet valve closed. Pump efficiency increases with capacity until the Best Efficiency Point (BEP) is reached, then decreases as flow rate is increased further. For minimum energy consumption, it is desirable to operate as close to BEP as possible.

The pump head varies as the flow rate changes. Figure 4.2 shows pump performance by means of a plot of the head as a function of the flow rate for a certain shaft speed. The efficiency-ratio of the output and input power, also a function of the flow-rate also appears on this plot.

The nominal rating of a pump corresponds to the head and flow rate at the best efficiency point. These are the conditions for which the pump was designed. It is expected that the flow through the pump would be most disturbance-free at these

conditions. Depending on the particular application, it is desired pumps with various head, flow rate, and speed combinations and expected these pumps to operate as efficiently as possible.

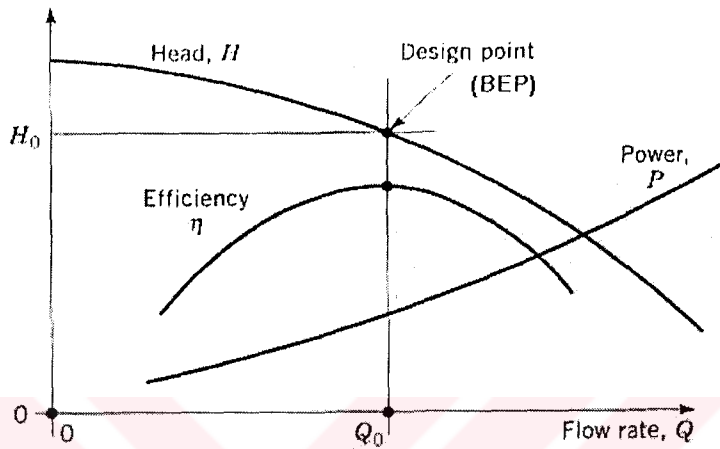


Figure 4.3 Typical pump head-flow curve, BEP at constant speed

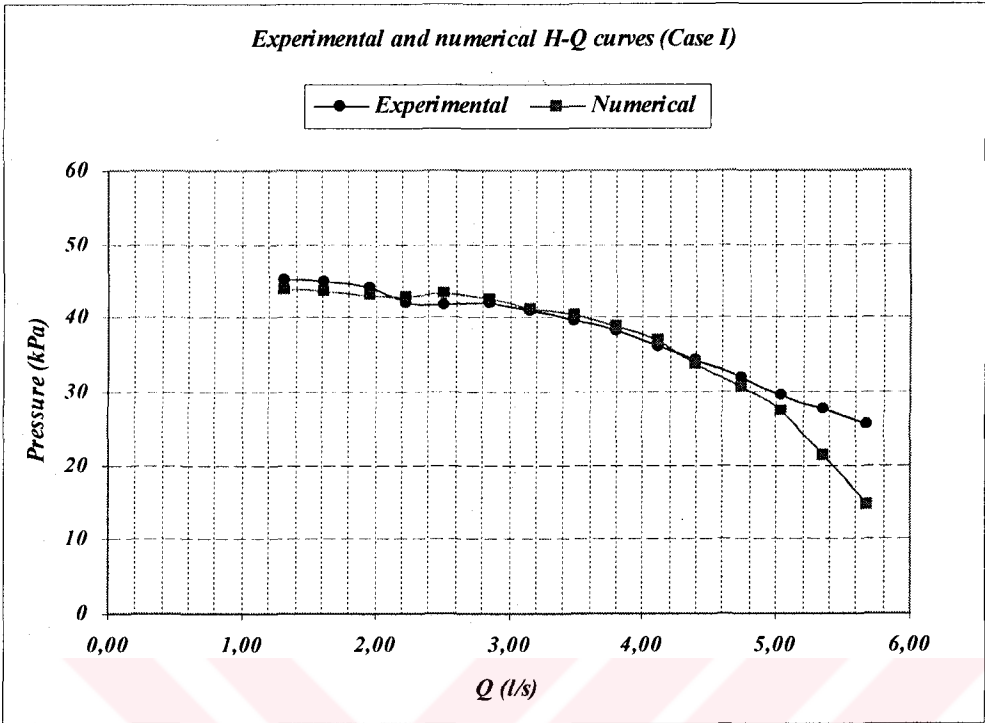
Although selected or designed for a particular flow rate, in practice, pumps operate across a certain range of flow rates. At off-design flow rates the flow pattern deviates from the ideal. The flow separates at the impeller blade tips, in the impeller, and in the diffuser or volute. Recirculation may appear at the inlet, occur at the impeller exit, fluid flowing back into the impeller from the diffuser or volute. Not only does the efficiency deteriorate at off-design conditions, but cavitation can appear, which can destroy impeller. Excessive vibration appears, which reduces bearing life and reliability, and even the entire pumping system can enter into resonant oscillations- into surging. Catastrophic failures are unusual, but off-design point operation for a longer period of time seriously reduces pump life (Kelecy (2003)).

As a reality check, the performance curve is predicted for various flow rates. In this study all the performance curve of the pump is obtained first. The analysis was performed for 15 different volume flow rate levels. To obtain pump performance curve while outlet pressure was fixed to 60 kPa, inlet flow rate was changed. Inlet

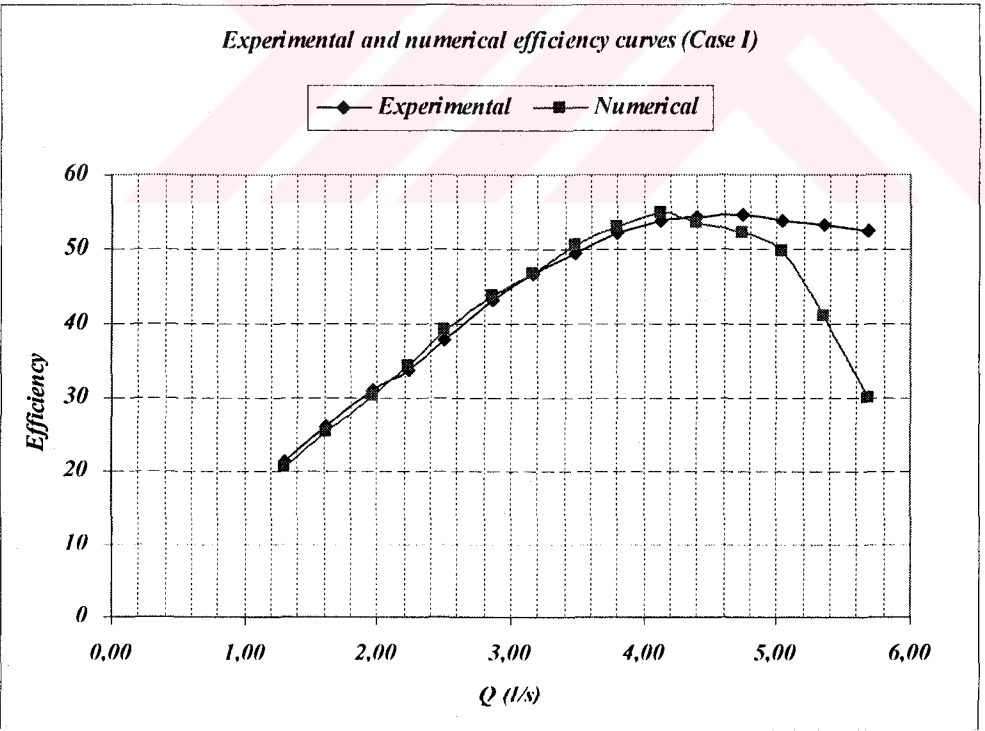
pressure was calculated as depending on inlet flow rate. Finally the head was obtained.

For 3-D the performance and efficiency curves are shown in figure 4.4 for Case I radial gap ratios. Comparisons are made between numerical and experimental data, which is seen to be in good agreement. When the flow rate increases disagreement increase between numerical and experimental results significantly for the lower volume flow rate (2.23-2.51 l/s) displays more unstable results due to stalled region effects. This means that in the rotating stall region, the flow does not converge as smoothly when compared with the higher flow rates due to unstable flow phenomena effects. However, by further increasing flow rate performance curves begin diverge due to the unstable effects and the increasing of turbulence.

When the numerical performance curve is compared with the experimental ones (correspond to 20% of gap distance) obtained by Sinha et al (2001), it is seen that the agreement improves if the volume flow rate is not very low and high. Comparing to the design point values, it is also seen that the best efficiency versus flow rate is described as BEP ($Q = 4.12$ l/s, $\eta = 0.5491$) in the centrifugal pump which traced in figure 4.4b. As seen from performance and efficiency curves, except stall region, for low flow rates the agreement between experimental and numerical results are better than for higher flow rates. For this reason, when numerical results is compared with the experimental results, the error at low flow rates is lower with respect to high flow rates, and the error increases with increasing flow rate.



a)



b)

Figure 4.4 a) The experimental and numerical $H - Q$ curves of the centrifugal pump, b) The experimental and numerical $\eta - Q$ curves of the centrifugal pump, for Case I

4.2 Comparison of Experimental and Numerical Results

In this part of the studies, PIV results obtained at five different flow rates around tongue region by Sinha (1999) were compared with the numerical results achieved at the same flow rates level in Case I. For the purpose of comparisons, PIV results and steady flow numerical results for steady flow conditions were used. In the case of and numerical results, impeller orientation was fixed at 6° for PIV images. Figures 4.5-9 show an averaged and instantaneous velocity map at chosen impeller orientation. Static pressure contours, velocity contours, velocity vector, vorticity and TKE (Turbulent Kinetic Energy) for each volume flow rates are displayed in figures 4.10-14.

Sinha (1999) performed the experiments in C passage and near it as indicated in figure 4.1. For better understanding of comparisons the analysis of the numerical work interactions were chosen on C passage. Passage C will be stated as “passage” in the following sections. If a different passage has to be considered it will be identified through the following work with a new nomenclature.

The averaged and instantaneous velocity fields for different flow rates (3.80, 3.48, 3.16, 2.86 and 2.51 l/s) obtained by Sinha (1999) were shown in figures 4.5-9. Figure 4.5 shows averaged absolute velocity distributions and instantaneous velocity vectors for 3.8 l/s flow rate. The arrows show axial and tangential velocities the contours consist of circumferential velocity around the tongue region of the pump. The jet wake flow pattern clearly is shown in both studies. Highly uniform distortions of TKE occurred around blade tip of the impeller and within the diffuser.

For 3.8 l/s, the static pressure contour, velocity contour, velocity vector distribution and TKE on the base of 3D flow model are shown in figure 4.10 for Case I, respectively. In figure 4.5 the average velocity and instantaneous velocity vectors result is given for the experimental results at 3.8 l/s mass flow rate. As is evident from all of the phase averaged data that as the flow passes through the diffuser becomes non-uniform and breaks into a series of jets whose impact extends well into the volute. This observation is clearly exist in numerical results which the present sample area contains traces in the volute of the flow structure that arise from the

diffuser vane passage located at 80° that is almost $1/4$ of the volute upstream. This non-uniformity is not only due to the wake of the diffuser vane but also due to a substantial slowdown of the flow in the convex side of the vane, particularly downstream of the point where the vanes overlap.

As the volume flow rate is decreased below 3.80 l/s, the jet velocity exiting in the sample area (passage C) continues to decrease as is observed from the magnitude of the velocity contour plots. When the flow rate decreases the jet wake start to destabilize and TKE level also increases. The jetting from the passage C decreases and disappears at about 3.16 l/s as seen in figures 4.6a, 4.7a. The same flow behavior also was obtained by the numerical analysis as presented in Figures 4.10b- 4.14b. In the lowest flow rate the jet disappears and reverses flow starts from diffuser to inside of the impeller. The stall occurs on the pump. There is a good agreement between numerical and experimental results.

When the numerical calculated results are compared with PIV results, characteristic velocity fluctuations and flow structures are similar in the same passage and the regions near to tongue as seen in figures 4.5a and 4.10b. The velocity magnitude increases while fluid moves further downstream. One should also notice the important velocity fluctuations registered between impeller and diffuser region. But, it is seen that velocity fluctuations decrease after diffusers.

All specified volume flow rates at the volute region, pressure values are higher than in the impeller and its distribution is more uniform. Through the diffuser pressures increases while flow moves further downstream, as indicated in figures 4.10a, 4.11a, 4.12a, 4.13a and 4.14a. Due to the growth of adverse pressure separation occurs further downstream of the diffuser. Typically, it is known that below design conditions the circumferential pressure gradient increases from the beginning of the volute up to the exit (Dong et al (1992)). The outlet pressure therefore may be a strong influencing factor in the inception of stall. Ogata et al (1995) observed the reverse flow in vane passages where the pressure coefficient was lower relative to the unstalled diffuser passages. This matches our observation of reverse flow where the local pressure is minimal. As, the pressure in the vane passage rises it is able to overcome the high-pressure in the volute just outside the

diffuser; thereby resulting in full recovery from the stall. However, the condition at the inlet of the diffuser and more specifically in the rather large gap between the impeller and the diffuser is an added factor that influences both the inception and propagation of stall. As shall be observed soon from the velocity measurements at varying flow rates the flow at the onset of stall separates near the concave side of the diffuser vanes. This in turn may increase the incidence angle with respect to the concave side for the following diffuser passage giving rise to rotating stall.

Turbulence production is also significant, and a considerable portion of energy losses occur within the volute. It is particularly high at the interface between the jet and the wake and also behind the blade. Since the flow separation and the vortex shedding are usually accompanied by large energy losses, these should be eliminated to improve the fluid dynamic efficiency. Thus, for comparisons the investigations of the flow in the pump used also TKE and vorticity contours. The Reynolds stresses, the turbulence fluctuations of absolute velocity and its component are higher in these regions with boundary layer effects and re-circulation.

The high turbulence intensities observed in the tongue regions are due to the presence of the wake as seen in figure 4.10e. This region of the passage is occupied by a flow with a low relative velocity. The flow in the passage C has jet-wake structure and the turbulence intensity is much higher. These distributions of the turbulence intensity remain identical for all volume flow rates. In this region where the flow has a high velocity, a substantial variation of velocity gradient is registered. The flow within the impeller is hardly affected by the presence of the vaned diffuser as noted by Sinha et al. (2000). High velocity is associated with a high turbulence fluctuation (and intensity). Boundary layer effects reduce the velocity while the turbulence fluctuation and intensity increase. At the inlet of the diffuser, the turbulence velocity fluctuation and the turbulence intensity are higher near the diffuser wall.

At the diffuser inlet, the mixing of the jet-wake structure depends on the proximity of the diffuser vanes. When approaching the diffuser throat, a high rate of velocity variations occurs on the pressure side (convex side) of the vanes. It is monitored that the flow structure within the entire sample area is dominated by

wakes of the impeller blades and diffuser vanes as well as shear layers associated with flow separation. Figure 4.10e displays the numerical turbulent kinetic energy levels in monitoring passage C. Highly turbulent wakes evident in between shear layer turbulence kinetic energies extend far downstream of the diffuser vanes. A high rate kinetic energy appears at the outlet of the diffuser passage. In fact, it dominates the flow within the entire volute. The turbulence level remains high near the wakes, but diminishes with increasing distance from the trailing edges. The computed velocity contour show, that the TKE has a qualitative agreement with the experiment.

Within the diffuser the turbulent kinetic energy is particularly high along the convex side of the vane as seen in figures 4.10e and 4.11e, and it extends well downstream of the vane along its wake.

Vorticity distributions show clearly the wake of the impeller blade reaches and interacts with the flow along the concave side of the diffuser vane as seen in figures 4.10d and 4.11d. The peak level of the vorticity accumulated around the vaneless part of the impeller blades. The blade trace effects also clearly discernable from both numerical and experimental results.

The observation of the flow velocity vectors seem to suggest that the two velocity deficits observed in the velocity vectors near the diffuser vane is due to the interaction effect of the impeller blade wake and diffuser leading edge. This effect is clearly captured in the experimental and numerical solutions. Interaction effects observed in the velocity contours of the numerical and experimental are quite similar. The numerical results of jet and wake type flow pattern clearly captured the trend of the experimental data quite well, which is seen in figures 4.5b and 4.6b, and comparisons also are presented in figures 4.10c and 4.11c.

Figures 4.8b and 4.9b show the below design conditions velocity magnitudes around the impeller, during the period between the passage of two consecutive blades in front of the tongue. The results are presented as measured and computed for the flow rates 2.86 l/s and 2.51 l/s, respectively. A common feature of the static pressure distribution at both flow rate levels is that the contour aligns in a circumferential position and becomes more radial in direction. The curvature of the tongue produces negative slope as the flow approaches to the exit. As the mass flow decreases, the jet

nearly disappeared (figures 4.13-14). For below design conditions, however, the region of the volute close to the tongue presents a simultaneous evolution of the jet-wake flow, with big differences in the whole region. They are the result of the strong interaction between the blades and the tongue. There is good agreement with experimental results for the low flow rates. TKE distributions at the below design flow rate, there is highly distorted region around the tongue and within the diffuser and the impeller blades, as shown in figure 4.14e.

Figures 4.5 and 4.9 contain representative samples of measured velocity maps and computed results (this time an average of 20 each) at 3.80, 3.48, 3.16, 2.86 and 2.51 l/s at the same blade position, respectively. These are shown in low volume flow rates 2.51 l/s, the experimental and numerical results are clearly identical in terms of the jet-wake flow pattern and reverse flow, which are presented in figures 4.9a-b and 4.14c for 2.51 l/s. Typical narrow jetting near the concave side of the vane with a wide separated region in the convex side dominate the flow in close region of concave side of the vane at 3.80 l/s, as shown in figure 4.5. At this flow rate the outward jet always exists in both computed and measured results. Note also that there is a recirculating flow occupies a large area between leading edge of the tongue and the trailing edge of the diffuser vane as seen in figure 4.10d. This flow structure indicates that the velocity gradients near the tongue are in the opposite direction compared to the velocity vectors in the gap between the impeller and the diffuser. So, in the vicinity of the tongue wall, the flow is directed towards the leading edge of the tongue in order to pass the exit of the volute. The recirculating flow around the tongue disappears at lower flow rates. With decreasing flow rate the characteristic of jet velocity decreases and the traces of the jet from passage C eventually disappear as seen in figure 4.9. Conversely, the leakage flow increases substantially, as is evident in comparison of samples shown in figures 4.5 and 4.9. In addition, the velocity in the outer sections of the volute becomes consistently higher than the flow in the vicinity of the tongue, a trend that is characteristic to operation off design conditions (Dong et al. (1992a,b)).

The decreasing jet velocities reduce the magnitude of velocity vector on the concave side of the diffuser vanes. Figure 4.7 indicates that an instantaneous velocity

field reveals that flow separation occurs on the concave side of the stationary vanes. This is due to the increased incidence relative to the concave side near the diffuser throat region that arises from the direction of the leakage flow. The other influence is caused by the geometry of the diffuser as observed from the interpretations of the varying propagation speed in close region of the concave side of the diffuser. As the volume flow rate is decreased further to 2.51 l/s volume flow rate, both the passages C and D consistently show a strong flow reversal (figure 4.9e). Also, note that the flow tends to move towards the exit at a higher flow rates immerging from the passage C. As, the volume flow rate is decreased the jet flow immerging from the C passage diminishes completely.

As seen from figures 4.5-6, indicate that the jet flow occupy the C and D passages. Furthermore, there is no separation or reversed flow in both passages. But, for the lower volume flow rate, the lower vane passage (C passage) has a strong reverse flow. The upper vane passage (D passage) has also a reverse flow but velocities are lower in magnitude corresponding to the value of C passage. The reverse flow velocities in general are only about 10% in magnitude as compared to the velocity of outflux flow from the impeller. Being below the design conditions and due to the stall, these outflux from the impeller leaks substantially towards the diffuser passages further away in the clockwise direction. The fact that a high rate of leakage flow between the impeller and diffuser indicates high circumferential pressure gradients in the gap. It should be noted from the velocity fields and specifically from the fact that the jet flow immerging from the diffuser passage C goes towards the exit, preventing the leakage between the diffuser and the tongue completely. This indicates that there are no strong lateral pressure gradients in the volute near the exit/beginning of the volute when the diffuser passage experiences a local maximum in the pressure (Sinha (1999)).

The pressure distribution at the exit near the suction surface was higher than it was in other regions; therefore, reverse flow will occur there as well. The high pressure that exists near the exit and is maintained locally near the beginning of the volute in the stalled case explains the persistence of stall in the passages of the diffuser. The lower vane passage at this time is recovering from the reverse flow and

marks the closure of the stalling phenomenon. As the pressure further rises in the passage it experiences 'full flow'. In fact there is high-speed jet flow that is largely separated from the convex surface of the upper vane. At this time even the following vane passage is flowing full.

It can be seen clearly that the pressure increases gradually in a streamwise direction, and normally it has higher pressure on the pressure surface than on the suction surface. But as shown in figures 4.11a,c 4.12a,c and 4.13a,c; the pressure distribution at the exit near the suction surface was higher than it was in other regions, therefore, reverse flow will occur there as well.

The pressure distributions differ from with each other for the same flow rate. A high pressure region occupies the area starting from the throat to the inlet of the vane channel, and the pressure gradually decreases at inlet in the consecutive vane channel. A reversed flow is observed in first vane channel, but not in second vane channel. The outward flow from inlet boundary is blocked by the flow reversal in first vane channel, and the flow turns into the second vane channel through the narrow clearance between the diffuser leading edge and inlet boundary. This leakage flow is accelerated around the clearance and the pressure at the inlet of second vane channel is lower than that in first vane channel. These relations can be seen in B, C and D vane passages.

The calculated BEP was found at a flow rate 4.12 l/s, the measured one at 3.80 l/s. The head and efficiency curves are slightly shifted towards the lower volume flow rate values compared to the experimental results. Theory indicates that if the leakage flow decrease, the curves shift to higher flow rates. Numerically calculated pressure losses in the volute are higher comparing to the experimental results because of unequal mass flow distribution between the different vane channels. The differences between the measurements and the numerical values on BEP depend on these phenomena.

Some differences have arisen in the comparison between the numerical and experimental pressure calculations in the tested centrifugal pump, especially for high flow rates. Nevertheless, and generally speaking, it is clear that the numerical model developed would have been able to predict the main characteristics of the impeller-

diffuser interactions for a wide flow range (see figure 4.4). On the other hand, those differences are not seen in the performance curve prediction. In the present study, comparisons of performance curves were made for the pump outlet section. The comparisons of the solutions were made at the same level of volume flow rates, which is regulated by the adjustment of the pump inlet velocity. A good tendency was achieved over the entire flow range, whereas for high flow rates, a deviation occurred in performance curves.

It was found that when the inflow rate is in the near of design flow rate (3.80 l/s experimental, 4.12 l/s numerical), the flow patterns look similar to each other. But if the flow rate drops below a volume flow rate value (2.51 and 2.86 l/s) of design flow rate, the flow pattern changes. A strong reverse flow occurs near the pressure surface, as is shown in figures 4.13-14. The flow patterns look similar to each other for the design flow rate give above. Also, similar conclusions can be seen in cases in which a pump operates at different radial gap ratios.

As explained in the next parts of thesis, for the design flow rate a better flow distribution inside the pump is obtained. For all radial gap ratios, the minimum loss condition is attained for flow rates in the region of 4.12 l/s as seen in figure 4.23 and at this operating point, the losses are minimum. On the other hand, for flow rates higher than the nominal one, the losses are increased and the relative position of the separation point agrees with the expected trend.

It can be seen that the agreement between experimental and calculated velocity contours and vectors is rather satisfactory. This shows the present code is a useful tool for flow simulation.

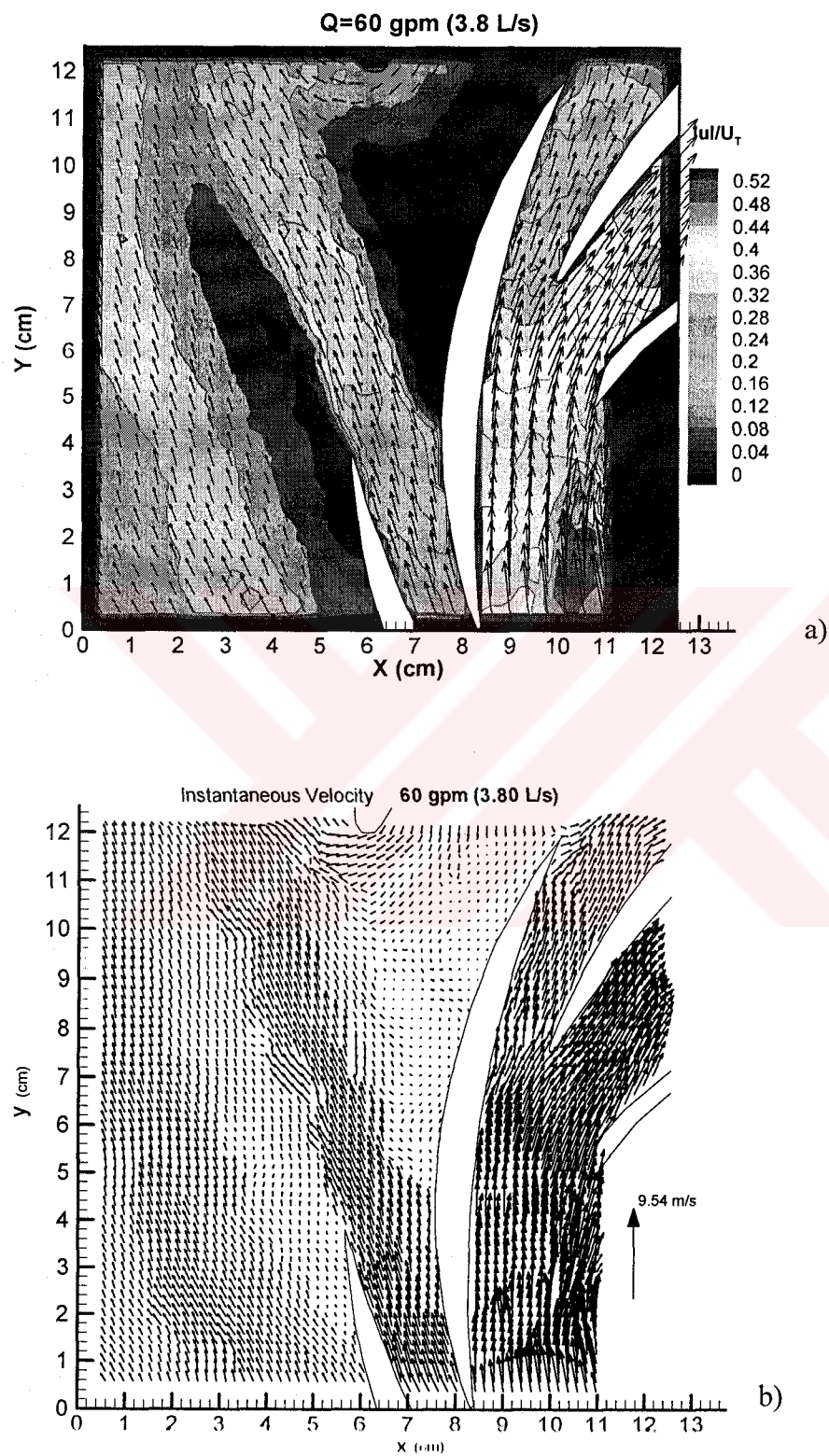


Figure 4.5 a) Averaged velocity b) Instantaneous velocity, for 3.80 l/s

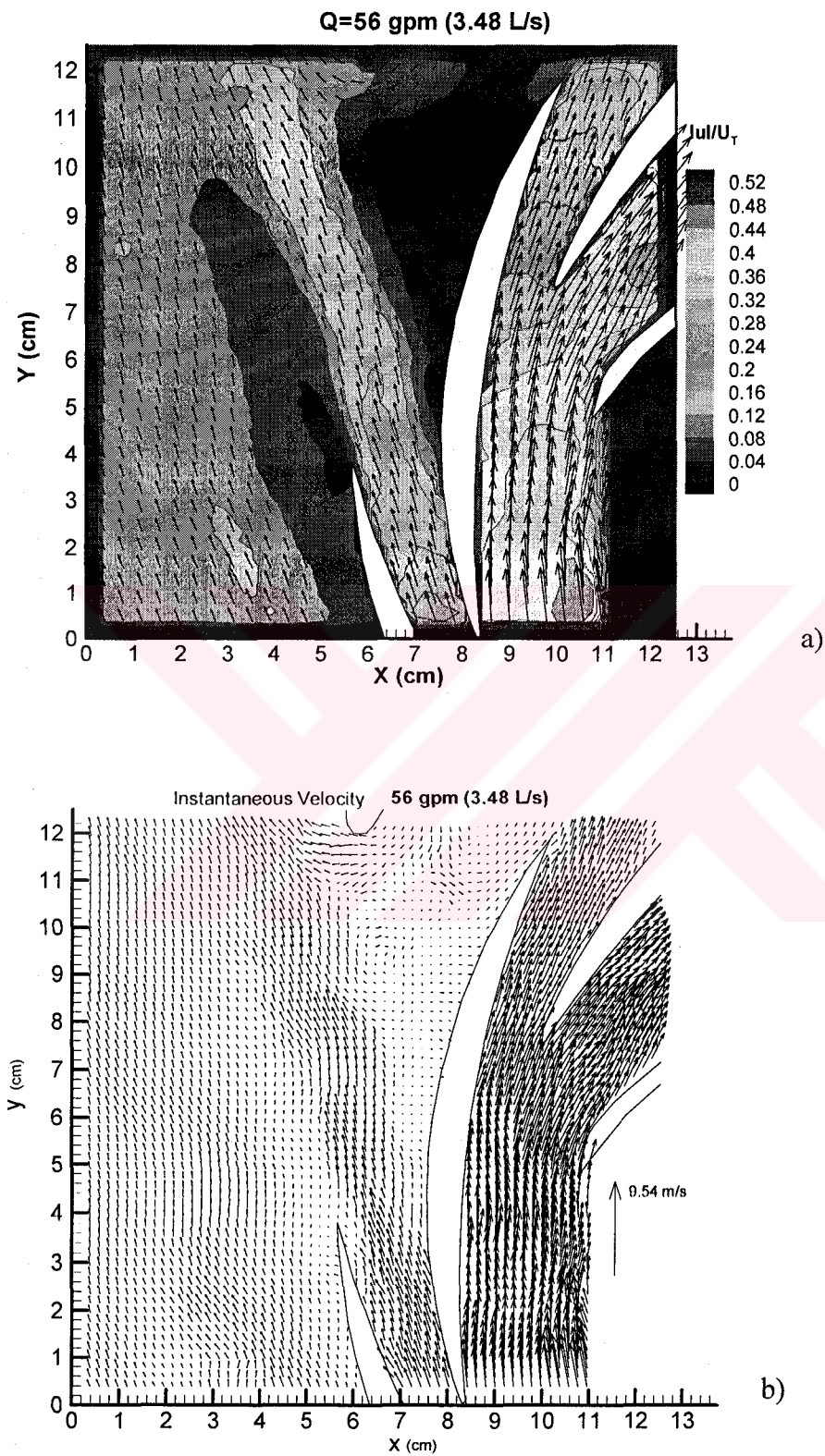


Figure 4.6 a) Averaged velocity b) Instantaneous velocity, for 3.48 l/s

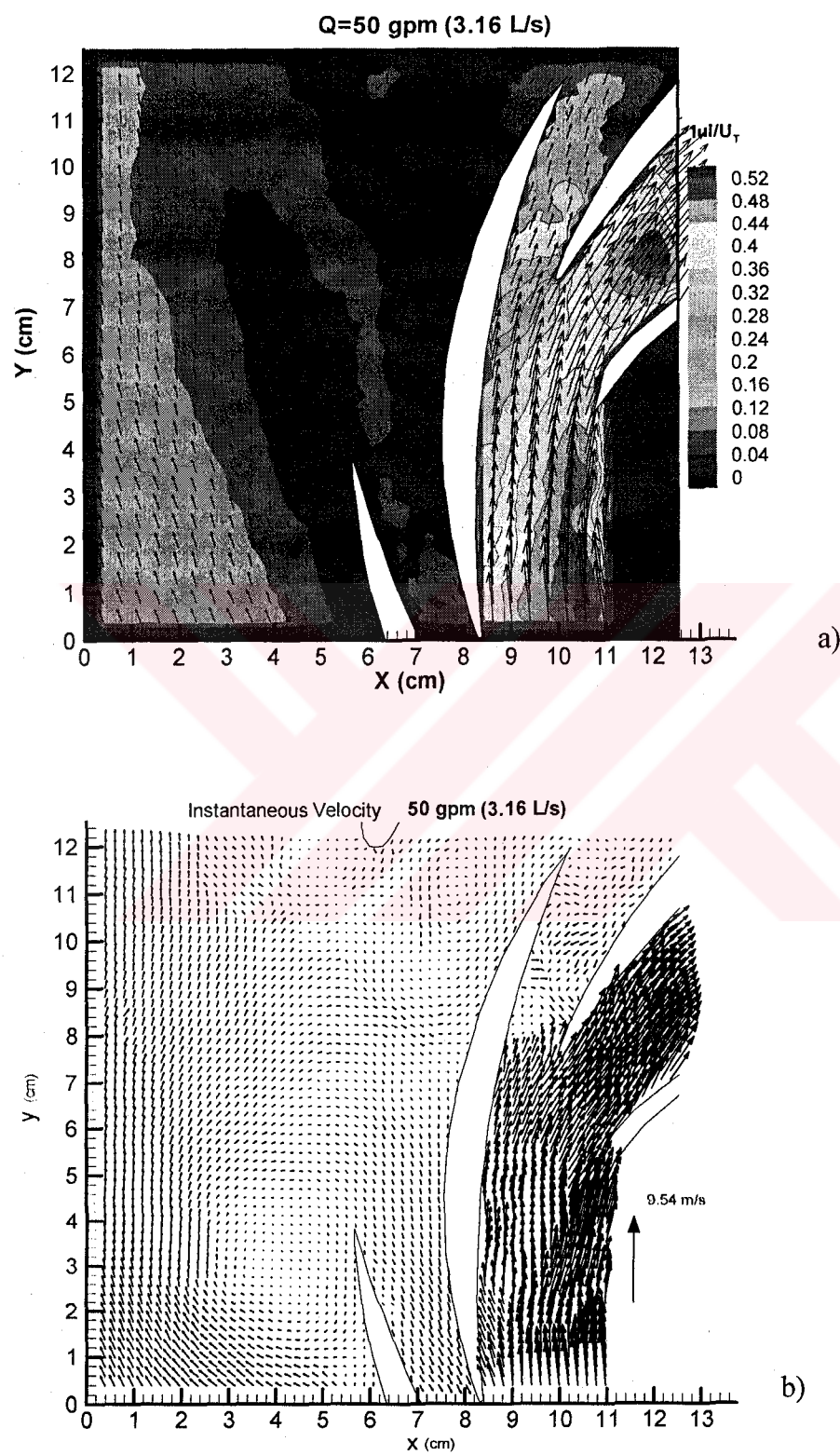


Figure 4.7 a) Averaged velocity b) Instantaneous velocity, for 3.16 l/s

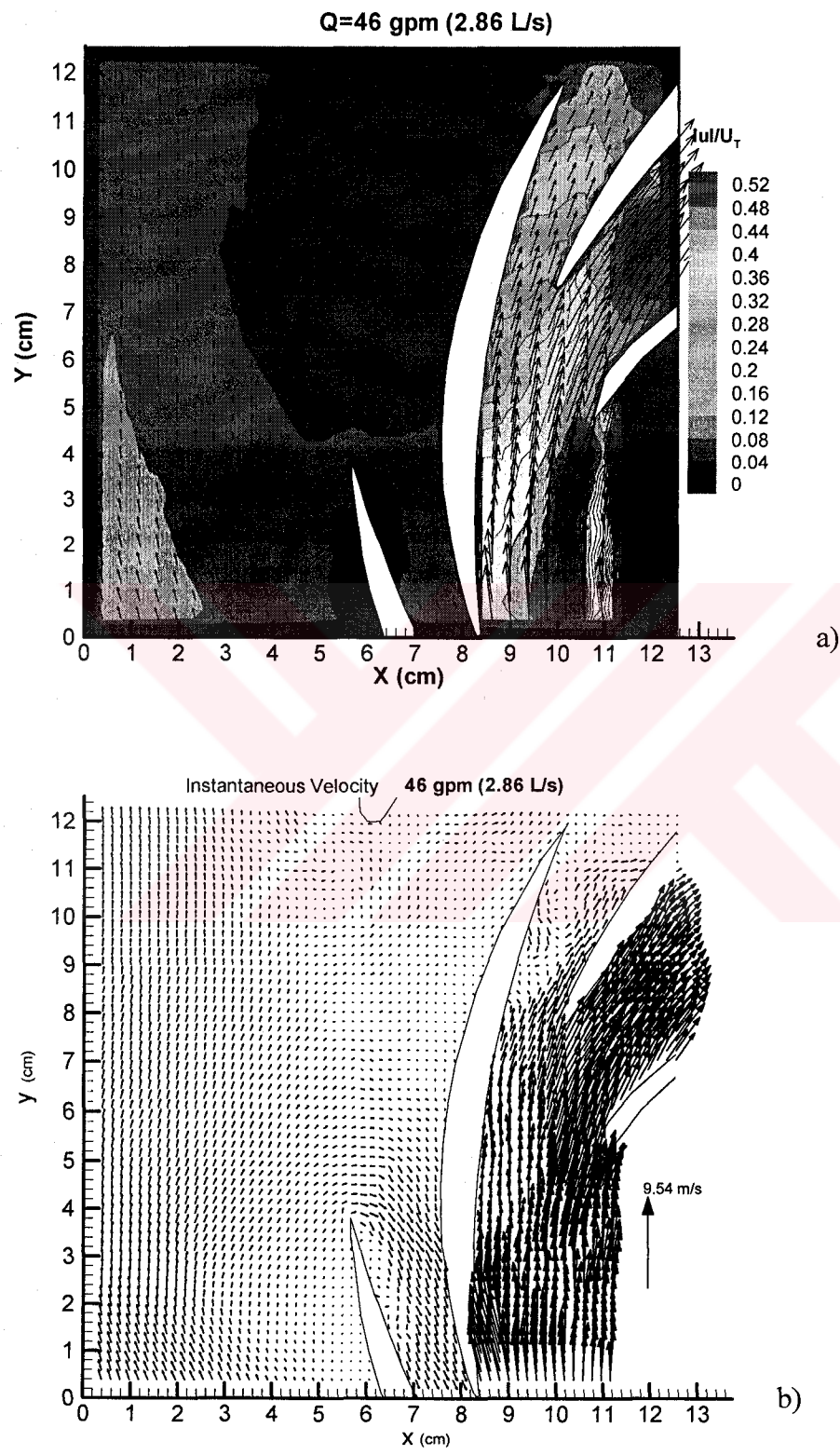


Figure 4.8 a) Averaged velocity b) Instantaneous velocity, for 2.86 l/s

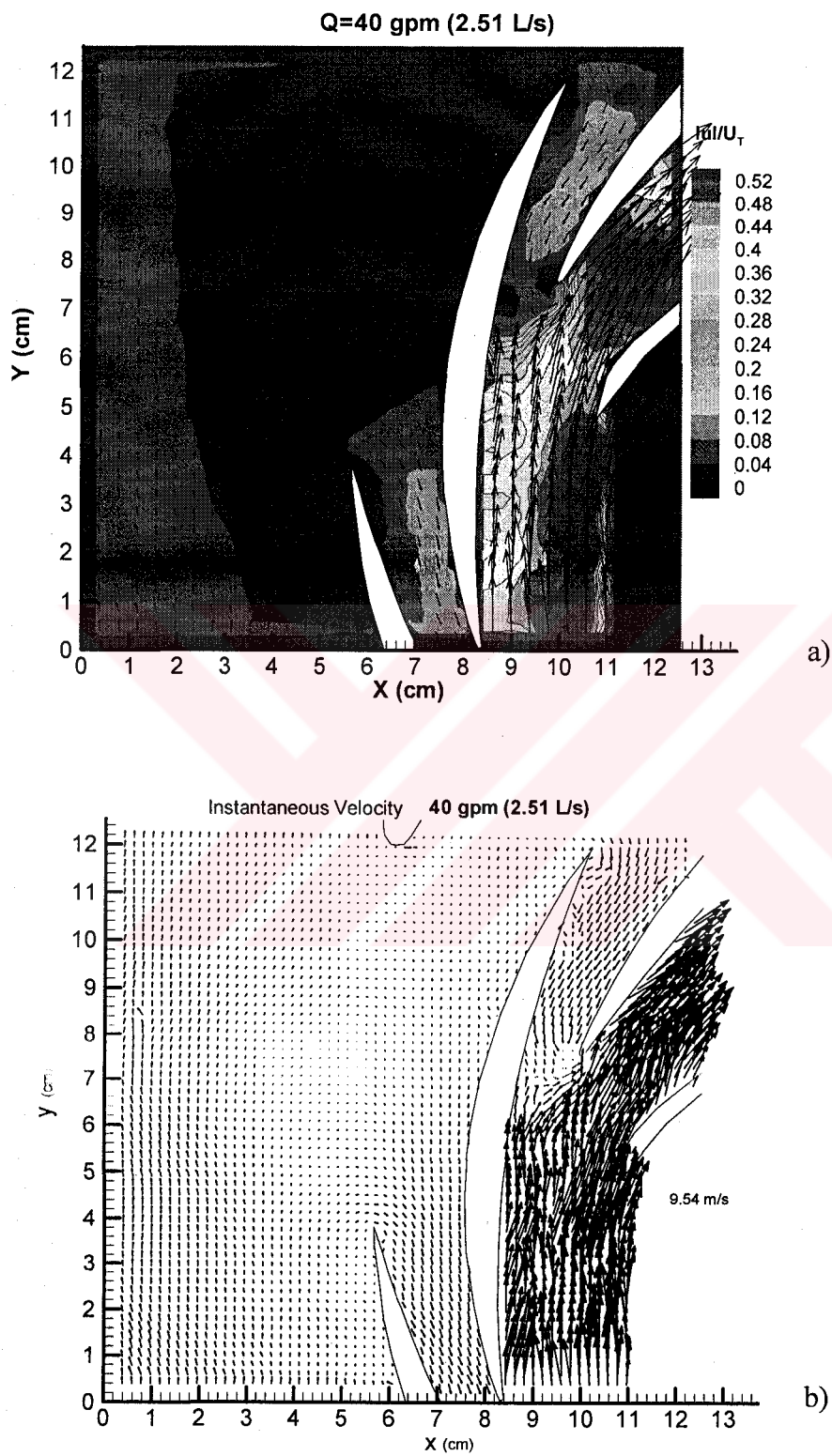
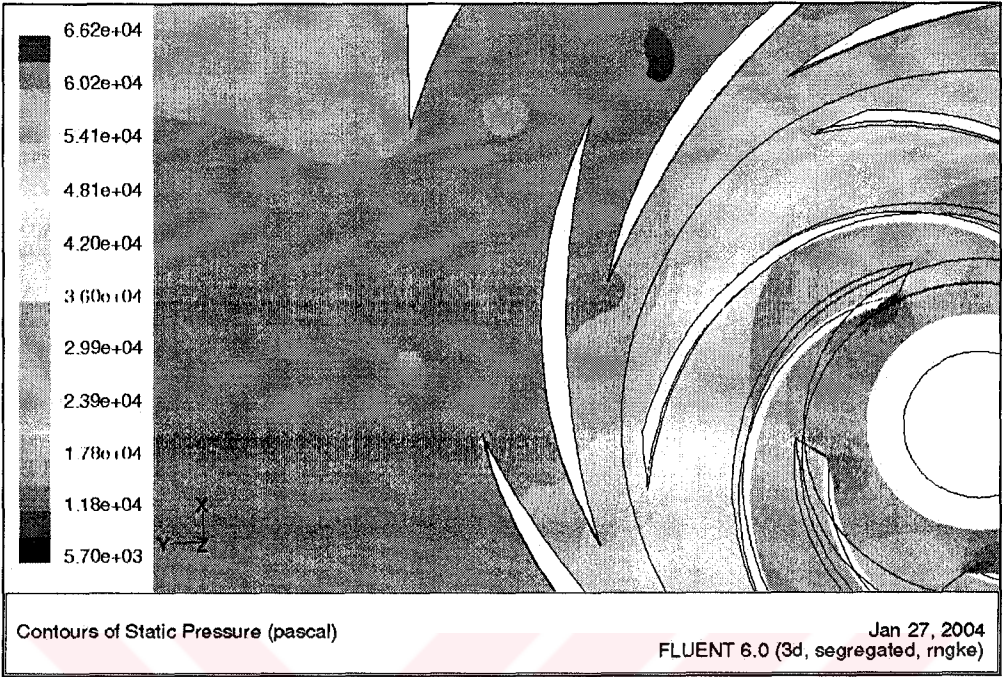
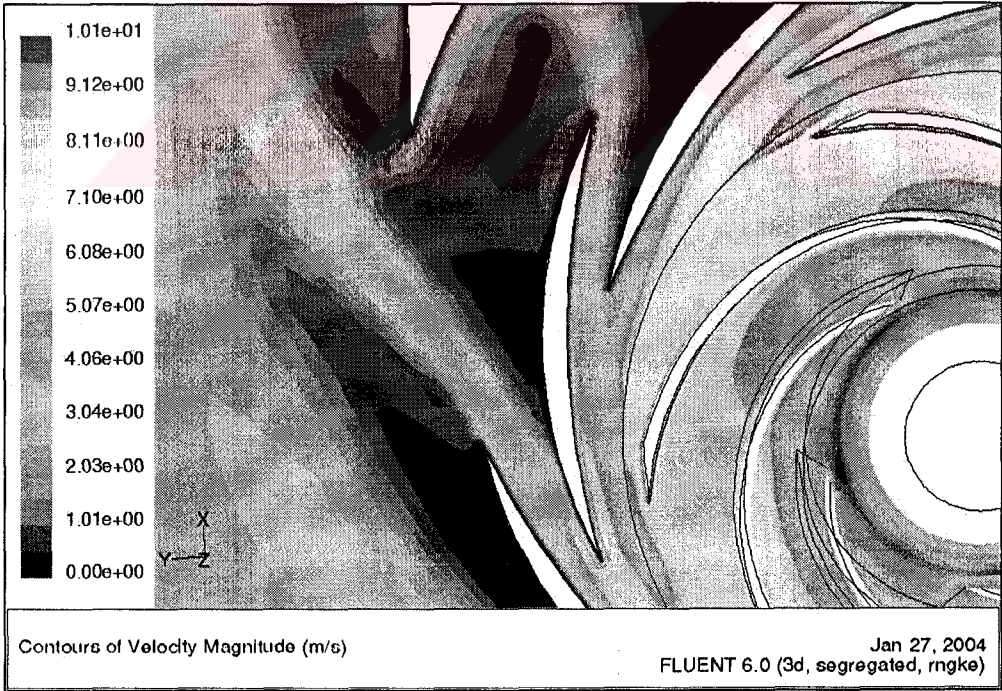


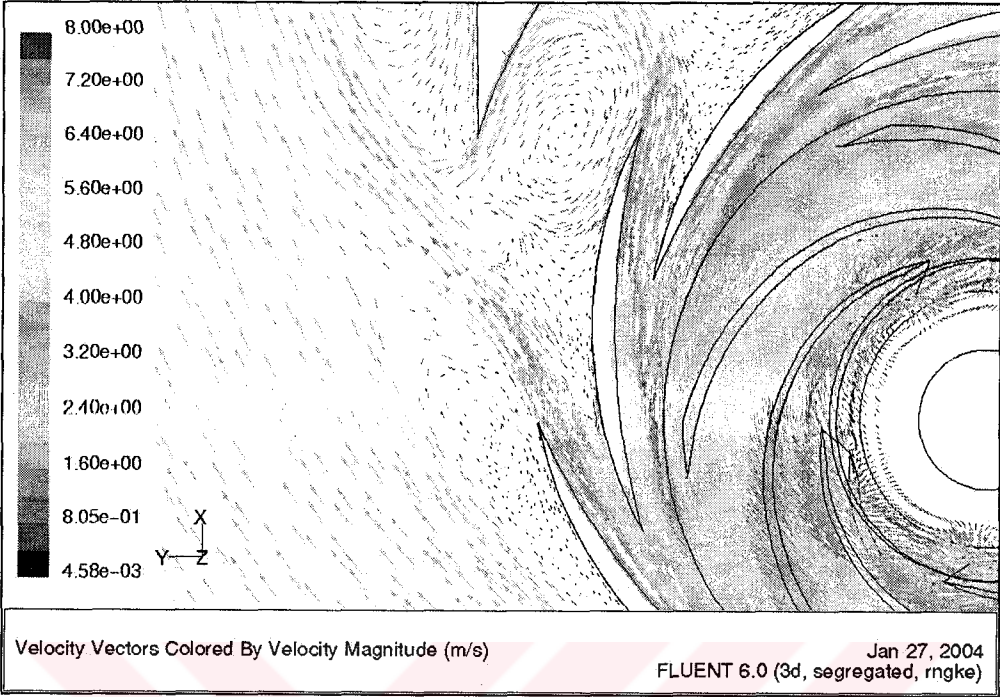
Figure 4.9 a) Averaged velocity b) Instantaneous velocity, for 2.51 l/s



a)



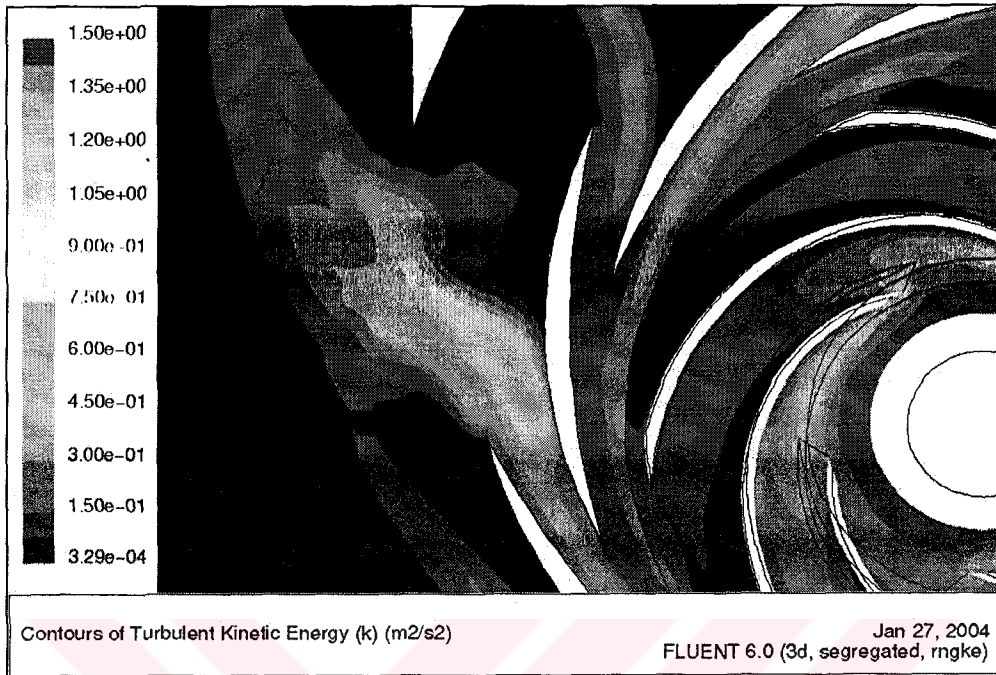
b)



c)

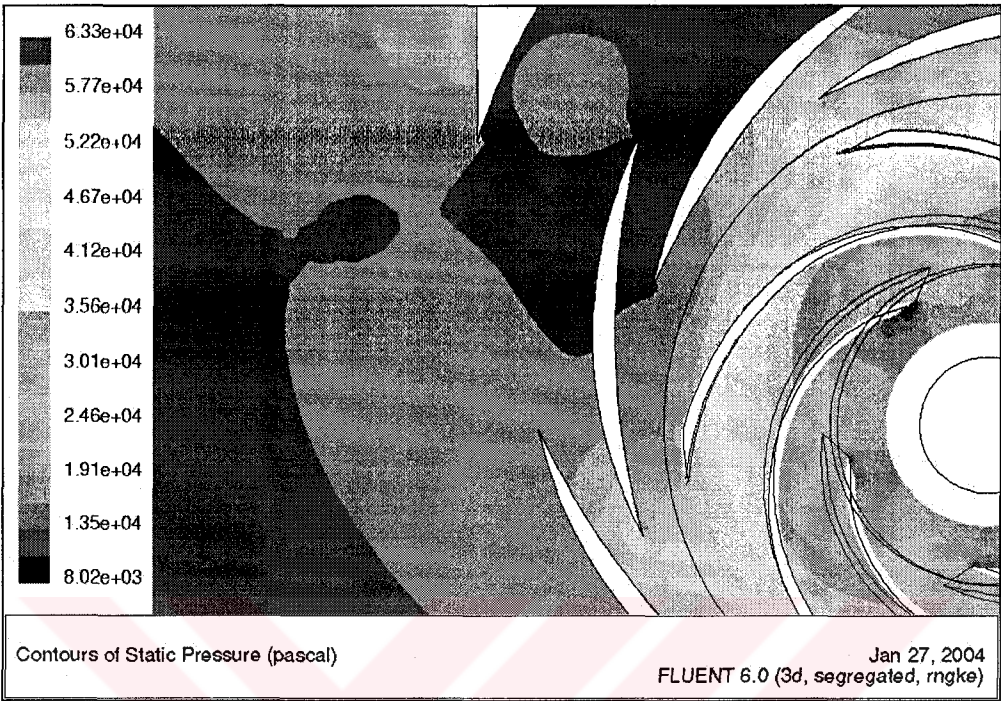


d)

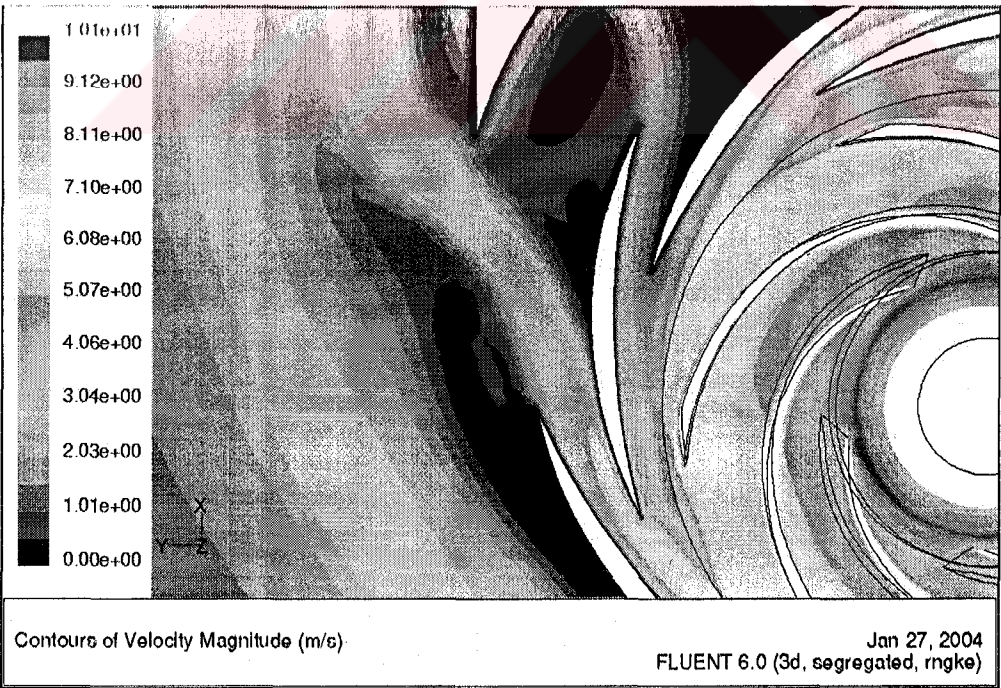


e)

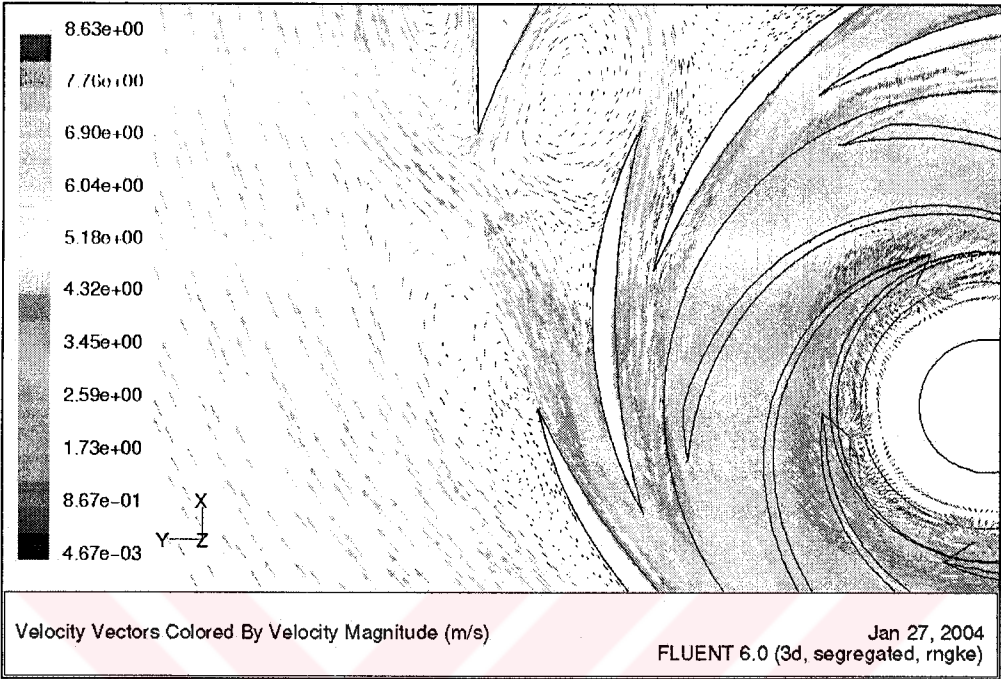
Figure 4.10 a) Contours of static pressure b) Contours of velocity magnitude
c) Velocity vectors d) Contours of vorticity magnitude e) Contours of TKE (for 3.80 l/s) – Case I



a)



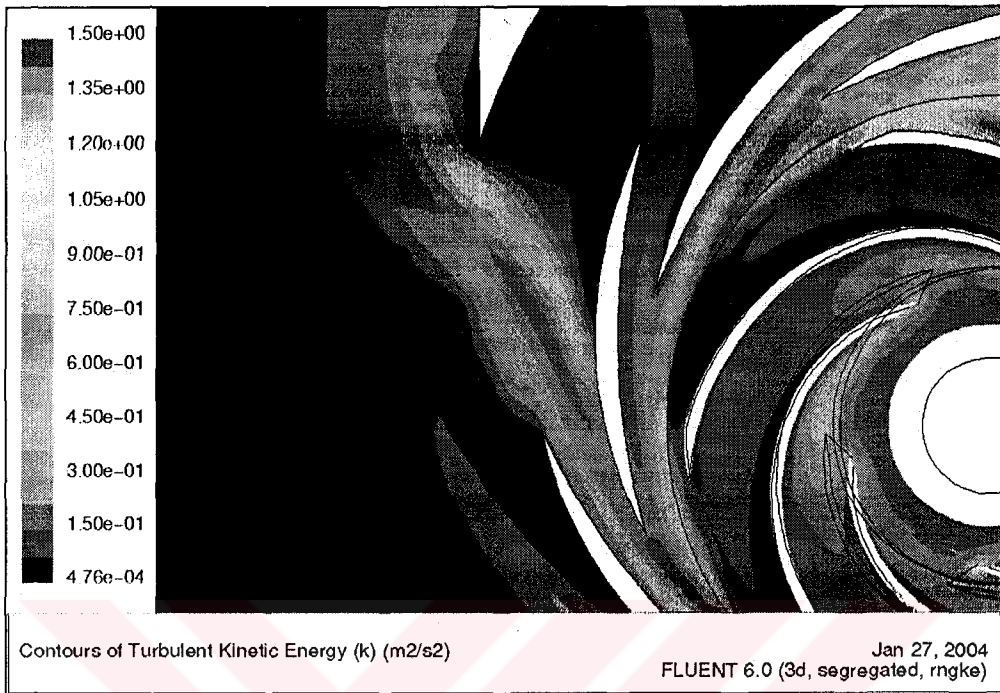
b)



c)

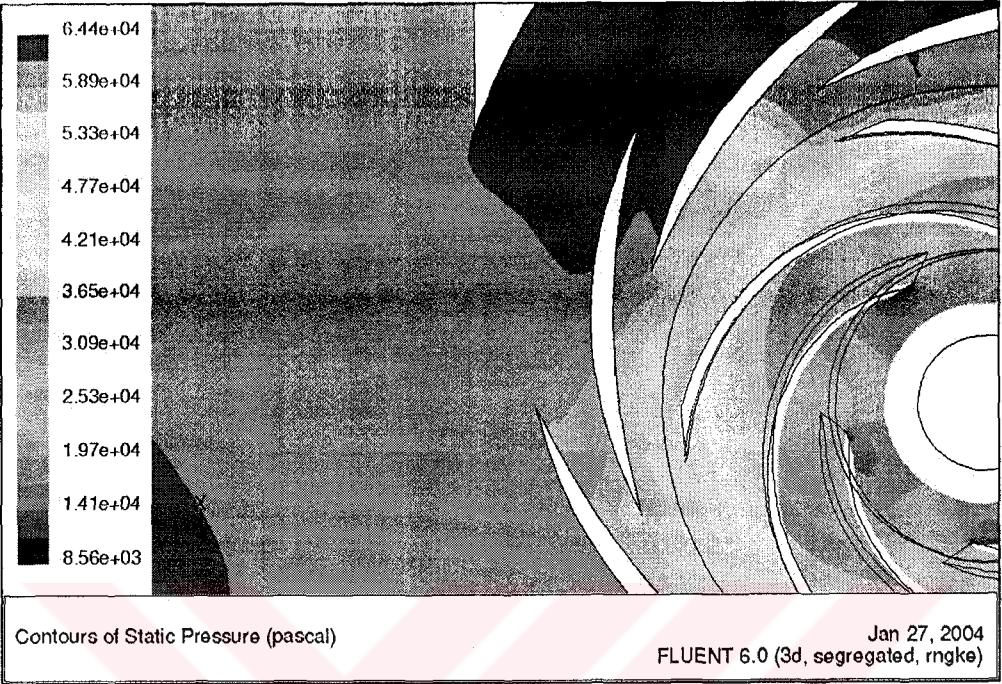


d)

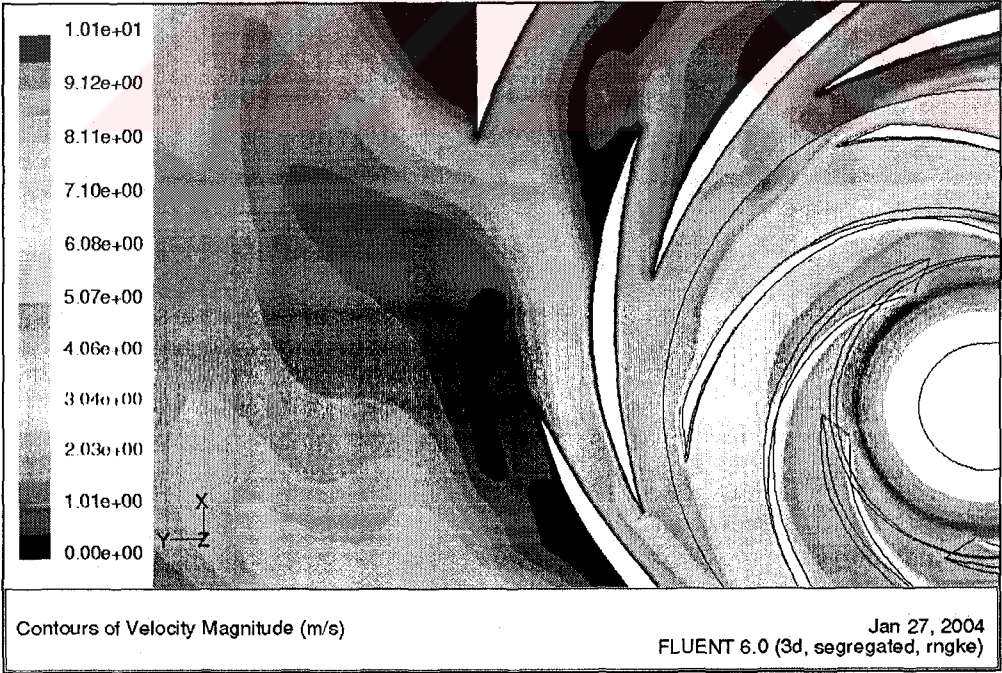


e)

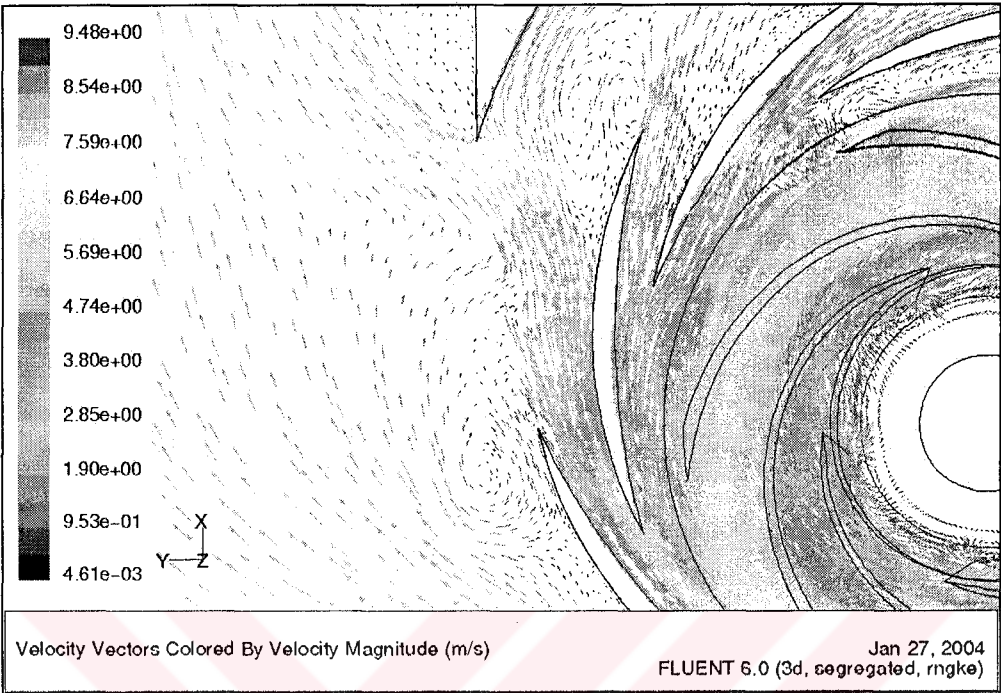
Figure 4.11 a) Contours of static pressure b) Contours of velocity magnitude
c) Velocity vectors d) Contours of vorticity magnitude e) Contours of TKE (for
3.48 l/s) – Case I



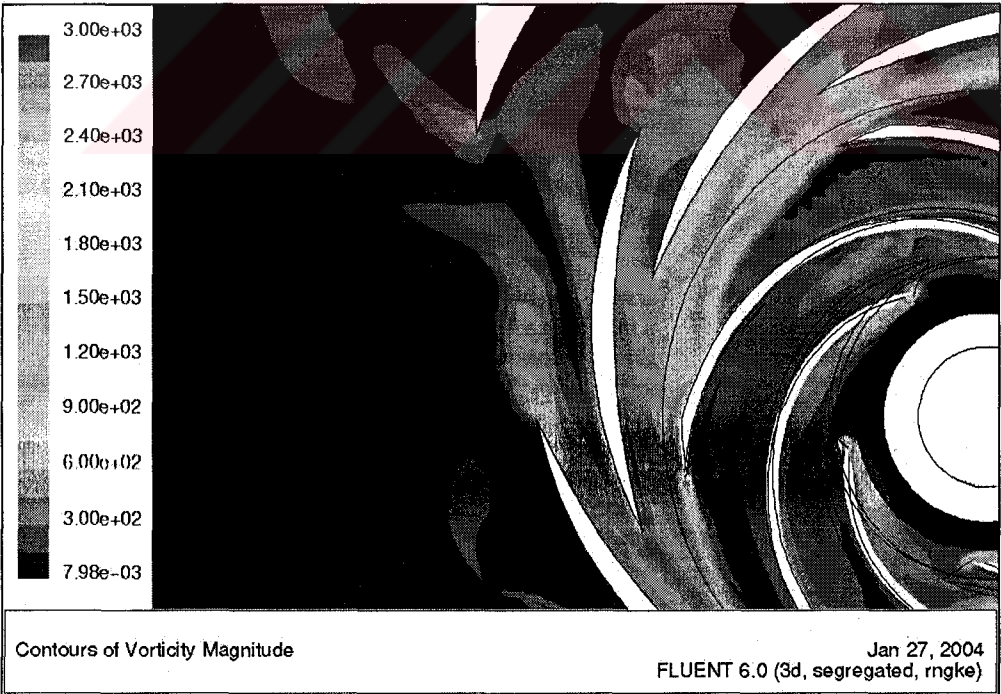
a)



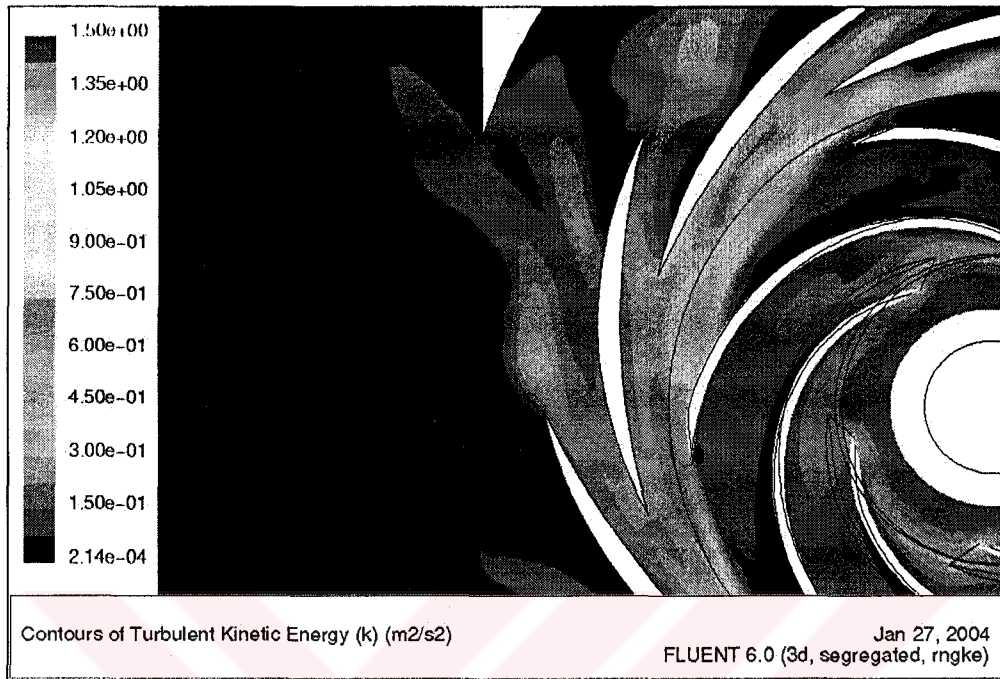
b)



c)



d)

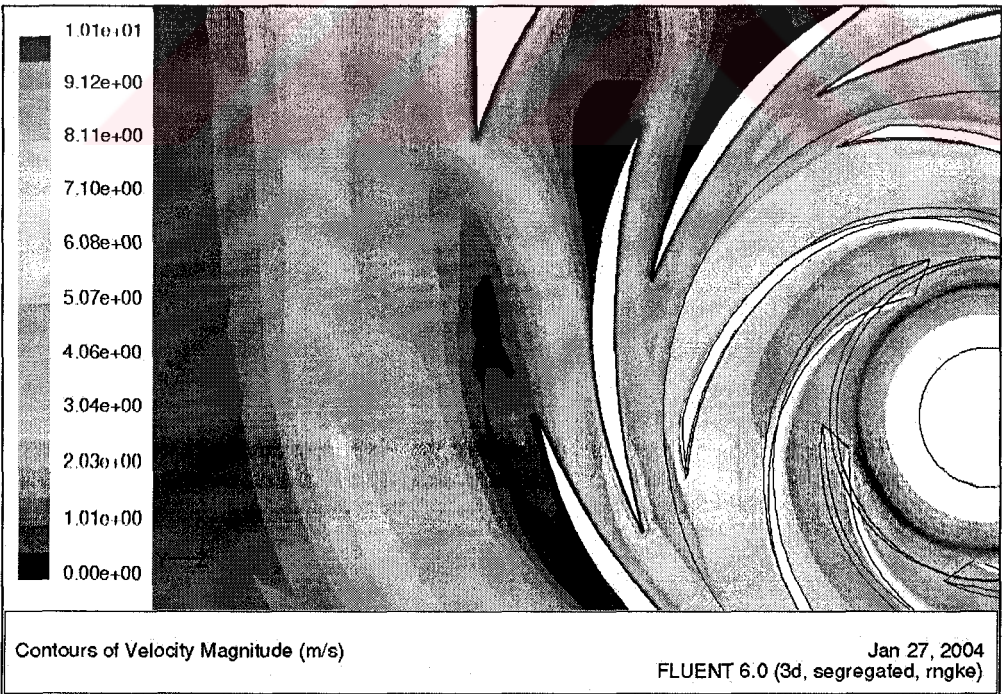


e)

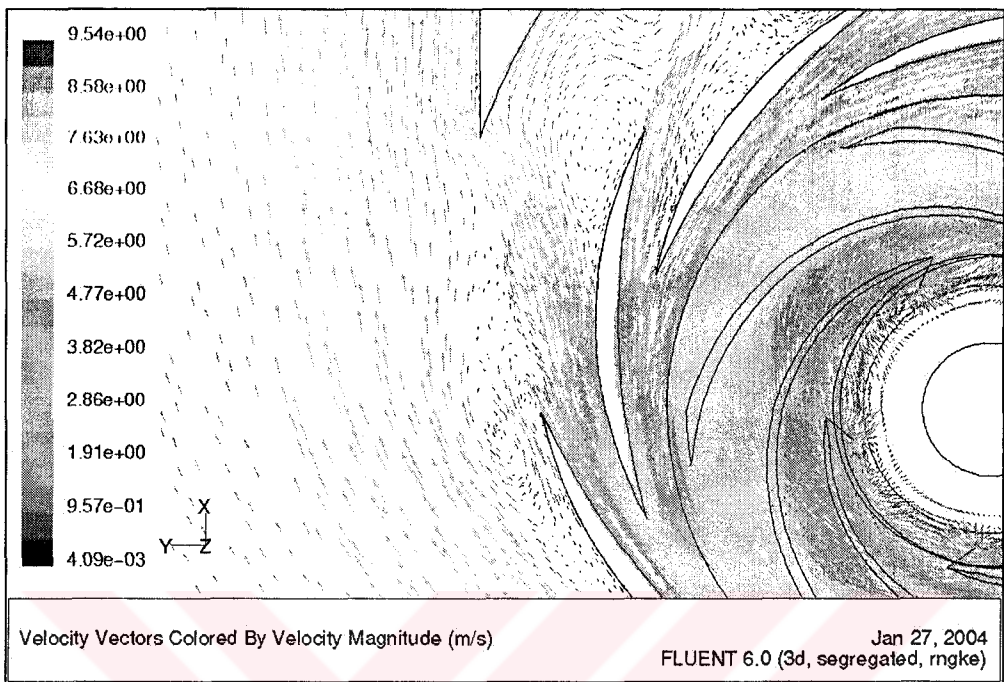
Figure 4.12 a) Contours of static pressure b) Contours of velocity magnitude
c) Velocity vectors d) Contours of vorticity magnitude e) Contours of TKE (for
3.16 l/s) – Case I



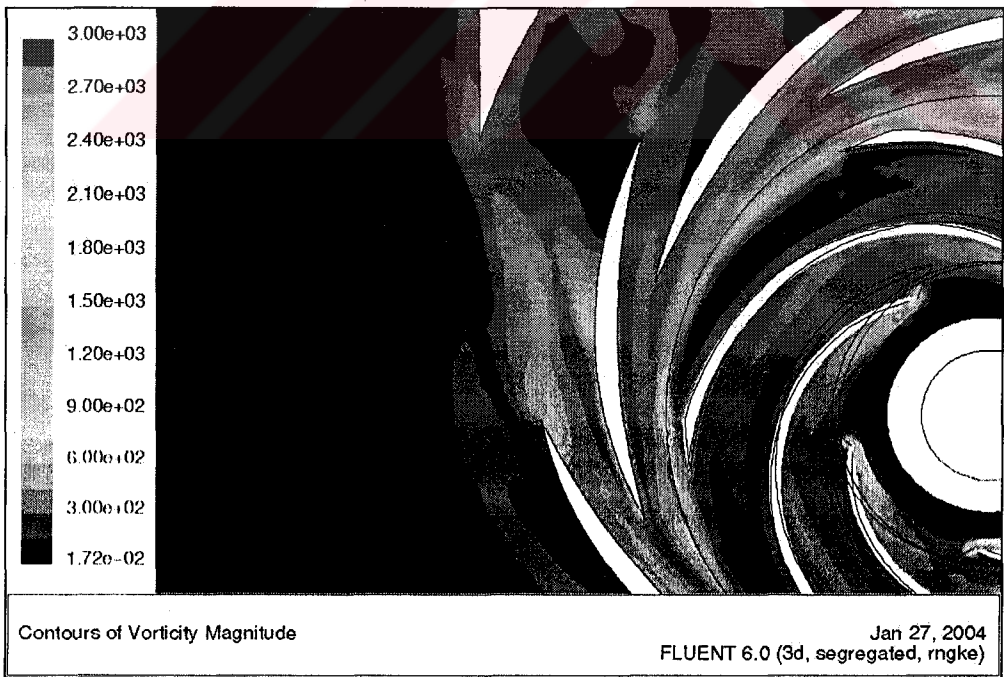
a)



b)



c)

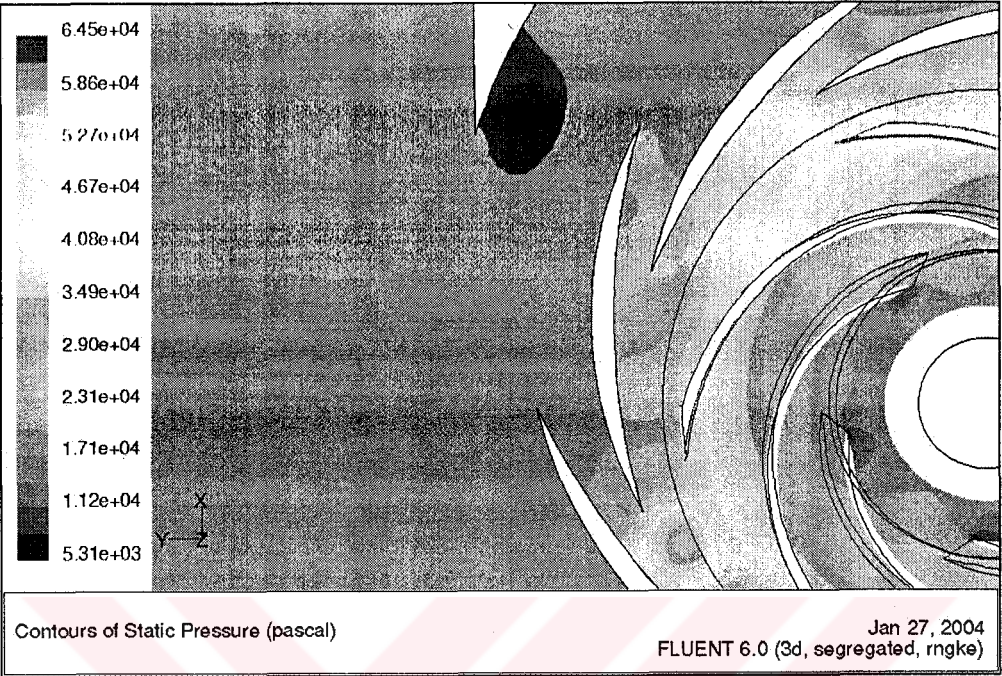


d)

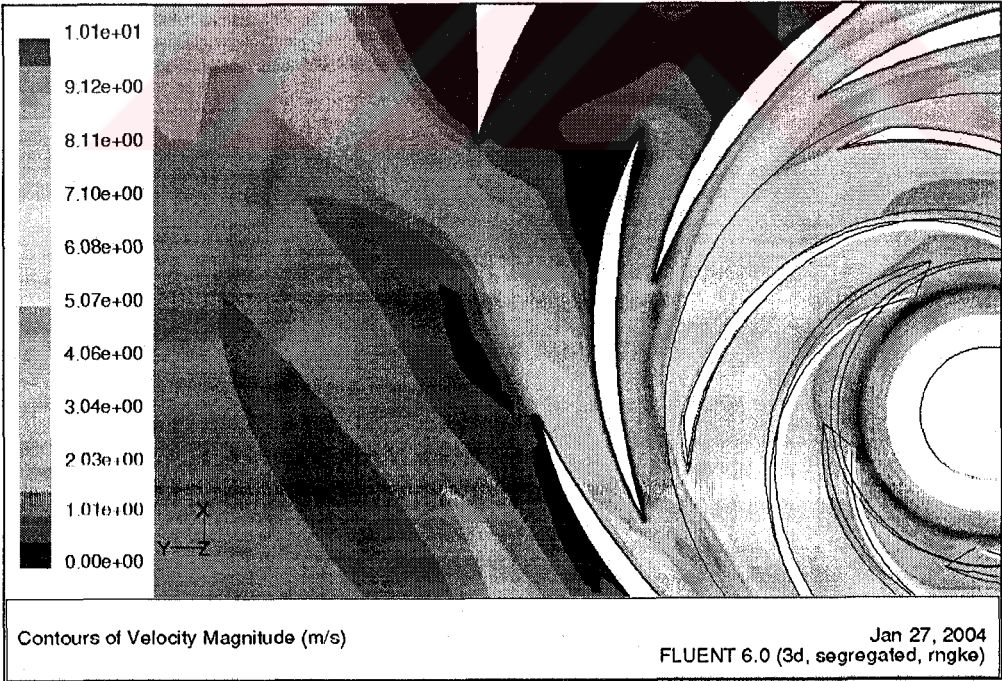


c)

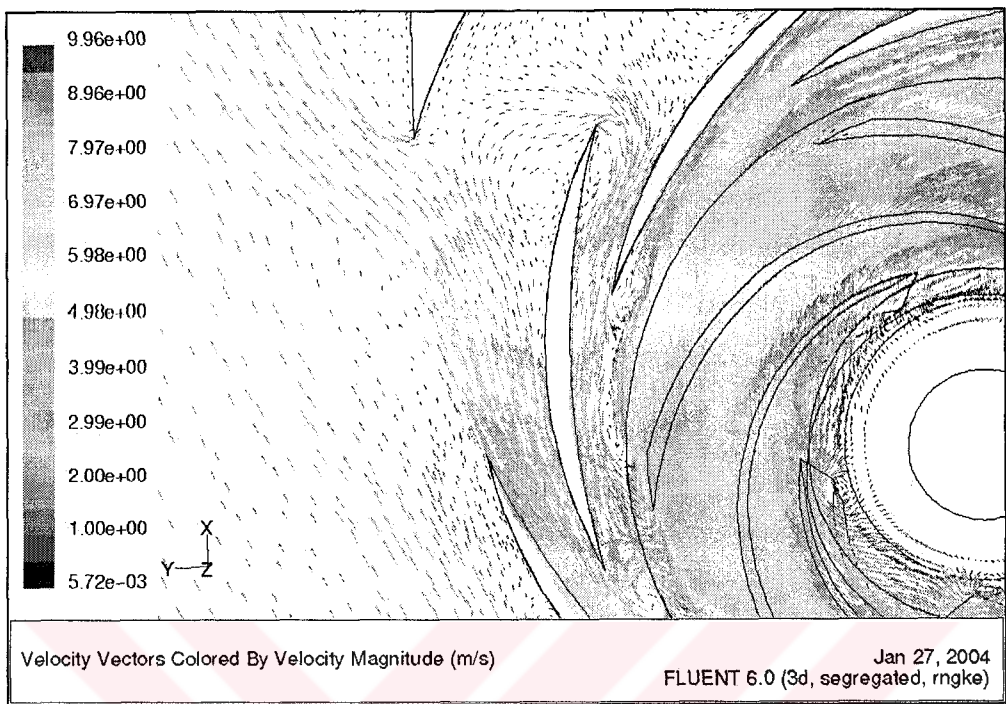
Figure 4.13 a) Contours of static pressure b) Contours of velocity magnitude
c) Velocity vectors d) Contours of vorticity magnitude e) Contours of TKE (for 2.86 l/s) – Case I



a)



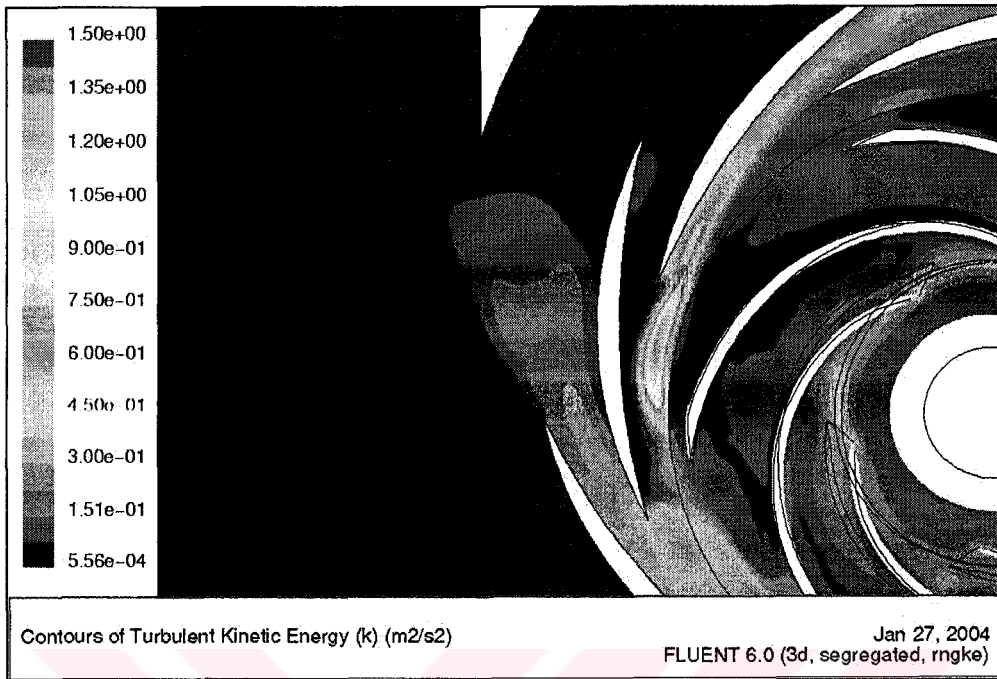
b)



c)



d)



e)

Figure 4.14 a) Contours of static pressure b) Contours of velocity magnitude
c) Velocity vectors d) Contours of vorticity magnitude e) Contours of TKE (for
2.51 l/s) – Case I

4.3 Comparison of Steady and Unsteady Flow

The non-axisymmetric shape of the volute imposes a circumferential distortion of the flow conditions at the outlet of the impeller. The flow structure interactions may lead to vibrations and noise, thereby causing extreme wear that can culminate in structural failures. A better understanding of the complex unsteady flow phenomena in centrifugal pumps is therefore vital to overcome these problems, improve performance and alleviate environmental concerns. In addition; the knowledge base is needed better pump design and for computational model purposes.

Unsteady effects have probably a prominent importance at off-design conditions, especially at part load operating conditions, especially at part load operating conditions. For instance, impeller-diffuser interaction phenomena can modify the stability of the operating characteristics. They affect also the formation of rotating stall-fluid zones of low momentum that are rotating around the machine at a speed lower than the impeller velocity-and they influence the appearance of surge-variation in time of the average mass flow through the machine. In addition impeller-diffuser interactions induce noise and vibrations, and if the unsteady pressure forces created by the interaction generate excessive loads, mechanical damage to the machine can occur. At lower and higher flow rates, the performance, flow rate and pressure of a centrifugal pump become unsteady. This unsteady condition restricts the operating range of the centrifugal pump. Therefore, even if it seems not necessary to account for impeller-diffuser interaction mechanisms for designing highly efficient machines, to assure a stable operation of the machines for a broad range of flow rates, and to improve the reliability of the machines, unsteady phenomena should be controlled. The averaged unsteady flow results obtained for different flow rates in order to analyze the different working conditions inside the pump. With this goal in mind, the pressure and velocity distributions for different radial gaps of the centrifugal pump are numerically obtained.

In the present study, if the pump revolution is considered as 890 rpm, the pump period is calculated as 0.0674 s. This is the time required for one revolution of pump. It is not enough to analyze the flow structure for only one period when

analyzing machines like centrifugal pumps. Because of this reason, initializing the unsteady flow calculation with the steady flow solution, over five impeller revolutions are necessary to achieve the periodic unsteady solution converge. The final numerical results for the comparisons with experimental data in this study were extracted from the unsteady tenth revolution of the impeller.

For unsteady flow calculations the time step, which is related to rotational speed and angular displacement of the impeller between two successive computations, was set to 0.00674 s (10 time step per revolution). The impeller grid rotation is related to this time step and also to the rotational speed of the pump ($n = 890 \text{ rpm}$). Therefore, a complete revolution is performed for each 10 steps. This is done in order to keep frequency resolution well above the blade passing frequency and its first harmonics.

With an increased number of time steps, the computations begin to capture the higher harmonic components, as a result of an improved time wise resolution. The results show some slight difference in the time averaged solutions and suggest that, for this centrifugal pump stage, at least 10 time steps per period is required to capture effect of higher order unsteadiness.

The number of the iterations at each time step was found to affect both accuracy and convergence. Generally, the numbers of the iterations need to be increased with an increase in mesh density or decrease in number of time steps per period. In unsteady flow calculations, it would be usually used the periodicity of flow variables as an indication of convergence. It is known that if the number of the sub iterations at each time step was not sufficient, the accumulation of errors could be lead a stability problem.

The number of iterations has been adjusted to reduce the residuals below an acceptable level or until excessively oscillation does not occur in each time step. In particular, the ratio between the sum of the residuals and the sum of the fluxes for a given variable in the entire cell is reduced to the acceptable values of $10^{-3} \sim 10^{-5}$. In these studies, for calculations, 20 sub-iterations were performed at each real time step. In the case diverge of solution; the number of iterations was increased to 30 or more to obtained converged solution.

In order to obtain unsteady solutions comparable to steady solution and experiments results, a numerical simulation was conducted at three-dimensional pump model. The pump geometries used in steady and unsteady flow calculations are the same from the point of view of the mesh and the geometrical dimensions.

Unsteady calculations were made for 15 different flow rates as made steady calculations at three different radial gap ratios. The comparisons of the solutions were made at the same level of volume flow rates.

Figure 4.15 shows the time histories of velocity, static pressure at inlet, but the time histories of mass flow rate at inlet and outlet for volume flow rate 4.12 l/s during the first five revolutions of impeller.

The $H-Q$ curves obtained from unsteady flow numerical analysis were compared with steady flow numerical analysis. As seen in figure 4.16, steady and unsteady results are agreement with each other. This figure compares the unsteady and steady flow performance curves for the present pump. These curves were obtained at three different radial gap ratios. The differences are not visible; therefore both curves are thought to be valid for calculations. The comparisons of the experimental and numerical results for steady flow and corresponding deviation; good consistency was achieved; but the differences at very small and large flow rates are visible (see figure 4.4). Numerical results of unsteady performance curve were available for comparison but the numerical results were considered to be within a reasonable data range corresponding to experimental results.

The magnitude at off-design flow is apparently larger than that at other flow rates, which is a result of the non-uniform pressure distribution caused by rotating stall in the impeller. The results also illustrate the effect of the unsteady flow, no significant difference was produced in the near design flow region, but at small flow rates the differences are visible as it happened in steady flow results.

Unsteady calculations have been performed with resulting characteristic curves from both approaches fitting quite well with steady results. Transient results have the advantage of better prediction for over load operating points. Further on, displaying time varying results may serve as a real tool for understanding the interaction between impeller and diffuser. Calculations were carried out for unsteady

incompressible flow cases, and the results show satisfactory agreement with well-established the steady numerical results and experimental data.



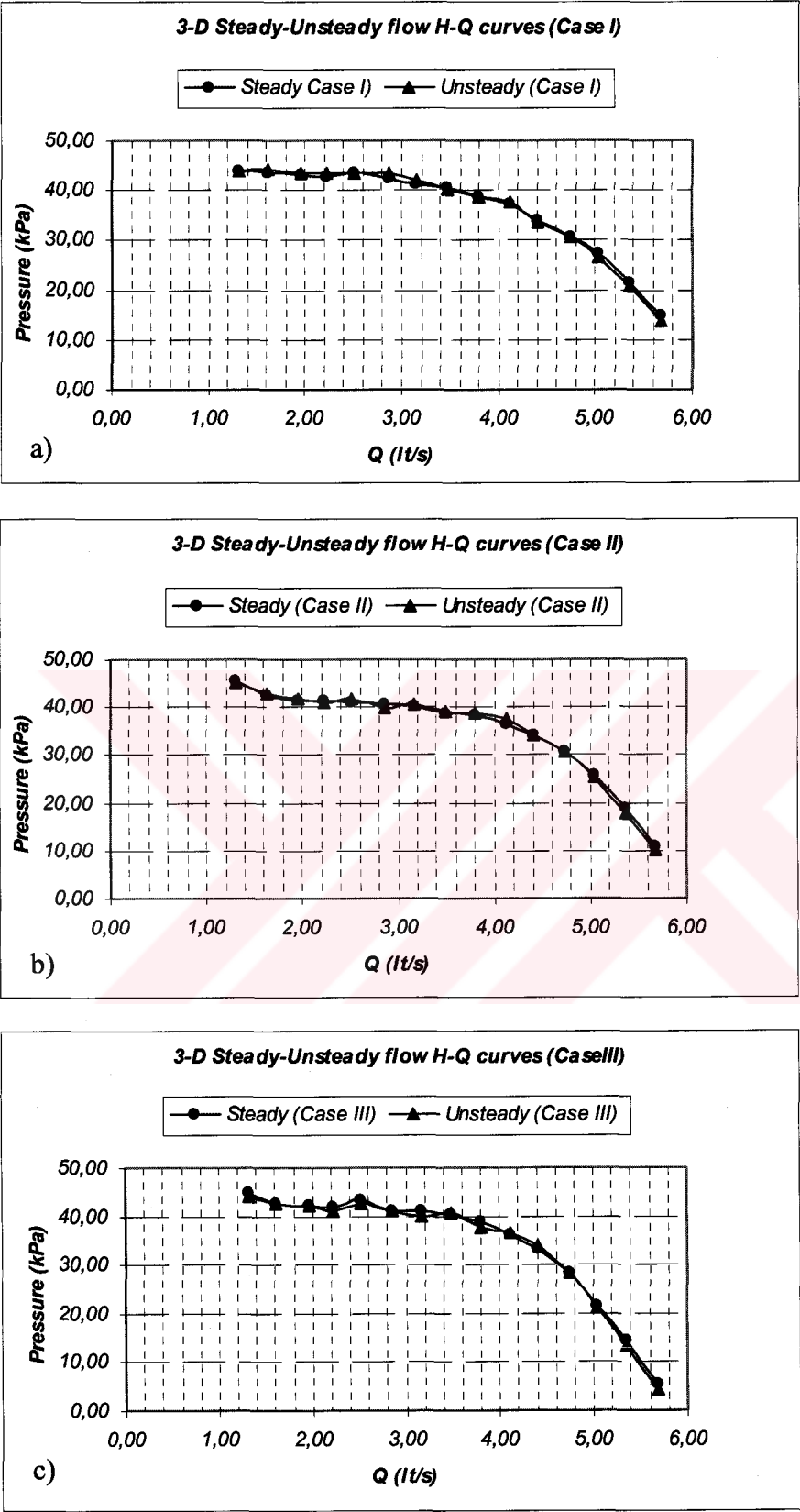


Figure 4.15 Steady and unsteady performance curves for a) Case I, b) Case II, and c) Case III

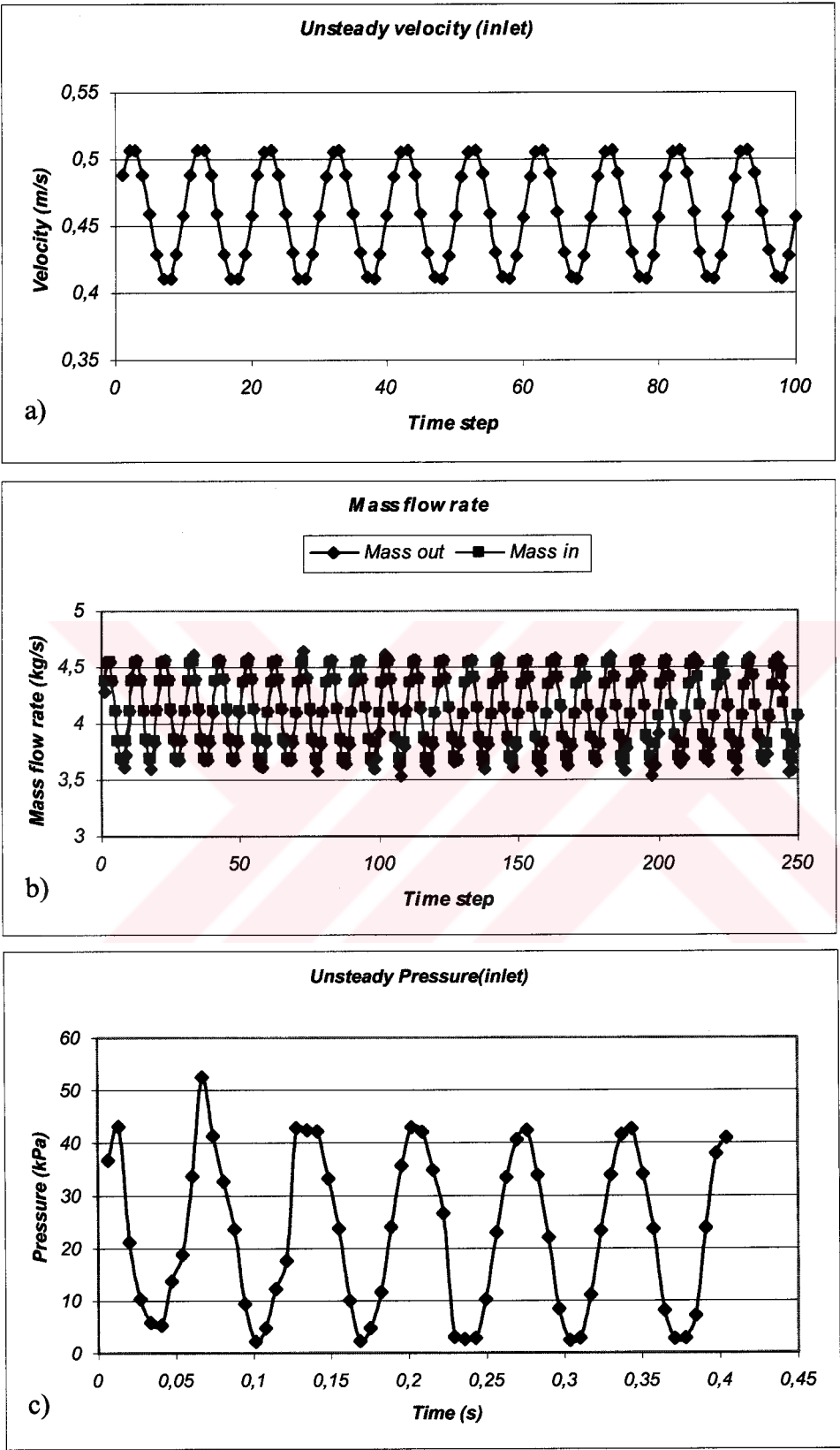


Figure 4.16 For unsteady flow a) Velocity b) Mass flow rate c) Pressure, for 4.12 l/s

4.4 Radial Gap Effect

For a centrifugal pump with a vaned diffuser, satisfying this assumption requires a large gap between the impeller and the diffuser so that the mixing process of the flow leaving the impeller can take place. Otherwise, an unsteady flow at the leading edge of the diffuser vanes represents a source of low efficiency and instability. Furthermore, the internal flow of the impeller can be affected by asymmetric downstream conditions, which results in extra flow unsteadiness and instabilities.

The diffuser is designed by assuming that the flow entering it is completely mixed. In centrifugal pumps, the close proximity of the impeller blades and the diffuser vanes causes strong interactions between the components. The very small distance separating the trailing edge of the impeller blade from the leading edge of the diffuser vane makes it doubtful that the flow leaving the impeller has enough time to mix completely before entering the diffuser. Despite the simplifications used in the design process, the efficiencies reached by modern machines are probably very close to the optimum values.

The vibration is governed by the vortices and wake effects on the impeller and diffuser. The wake causes pressure fluctuations, which in turn give rise to the vibration and noise. The pressure fluctuations are most strongly affected by the radial gap between impeller and diffuser. The radial gap, however, also governs the efficiency of the pump. Hence, there is a tendency towards keeping the radial gap small. The wake effect depends upon the geometry of the impeller vane trailing edge and diffuser vane forward edge. In traditional centrifugal pump, radial gap is suggested is seen to be 5% of the impeller radius (Tuzson, 2000).

Unsteady analysis is suitable in fluid mechanics applications to investigate the flow structure in the pump. However, in some problems time independent approaches can be applied where there is little change with time or no change with time. It can be said that the factor changing with time are very limited since pumps work at very high revolutions. Because of that, steady approaches at pumps are very common. But, especially due to pressure fluctuations at pump outlet the time

dependent studies are needed. The difference between steady and unsteady analysis is that iterative solver solves the problem with respect to divided time slices. In order to solve the problem, because of that the information based on time interval and time slice the problem will be solved is needed. The period and time slices should be determined in fluid mechanics problems with rotational movement. Time dependent and independent applications are seen at pump applications.

At previous part, for different gaps steady $H-Q$ curves were compared with unsteady performance curves, and these curves show that as seen in Figure 4.15, steady and unsteady flow values are quite close to each other. In this part of the thesis, the effects of the radial gap space were investigated computationally at the condition of steady flow assumption in the pump. The internal flows inside the centrifugal pump at off-design and design point conditions were analyzed by solving the fully 3D Navier-Stokes equations coupled with the RNG $k-\varepsilon$ two-equation turbulence model. In totally, three range of radial gap distance were run to see the level of effect on the pump performance as explained in previous section. The remarkable effect of different radial gaps on the flow of impeller-diffuser interactions are compared in this part. The impact of changing the radial gap on the static pressure contours, velocity contours, velocity vectors, vorticity contours and TKE distributions are illustrated for different radial gap ratios (these results, unless clearly stated in the caption, all figures in this paper contain phase averaged results).

It seems that pump geometries produces a deviation from the axisymmetric flow distributions even for small gap differences from nominal flow rates. This might lead to important radial forces that could limit the use of this particular design. The flow in the pump is changed with the effect of the flow rate. Because the effect on flow structure of volume flow rates decreases, the effect on the flow of radial gap ratios were investigated at two volume flow rate about design point value. Thus, the selected present flow rates, 3.48 and 4.74 l/s, is close to the numerical design point of this pump.

The computational results deal with static pressure contours, velocity contours, velocity vectors, vorticity and TKE distributions were shown for 3D

geometry in the mid-plane of pump (4 mm), half of volute thickness at radial direction, at different flow rates.

Three radial gaps, namely Case I, Case II and Case III as related to the impeller diameter were chosen for the present study. The larger gaps were achieved by moving the diffuser blades. If the radial gap is increased by trimming the impeller, its trailing edge geometry changes. In this part of the study was performed on the condition of steady, incompressible on different 3D pump geometries with different at design flow rate level. Figure 4.17 presents $H - Q$, efficiency curves and error rates of the pump at each radial gap ratio. The calculated results are compared with experimental data and the comparison shows good agreement.

The fifteen flow rates level were simulated, the numerical nominal flow rate was 4.12 l/s and the range of volume flow rate $1.31 \leq Q \leq 5.68$ l/s, the head obtained in these range of flow rates points matches the experimental results (see figure 4.3). Analyses was made at 15 different flow rates, however the results relating to the below design flow rates which is less than nominal flow value (3.48 l/s) and 15% higher than nominal flow rates (4.74 l/s) in the case of above design flow rate comparing to the design value. It has been commented before; there are great differences in the pump performance having small changes in the radial gap geometries. However, the pump characteristics with various models are reported in this part of the work. It seems that the best curve is for the geometry with the original radial gap geometry stated as Case I. The performance and efficiency correspondence are also very good though they are little worse corresponding to the higher flow rate. This deficiency occur due to the velocity inlet condition being inadequate for higher flow rate and also depends on the hydraulic losses in the pump. The jet-wake flow becomes strong and the turbulent kinetic energy is also increased in most of the passage where the wake is accumulated. This is believed to be caused by un-stability of the wake formation, which is also meandering of the secondary flow around the diffuser passage. The blade trace effect is also discernable from vorticity distributions, which is higher at the inlet of the impeller and the diffuser blades as seen in figure 4.18. In case one 20% gap ratios above design flow rate level

the jet has strong effect while TKE level has also accumulated around diffuser leading edge. The static pressure also affected.

A common feature of the static pressure distribution at low flow rate levels reveals that the pressure increases in the radial direction. The curvature of the tongue produces negative slope as the flow approaches to the exit. As the mass flow rate decreases, the jet flow nearly disappears as shown in figures 4.13 and 14b. The losses in the vaneless part of the pump and tongue make reduce the pump efficiency at the design flow rate and more deficiency is observed at lower mass flow as seen in figure 4.17.

Figures 4.26a and b show the pressure contours from the hub to the shroud at the design volume flow rate, indicating that the pressure distribution on the hub of the impeller is quite uniform and low. The low pressure region is observed on the suction side, especially in the region near the outlet of the impeller.

It can be seen that the radial velocity on the outlet section of the impeller is obviously larger in the region near the suction side of the blade than the region near the pressure side under the design operating condition as seen in figures 4.27a and b. Figures 4.28 and 29 show the static pressure and velocity contours on 3D impeller blades and impeller. Figure 4.29 shows a 3D view of the pressure over the shroud, blades and part of the volute for the nominal flow point. It can be seen that pressure in the hub side is higher than in shroud, due to the transition of the flow from the axial to the radial direction. Pressure distributions over the suction and pressure side of the blades are clearly observed (figure 4.28). Near the shroud, the channel between blades is getting closer and less curved increases the pressure gradient. Also the flow impact over the tongue varies from the center-where the flow comes directly from the impeller-to the sides. The fluid pressure starts to increase after the pump inlet and it continues to increase as it flows after the diffuser.

Turbulence Kinetic Energy values which is the parameter defining turbulence, reach the maximum value at the trailing edge of the impeller. The reason for that is the separation of the flow due to the thickness of the blade and the reduction in energy intensity because of the pressure decrease. These values decrease through the impeller, but increase through the diffuser vanes. Further more, they

continue to increase from the diffuser vanes to the outlet. These variables reach the maximum value at trailing edge of the diffuser vanes near the tongue and show little fluctuations at the pump outlet regions as seen in figure 4.18e. The reason for this is the increase in kinetic energy due to the decrease in distance between tongue and diffuser vane. The pressure at suction side of the impeller blades is lower whereas it is higher at the pressure side (figure 4.18a).

It is commonly known that as the impeller rotates, the fluid that is drawn into the passage of the impeller blades accelerates as it is forced radially outwards. On the other hand, as the fluid moves through the passage of the diffuser blades the kinetic energy of the fluid is being reduced while the pressure energy is further increased. In order to reveal the detailed information about the flow characteristics one must analyze properties of the fluid countatively. The jets which characterize the design region occur at the inlet of diffuser vanes. As the fluid flows through diffuser vanes, the kinetic energy of the turbulence decreases since velocities decrease. Especially, if velocity vectors around the periphery of the tongue observed, it is seen that the fluid is stabilized in the between diffuser vanes and tongue. Also, it is seen that pressure is high at the same region. The jet between diffuser vanes is seen clearly. The fluid moves to the pump outlet due to this jet flow. The increase in loses of hitting and hurling in tongue region can be observed from TKE contours as seen in figure 4.18b.

The results of computation deal with static pressures, velocity contours and velocity vectors, vorticity and TKE are shown in figure 4.19 at design volume flow rate (4.12 l/s) in Case I. The inlet radial velocity is relatively uniform and exit radial velocity distributions indicate that the boundary layers developed on the diffuser vanes, which is in agreement with Sinha's (2001) experimental observation corresponding to level of flow rate at 4.12 l/s and 4.74 l/s. To confirm the validity of the numerical method, for design flow rate of 4.12 l/s the comparison of the numerical and measured velocity contours is shown in figure 4.19b. Increasing flow rate, increases magnitude of velocity and velocity dependent parameters in volute and also kinetic energy increases too. Maximum velocity at blade edges and low velocity at pump inlet are observed (figure 4.19e).

The static pressure in the impeller passage increases gradually from inlet to exit of the impeller and drops rapidly at the suction side of the impeller blade. That means, the static pressure increases also rapidly at the pressure side and falls very slowly towards to suction side. In general, the static pressure changes very little compared to the exit part of the impeller blades circumferentially. In the case of the diffuser vanes, there are some non-uniformities within the vanes which is inherited from the impeller-diffuser gap distance as seen in figure 4.19a. Similar observation could be seen at velocity contours presented in figure 4.19b and velocity vectors graphs presented in figure 4.19c. The jet-wake type flow phenomena exist within the diffuser blades where high velocity distributions on the pressure side of the diffuser vanes exist while the low energy fluid covered on the suction side. Turbulent kinetic energy distribution displays that the highest value accumulated within the diffuser vanes which expected due to non-uniformities mainly observed within the vaneless space of the pump as illustrated in figure 4.19e.

The turbulent kinetic energy magnitudes are higher within the impeller and diffuser gap. This is traced especially at higher level of flow rate from turbulent kinetic energy contours in figures 4.11e and 22e displaying below design and above design flow rate levels respectively. The blade trace is also affected by the flow rate level which accumulates inside the gap like a strip. The losses in the vaneless part of the pump and tongue cause the pump efficiency to reduce.

The flow characteristics at lower flow rate levels have been examined computationally to analyze stalled flow at the pump. Figure 4.11b shows flow velocity contours at flow rate levels of 3.48 l/s. In this case the flow is well below design conditions. The different inflow patterns result in different flow structure near the tongue, even though the variation of the diffuser inlet patterns is influenced by the tongue itself. Near the tongue region, each diffuser passage displays a reverse flow from the diffuser discharge back to its throat. This is clearly discernable from velocity vectors around stalled conditions (figure 4.11c).

The static pressure around the volute is quite uniform for the flow rates in the range of the best efficiency point. However, it exhibits non-uniform pressure distributions either a maximum for low flow rates or a minimum for high flow rates

in the region of tongue. These non-uniform pressure distributions are related to the acceleration (high flow rates) or deceleration (low flow rates) of the fluid along the volute for off-design conditions. The non-uniformity of the pressure at the tongue is seen from the flow in the rotating impeller as an unsteady boundary condition, thus it affects the resulting fluctuations in the pressure field. It is organized in a jet-wake structure. The wake is characterized by low relatively velocities and is localized in the suction side. At this flow rate, it seems that presence of the vanes has only a limited effect on the impeller flow structure, except when the blades suction sides are facing the diffuser vanes.

At the impeller discharge, the flow leaving the impeller is affected by the presence of the diffuser. At the leading edge, at the suction side of diffuser vanes, the flow is rapidly mixed. Whereas, when approaching the diffuser throat, the flow still shows that its periodic characteristic observed at the impeller outlet. This indicates that the diffuser is subjected to unsteady inlet conditions that can alter its performances.

The equilibrium between pressure forces and Coriolis forces in the main flow is lost in boundary layers at hub and shroud surfaces, leading to a secondary flow in the boundary layers directed from pressure to suction side. In the main flow a reverse secondary flow direction is observed. The secondary flow doesn't seem to influence the static pressure distributions.

The static pressure is little affected by decreasing the gap distance between the impeller and diffuser in low flow rates. At the design flow rate, comparing with the above flow rate level the low TKE has been observed and pressure is also relatively moderate. At the below design flow rate, the jet still dominates but reduces the effect. TKE is also observed relatively higher especially within the diffuser.

Decreasing the gap ratio has a significant effect on TKE. Comparisons of figures 4.19e, 21e and 24e for 4.12 l/s indicate that the low level of TKE is obtained in Case I while the high levels of TKE obtained in Cases II and III with respect to radial gap distance. In the case of above flow rate level, when the radial gap distance is decreased to 15% the flow starts to squeeze between the impeller and diffuser. As can be seen from the velocity contours, separation starts on the diffuser wall far from

the leading edge. At the design point, the higher pressure contours are observed in the region between tongue and diffuser. Higher TKE levels observed where the jet accumulated. During the below design flow rate, TKE levels are unstable especially on the jet trace. The static pressures are also un-uniform around the leading edge. The separation point on the diffuser walls are expanded towards to trailing edge.

This is believed that the narrow gap is associated the jet remains “stuck” in the corner as the blade moves behind the tongue, and coincides with the wake when it arrives the tongue. Within the diffuser deterministic kinetic energy is particularly high along the convex side of the vane as shown in figure 4.19c and it extends well downstream of the vane along its wake. This phenomenon is also demonstrated by comparing the corresponding circumferential velocity distributions shown in figures 4.21b and 24c and vectors shown in figures 4.21c and 24c. The impact of these changes on the static pressure is demonstrated. There is no reverse flow observed in the case of design flow conditions which is possible effect that the flow rate level far away from the stall and off-design conditions. When the diffuser blades are located at Case III the flow around it and resulting tongue effects are dominated by non-uniformities in the outflux from the diffuser. Thus, it is likely that any reduction in the jet-wake phenomenon would have a direct impact on decreasing the radial gap distance. On the above level of flow rate, the static pressure distributions are non-uniform around the leading edge of the diffuser. TKE level has also observed higher than previous cases. The flow squeezing affected the resulting of uniformities in the impeller. The jet is getting smaller in size. Design point on the case of 10%, all of the diffuser walls has been affected by TKE. And squeezing flow also gives effect on the pressure due to small gap clearance. The jets moderate and, but the wake expands. Below design point small jet observed, mainly reverse flow dominates around the tongue region.

For design flow rate level, comparisons of the static pressure contours in the case of Case I, Case II and Case III gap distance (in figures 4.19a, 21a and 24a) which is evident that near the tongue and at the exit, the static pressures are considerably smaller, whereas somewhat far upstream of the tongue they are slightly higher. Similar to the impact of the blade, the pressure decreases everywhere, in the

exit region when the vane lines up with the tip of the tongue. Pressure maxima occur simultaneously when the blade is past the tip of the tongue and the vane is still far upstream of it.

The dominant effect of gap distance on the static pressure and turbulent kinetic energies is evident within the impeller, in the gap between the impeller and the diffuser, and within the diffuser. As is indicated from the pressure fluctuations and also known from previous literature (Dean and Senoo (1960) and Dong et al. (1992a,b)), the radial velocity at the pressure side of the impeller blade is higher. Note that the high radial outward velocity on the pressure side is due to the fact that the absolute radial velocity corresponding to the jet in this region is greater than the passage averaged flow (which may be interpreted as a steady outflux from the impeller). Therefore, the velocity distributions (figures 4.21b and 4.24b) that show a reverse flow from the pressure side to the suction side of the blade highlights the jet-wake phenomenon that is the characteristic of flows in centrifugal pumps. It is also evident in the sharp increase of the phase averaged velocity (primarily tangential) across the wake of the impeller blade. The same flow characteristics are also observed heading to exit jet flow, wake flow as the impeller wake along the convex side of the diffuser blade, and maximum turbulent kinetic energy occurs along the jet flow such as shown in figures 4.21e and 24e for Case II and Case III respectively. Thus the transition between the high and low velocity regions also occurs along the wake line of the impeller blade.

The main objectives of the present study are to investigate the flow behavior around the tongue in order to compare these numerical results with experimental results of Sinha (1999). Through out the range of volume flow rate that are considered in the present study, a substantial acceleration of flow and considerably low speed regions are observed in the vicinity of the tongue.

The separation point is placed somewhere downstream of the volute tongue in the discharge direction, which agrees with the expected trend, as presented in figures 4.11b, 22b and 25b.

In the region close to the tongue, the flow behavior is quite different comparing other part of the pump, although the effect of the diffuser on the flow of

this region is clearly seen. It is found that there is a pressure increase in the region in between blade and tongue. For low volume flow rates $1.31 \leq Q \leq 4.12$ l/s the numerical performance curve is agreed well with the experimental results for all gap ratios as indicated in figure 4.17a, but after certain values of volume flow rate a change in performance curve is quite rapid. Agreement of numerical results with experimental results is deteriorated gradually when volume flow rates is increases beyond 4.40 l/s. In other words, there is also disagreement between experimental results for the volume flow rate in the range of $Q \geq 4.40$ l/s.

In general, the efficiency of the traditional pumps is more than 90%. In the present work a pump parameter as selected for special purposes. Although the experimental efficiency 54% for the design point volume flow rate (4.74 l/s) as seen in figure 4.17b. It is aimed at studying the structure of the flow as well as the performance of the flow. These efficiency curves are presented in figure 4.7b. The efficiency increase gradually where the flow rate level starts at $Q = 1.31$ l/s to $Q = 4.40$ l/s. On the other hand discrepancies between experimental and numerical values of efficiency increase rapidly starting from the $Q = 4.40$ l/s. The disagreement is substantial at $Q = 5.68$ l/s. A significant change of flow characteristic for higher volume flow rate causes over a discrepancy in between numerical values for different radial gap ratios. Uncertainty of numerical work of efficiency is presented in figure 4.17c. First of all, assuming that; there is not any error in the experiment results, but in the case of below design level of flow rate, uncertainty limits varies between 6% and 0.5% and at the design point, the disagreement is about 0.5% between experimental and numerical results. As indicated before for higher values of volume flow rate the rate of error increases substantially.

The effect of different gap ratios can be clearly seen for comparing three radial gap ratio configuration comparing the pressure drop level, a higher lack of uniformity are seen in figure 4.17.

The results indicate that in most cases smaller radial gaps are leading to complex flow field at diffuser vane exit and a lower diffuser pressure recovery

resulting in lower pump efficiency. On the other hand, impeller efficiency is slightly deteriorated by decreasing the radial gap for $1.31 \leq Q \leq 4.40$ l/s.

Figure 4.30 shows a 3D view of the pressure distribution around the shroud, through the blades and along the volute for the nominal flow point. It can be seen that pressure in the hub side is higher than in the shroud due to the transition of the flow from the axial to the radial direction. Also the flow impact over the tongue varies from the center to the sides.

Pressure distribution over the suction and pressure side of the blades is clearly seen. Near the shroud, the channel between blades is shorter and less curved, increases the pressure gradient. The pressure changes are visible the circumferentially because of the tongue interaction: pressure at the channel outlet decreases, but suddenly rising when the channel confronts the tongue. These differences are larger for higher and lower volume flow rates.

The impeller wake with high level turbulent kinetic energy moves out of the impeller and finally is discharge and is chopped by diffuser vane when it approaches the leading edge of the diffuser vanes. Then the wakes decay gradually in the diffuser passage becoming insignificant after passing through the diffuser passage, as can be seen in figures 4.11e, 18e and 19e.

As it will be demonstrated, the flow structure changes within the vane passage, caused by impeller-diffuser interaction. The variations in flow structure and turbulence occur in the boundary layer on both the pressure and suction sides of the diffuser vanes.

A jet-wake structure occurs near the outlet of the impeller for each of the two off-design flow rates and the location and size of the jet-wake structure seem to be dependent of the flow rate (figures 4.11b, 18b and 19b).

The effect of pressure drop at smaller radial gap ratio is also discernable in pressure contours by comparing for all three pumps geometries. Jet-wake flow type is traced at velocity vector but the effect diminished by the gap distance, which the flow squeezed within this part of the pump. In this region, the kinetic energy level is displayed to be higher in magnitude. Also, the level is seen to drop significantly as the gap increases.

At off-design conditions, the distortion of the velocity is exaggerated from the inlet to the exit. This indicates an axial mass migration from the diffuser exit to the impeller at lower volume flow rate (see figure 4.11b). The exit velocity distributions show that the thicker boundary layer on the suction side of the passage has merged with the thinner boundary layer on the pressure side. The jet exists at higher mass flows with no reverse flow being observed at the exit. The large reverse flow distributions show that the lower flow rate forces the flow to turn back into the passage which separates near the trailing edge of the lower diffuser vane.

Also a strong peak of the positive pressure can be found on the leading edge near the shroud. The similar behavior mentioned above can also be observed under the off-design operating conditions.

It is interesting to notice that a strong flow along the leading edge of the blade from hub to shroud can be found, which results in a strong inlet reverse flow in the suction region of the impeller, especially in the low flow rate. The lower flow rate shows a tendency to separation in the suction side while this tendency happens in the pressure side for the higher flow rate. This effect is more evident in the hub section than the shroud one, where relative velocities are less influenced by the flow rate.

TKE was observed at leading edges of impeller blades. But a clear symmetrical result hasn't been reached in 3D analysis. The reason for that is an axis symmetrical distribution occurs due to the flow angle of the fluid going into the diffuser vanes is regular.

In most cases the flow field at the exit the diffuser vane exit is more homogeneous at smaller radial gaps indicating a much better diffusion. The reason is that a decrease of the radial gap is leading to unloading of the typically highly loaded vane at diffuser vane exit is hardly affected by the operating point, but determined by the radial gap possibility due to an unsteady effect at diffuser inlet.

The effect of the radial gap ratios is then considered. For this purpose, averaged pressures at different flow rates are presented here in terms of 3.48, 4.12 and 4.74 l/s flow rate levels respectively. The pressure is considered for all radial gap ratios and three different flow rates (below, design and above). The selected design flow rates level is very close to the numerical design point of this pump.

It is identified that impingement of the wake and its associated vorticity field on the tip of the tongue are primary causes the pressure fluctuations on flow inside the pump. The existence of a jet ahead of blade and a wake behind of the blade are clearly evident in the velocity distributions, irrespective of radial gap geometries. The wake is also clearly marked with vorticity train along its leading edge. The jet also bounded by pair of vorticity peaks.

Generally, the region, where the efficiency is maximal; on the pump efficiency curve is desired to be larger. The reason for that is pump should operate in different flow rates without changing efficiency. When obtained results are compared with experimental results, at low flow rates better agreement is seen for all radial gaps, as shown in figure 4.17. The differences in between experimental and numerical curves increases as the flow rate increased. The reasons for the differences in between experimental and numerical curves can be explained as follows:

- The cavitation effect at high flow rates increases and this can't be modeled numerically.
- Used turbulence model doesn't meet the requirements.
- Losses at high velocities increase and this can't be modeled numerically.

Efficiency curves have to be obtained in order to analysis to design points. Pressure values are used for obtaining efficiency curves. Efficiency value is calculated as the ratio of the product of pressure and flow rate to the motor power.

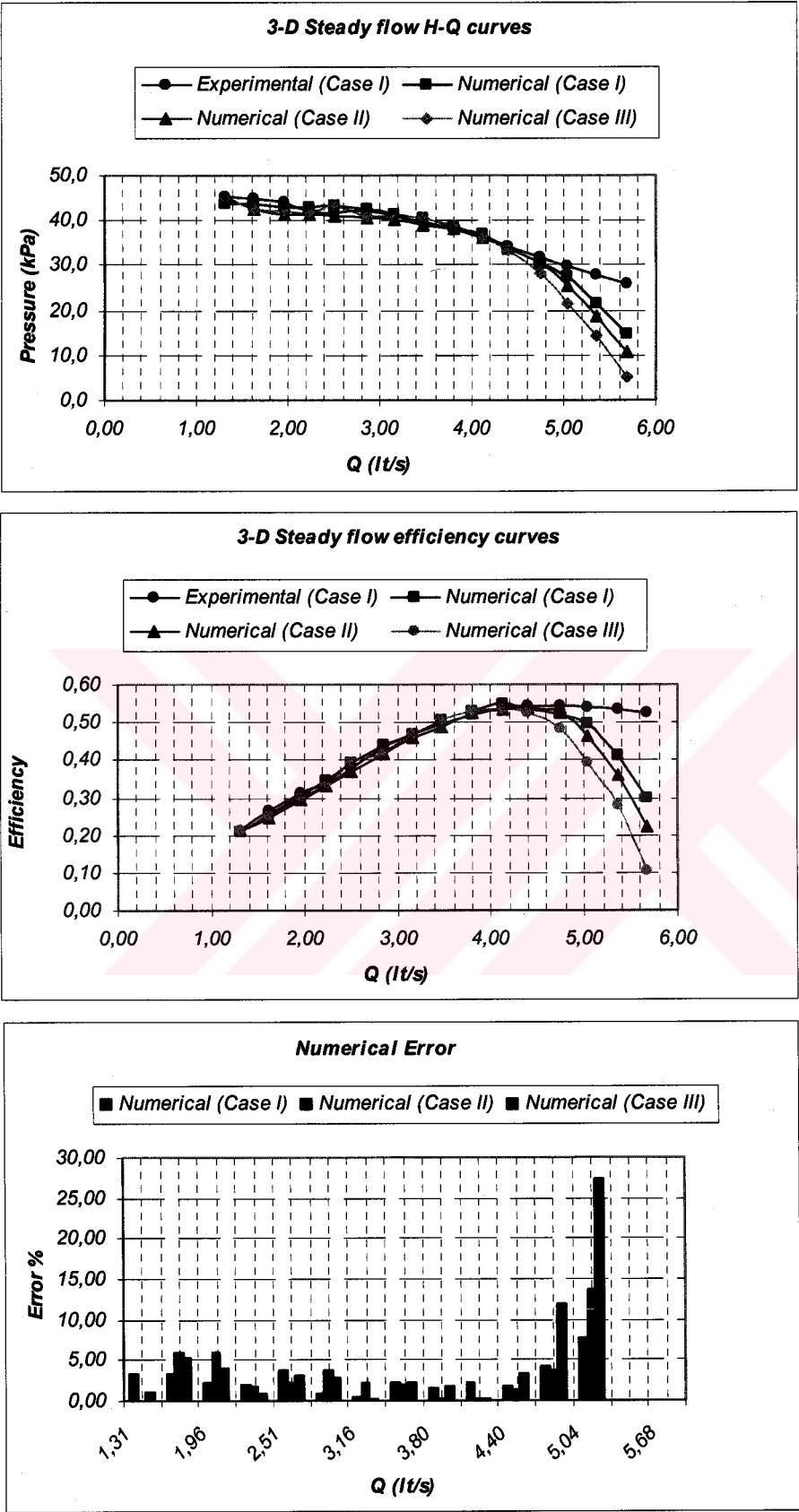
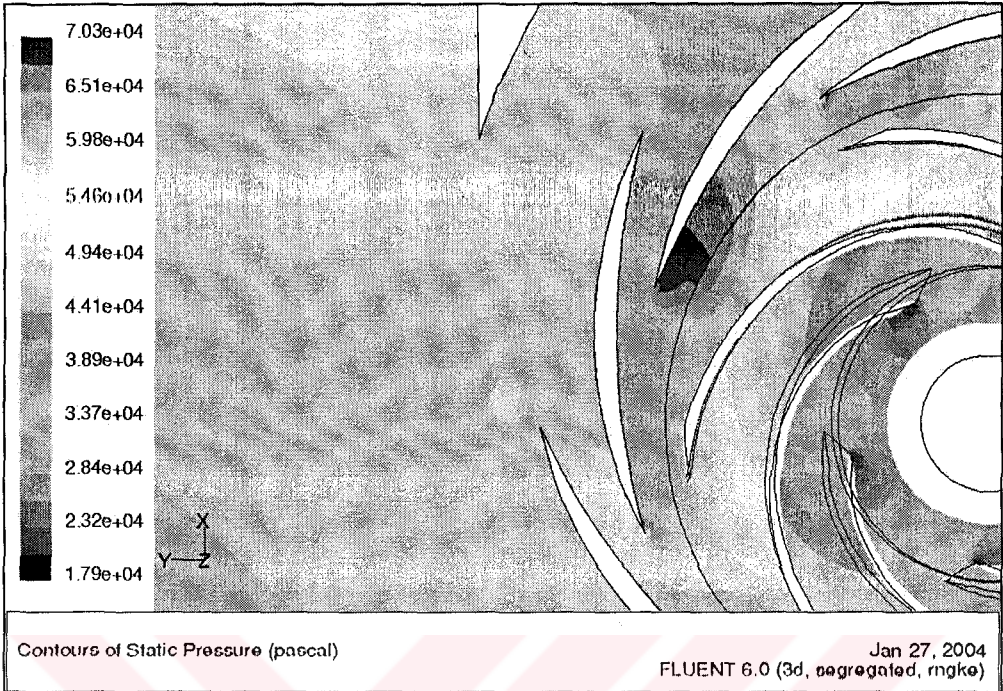
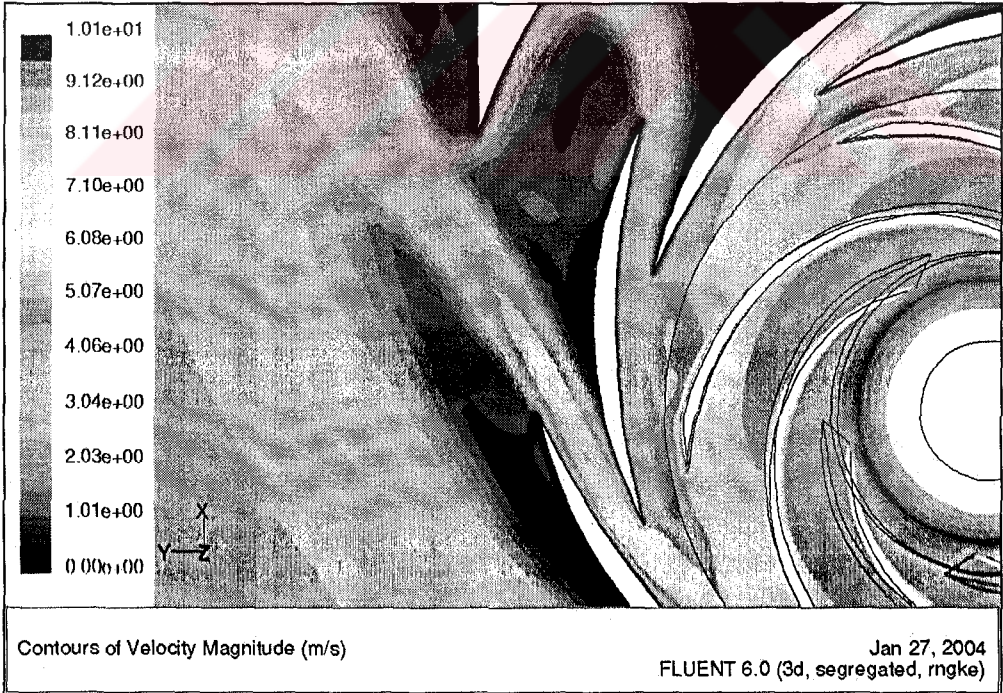


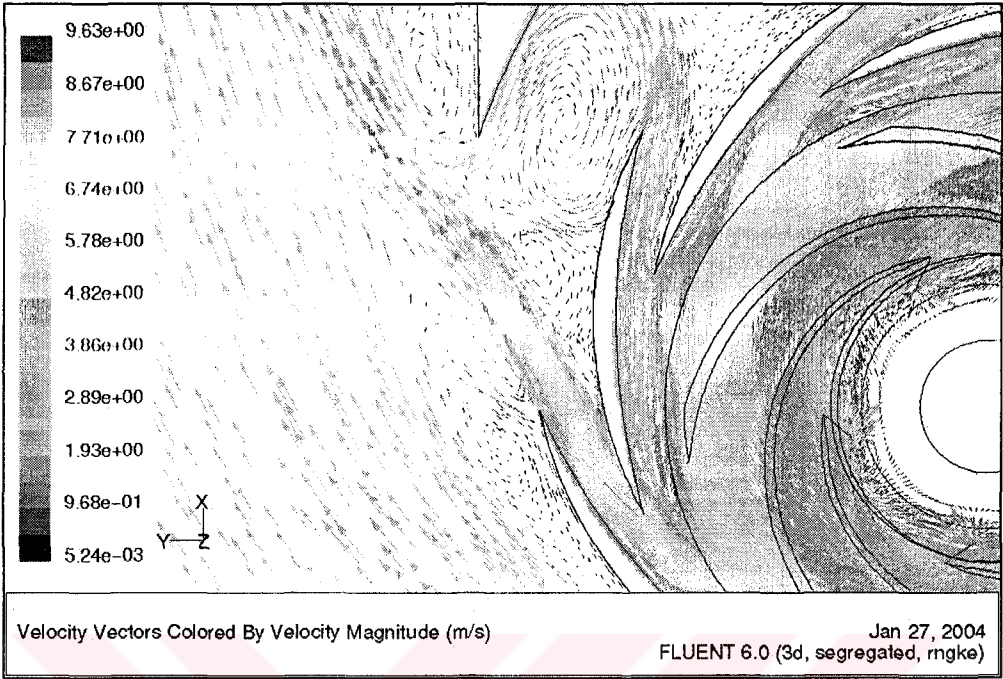
Figure 4.17 a) $H - Q$ curves b) $\eta - Q$ curves c) Numerical Error



a)



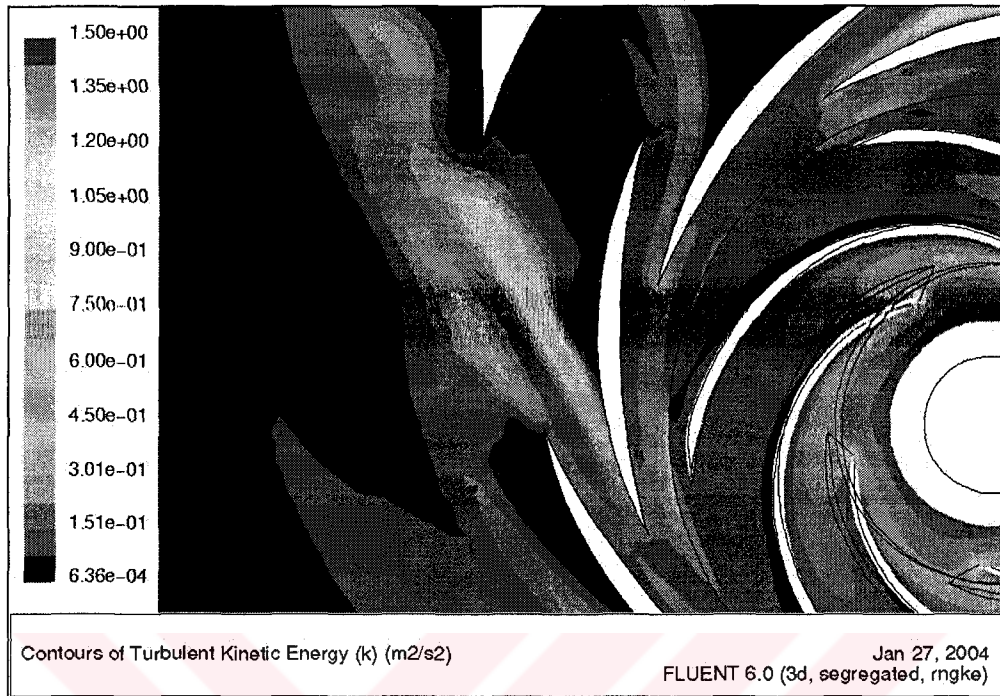
b)



c)

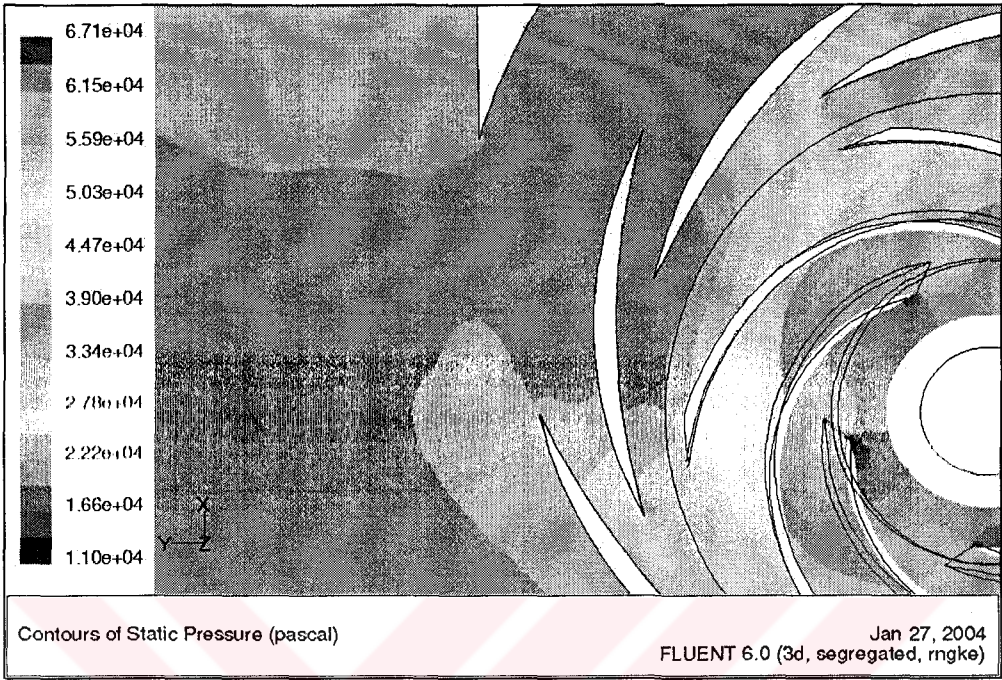


d)

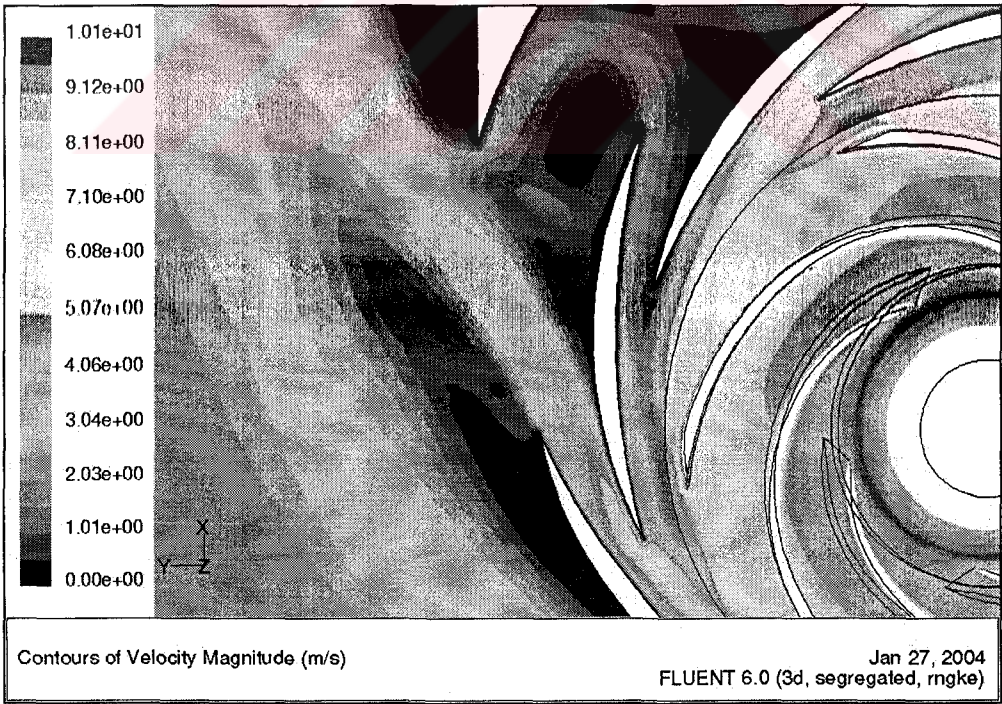


e)

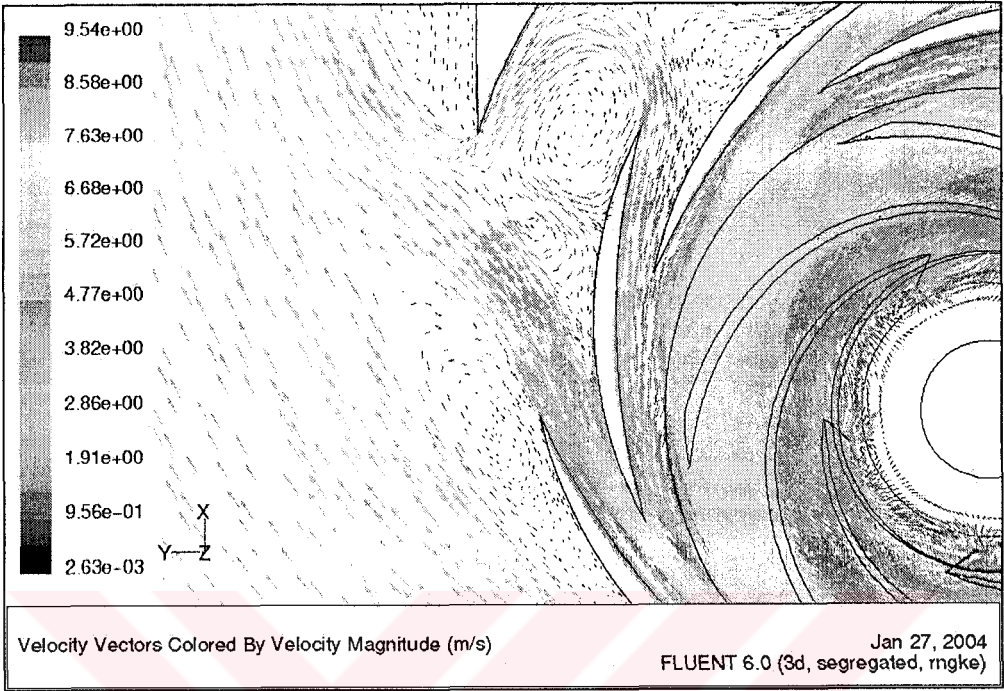
Figure 4.18 a) Contours of static pressure b) Contours of velocity magnitude
c) Velocity vectors d) Contours of vorticity magnitude e) Contours of TKE (for 4.74 l/s) – Case I



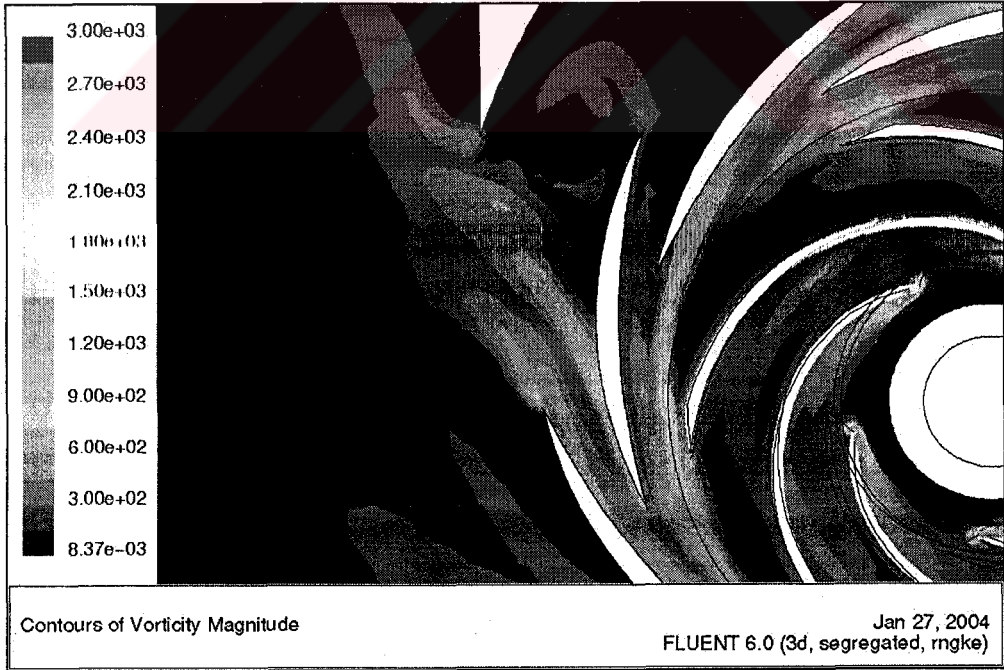
a)



b)



c)

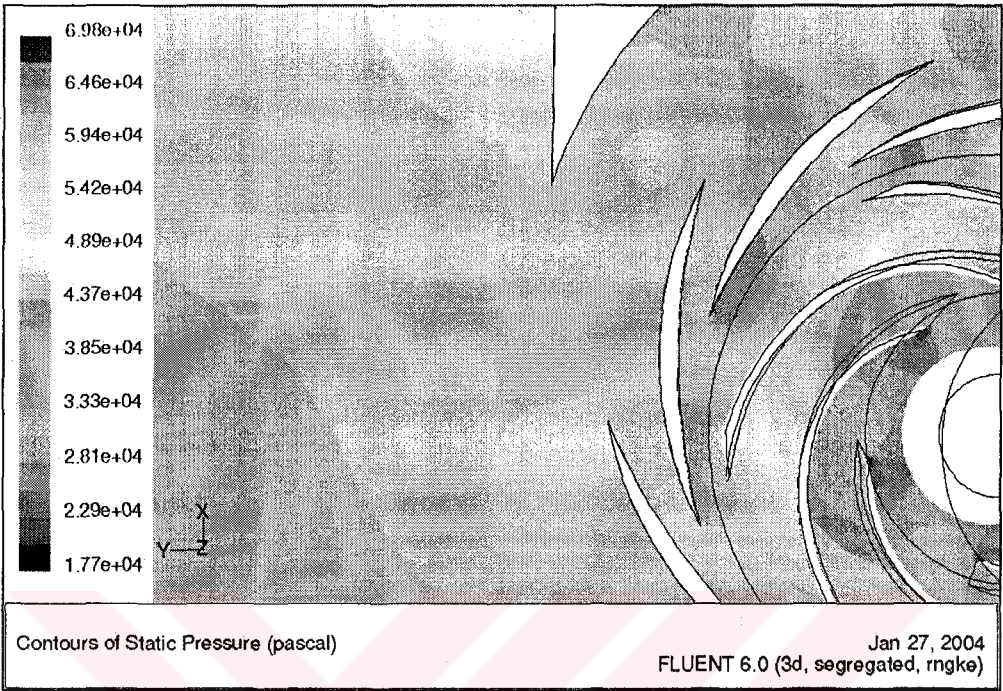


d)

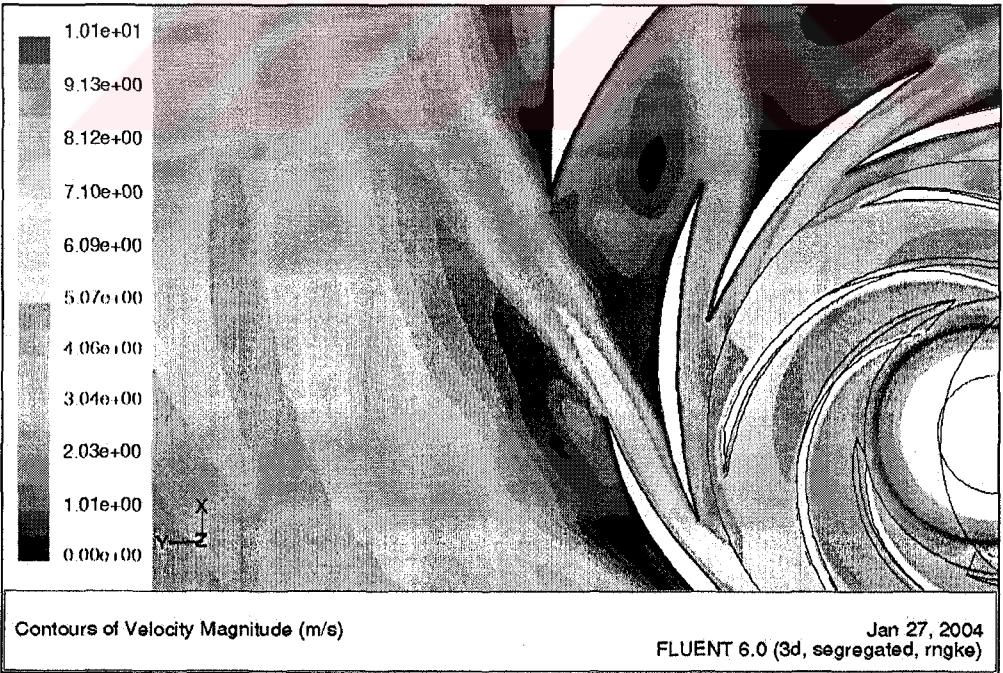


e)

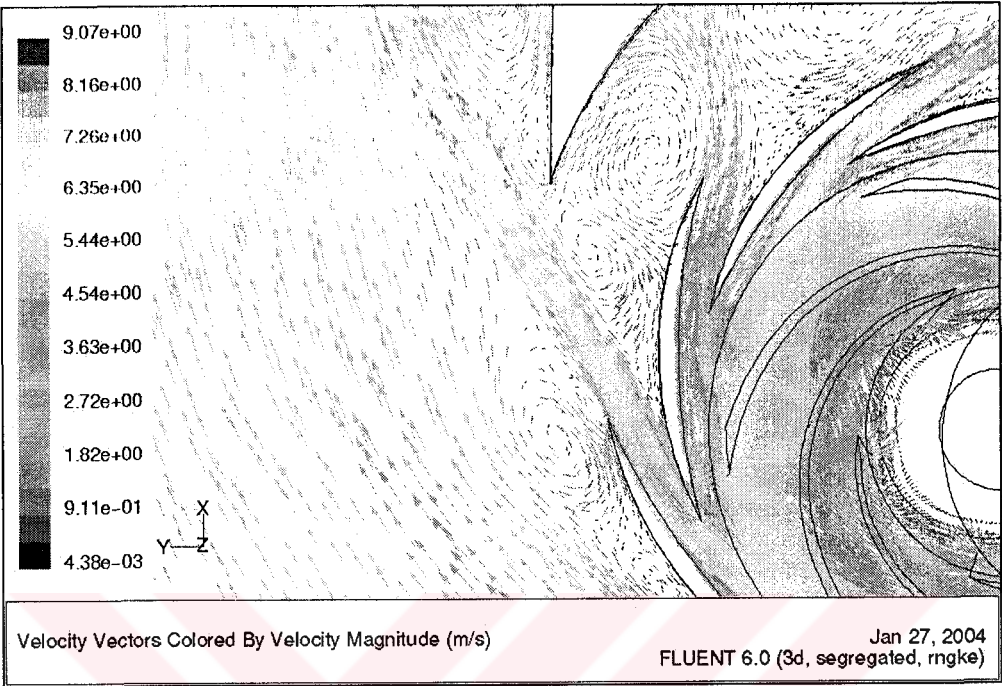
Figure 4.19 a) Contours of static pressure b) Contours of velocity magnitude
c) Velocity vectors d) Contours of vorticity magnitude e) Contours of TKE (for 4.12
l/s) – Case I



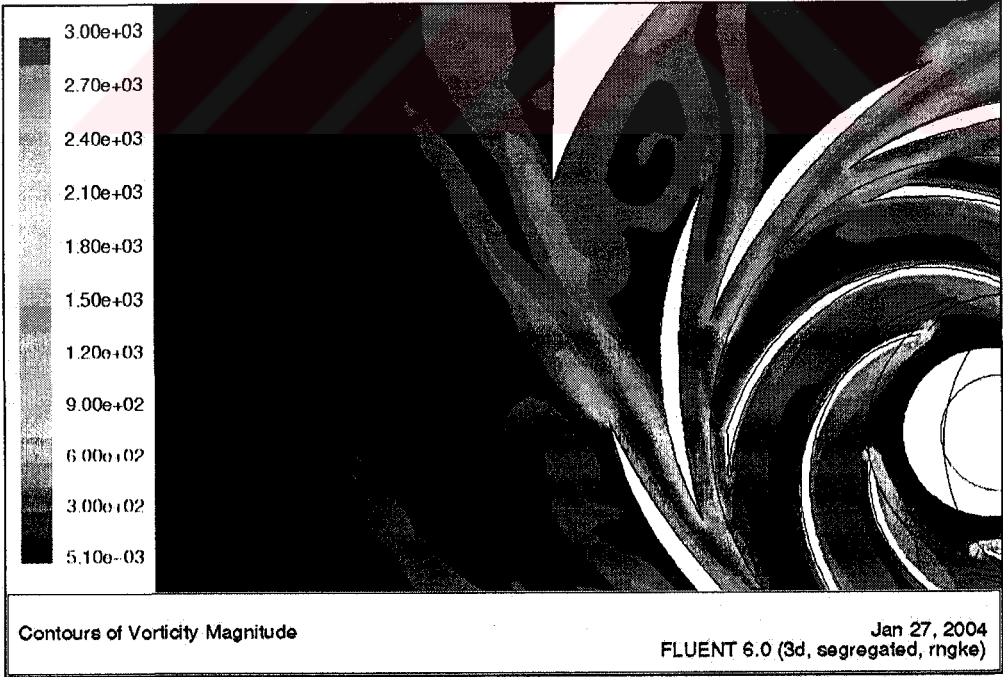
a)



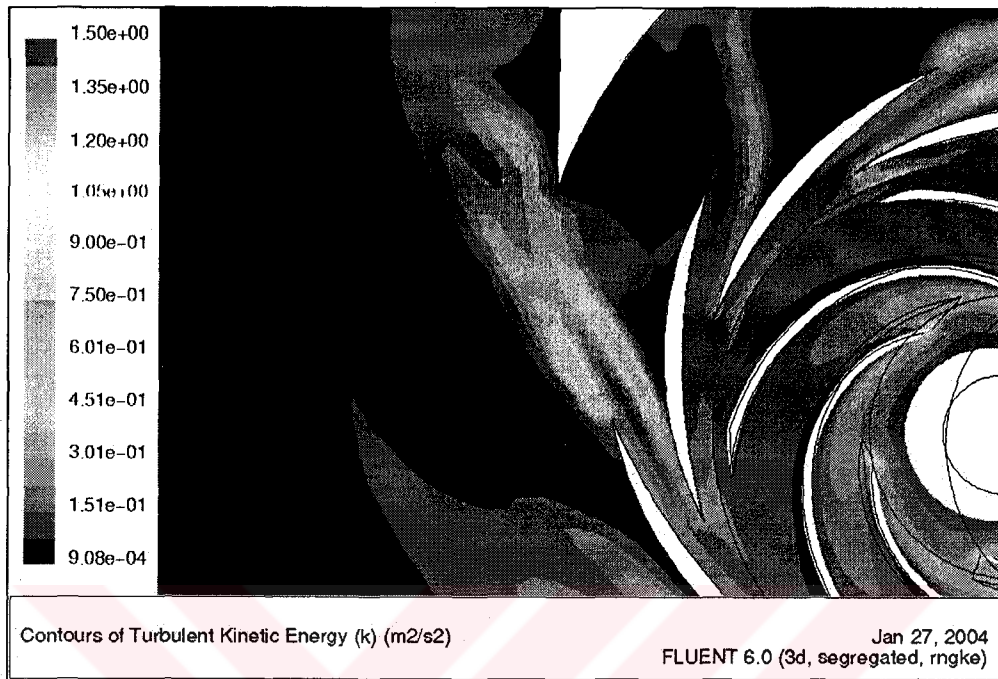
b)



c)

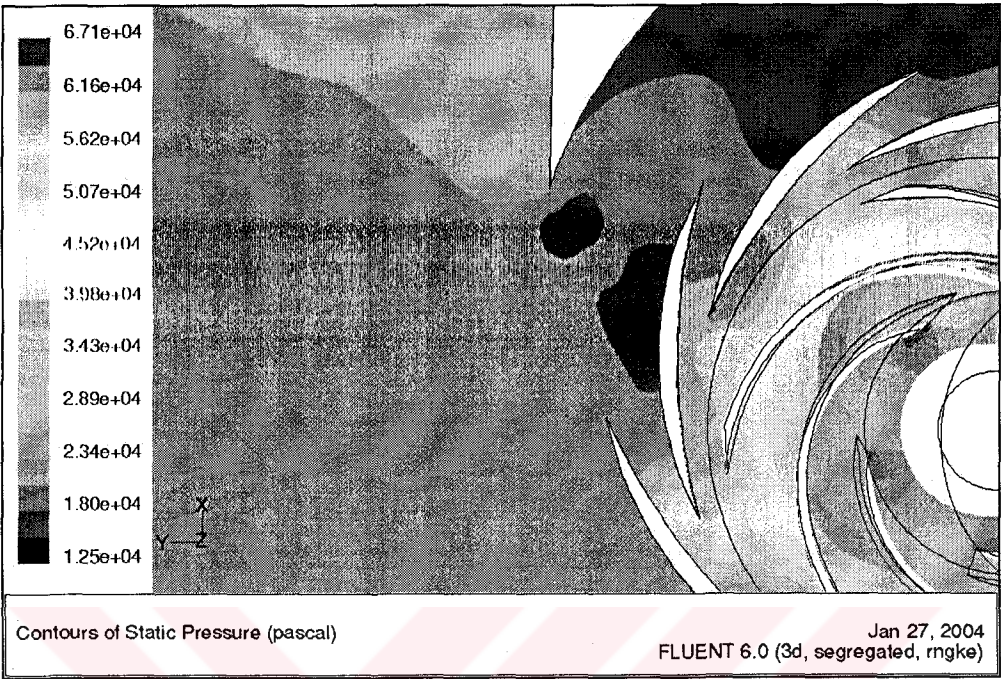


d)

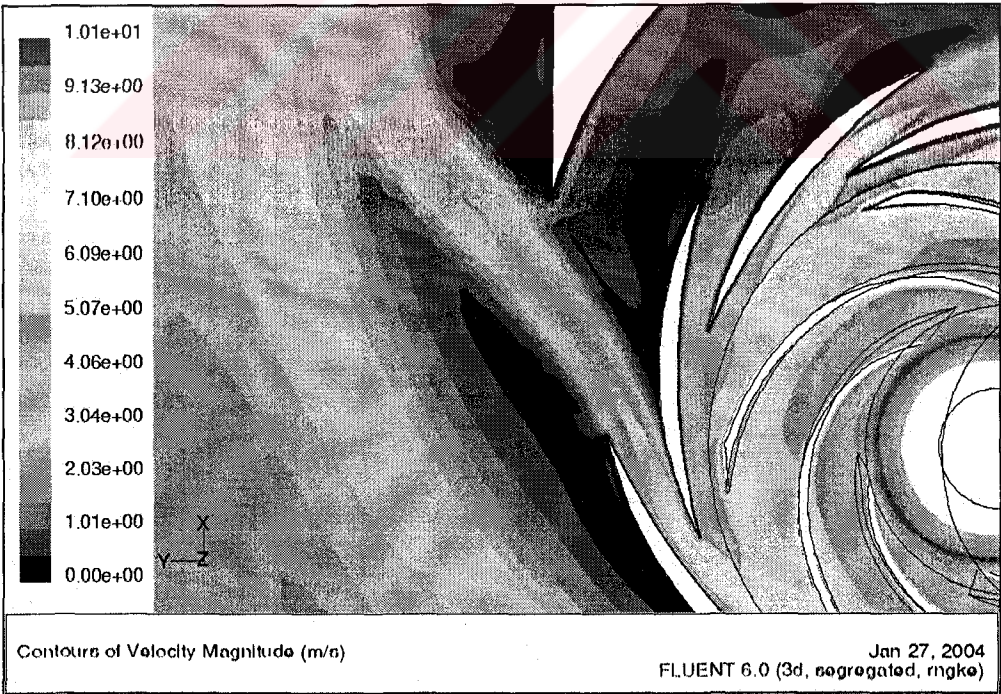


e)

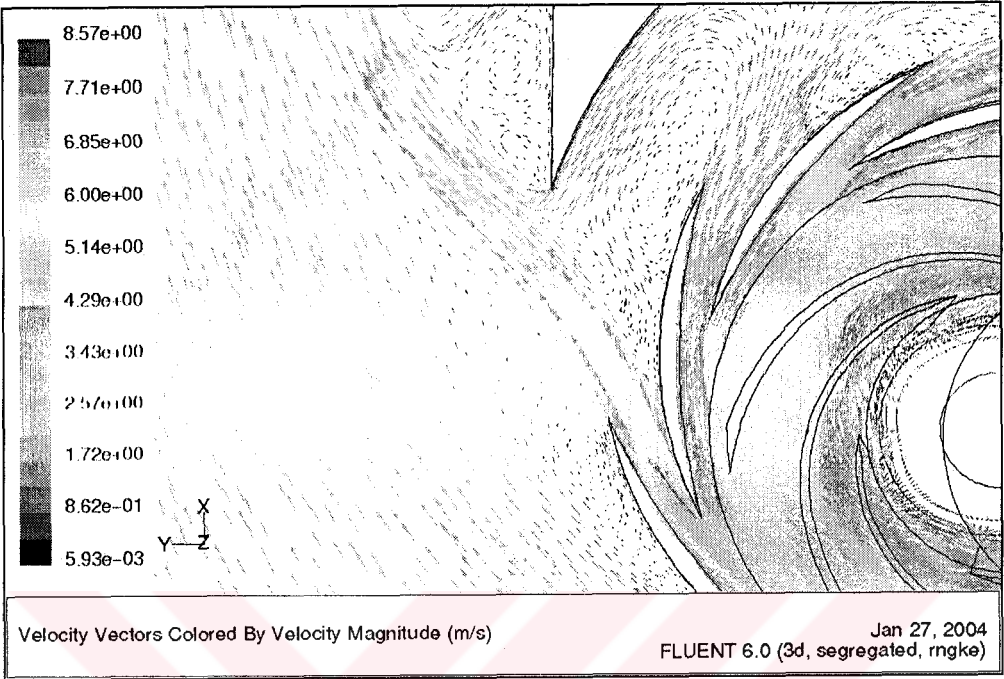
Figure 4.20 a) Contours of static pressure b) Contours of velocity magnitude
c) Velocity vectors d) Contours of vorticity magnitude e) Contours of TKE (for 4.74 l/s) – Case II



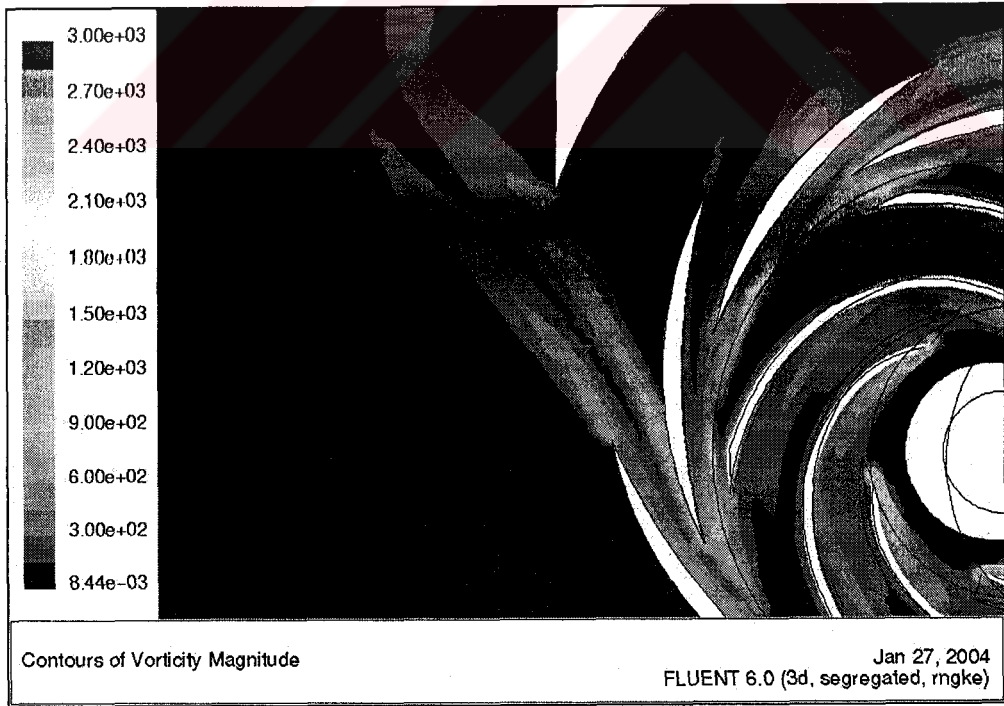
a)



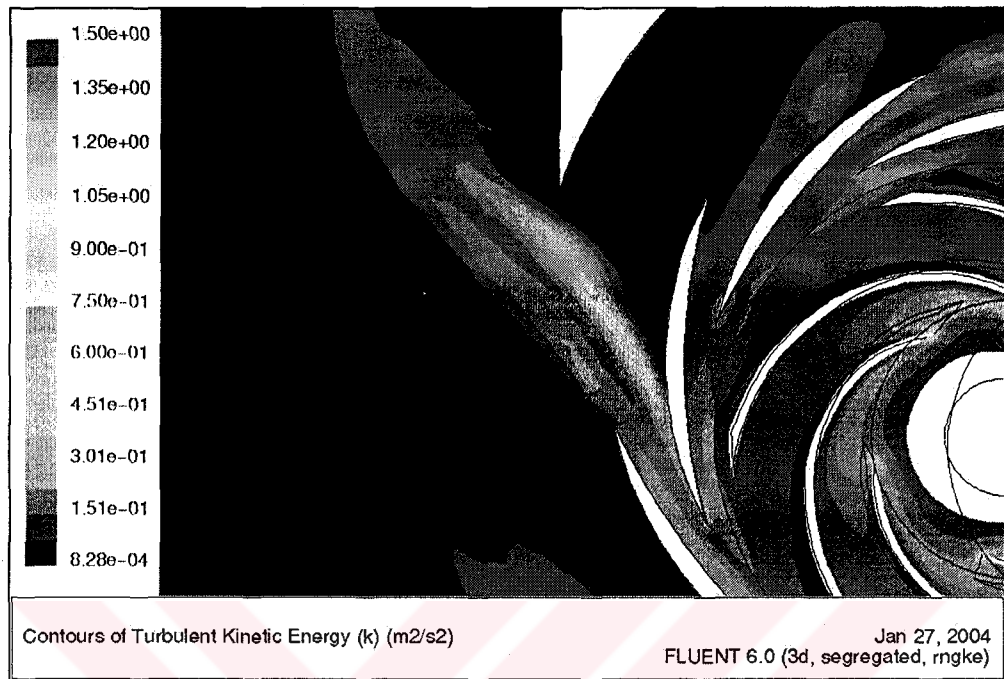
b)



c)

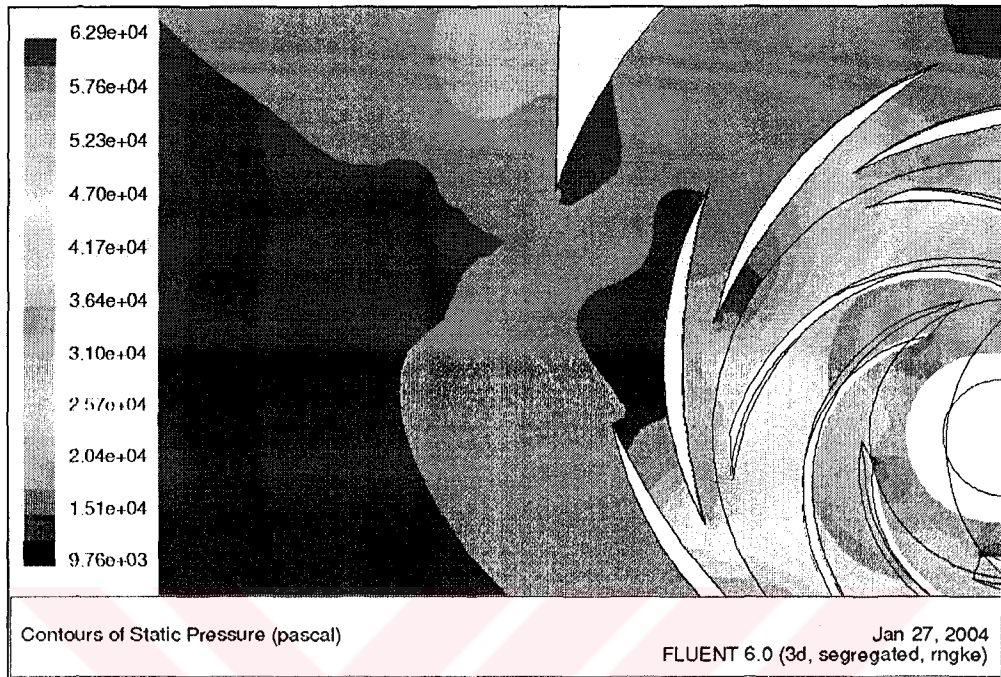


d)

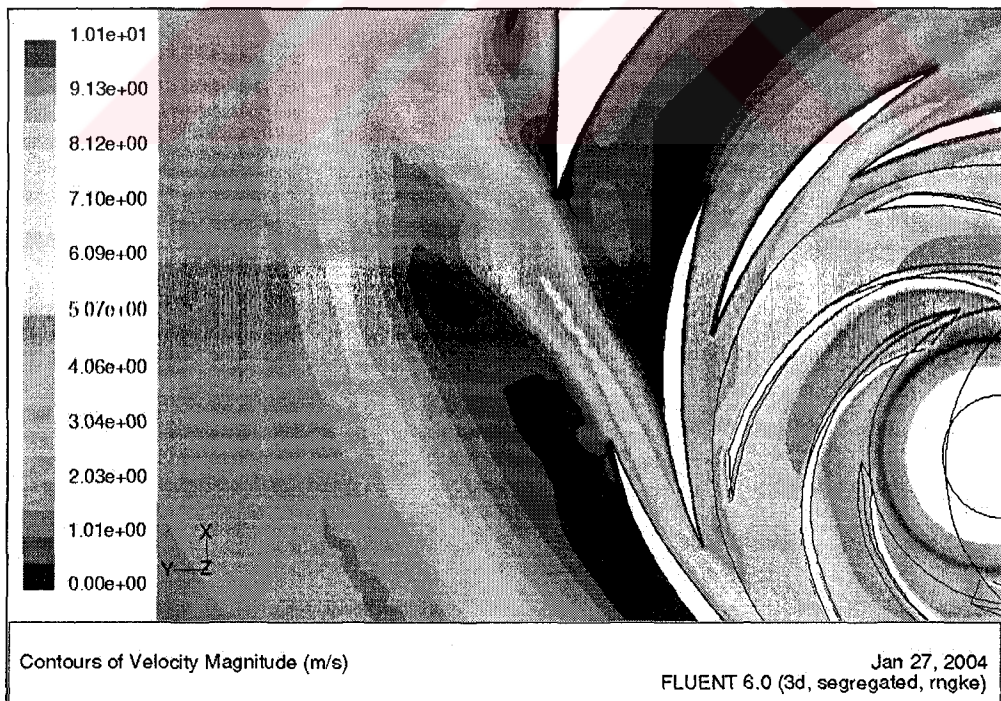


e)

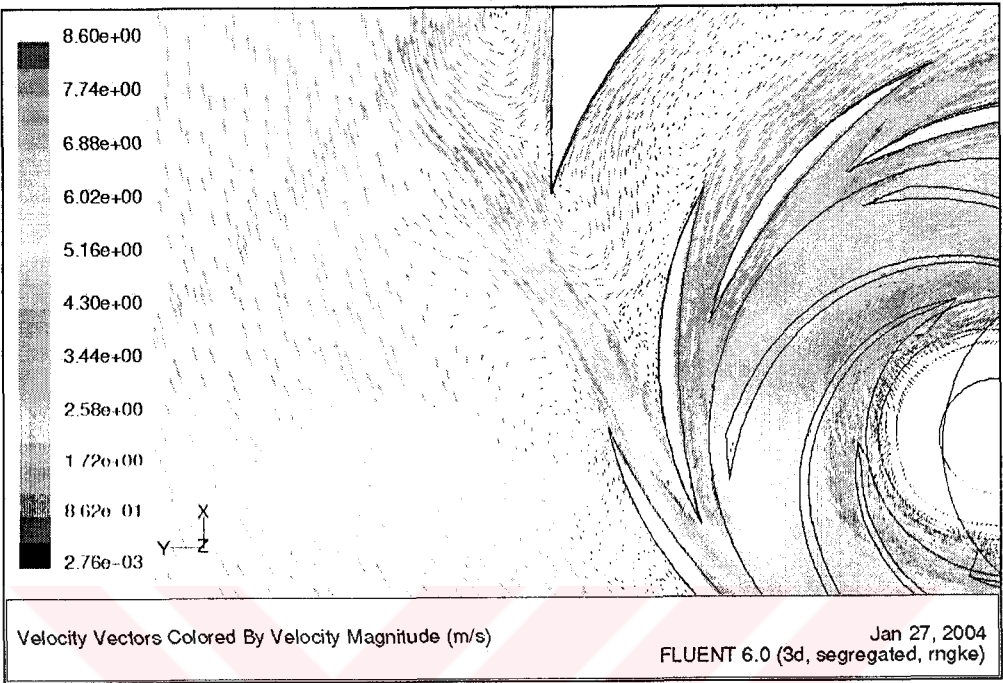
Figure 4.21 a) Contours of static pressure b) Contours of velocity magnitude
c) Velocity vectors d) Contours of vorticity magnitude e) Contours of TKE (for 4.12 l/s) – Case II



a)



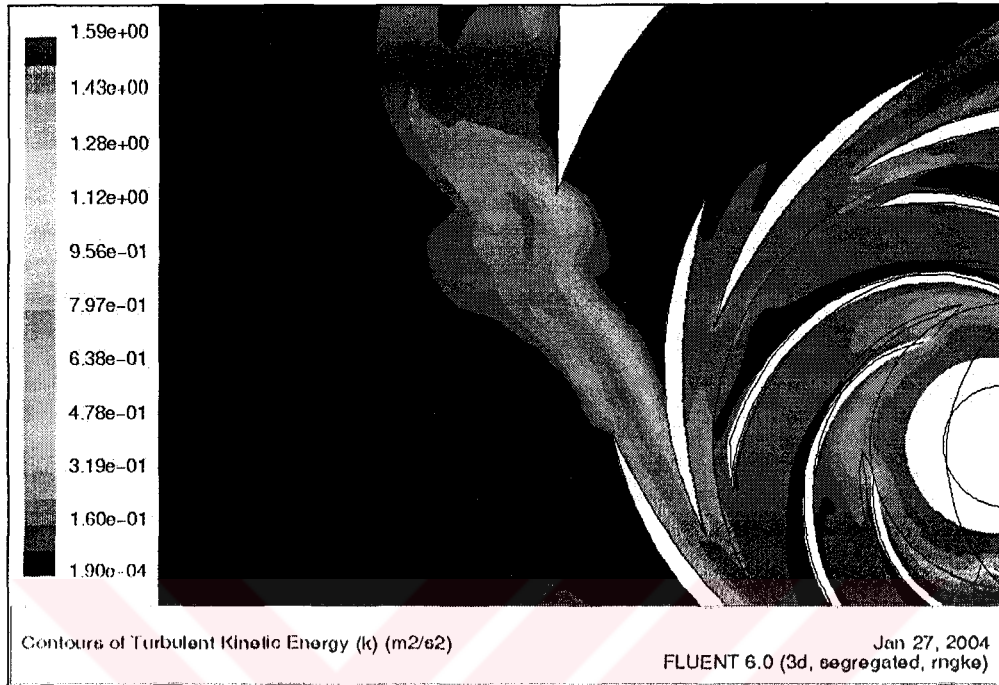
b)



c)

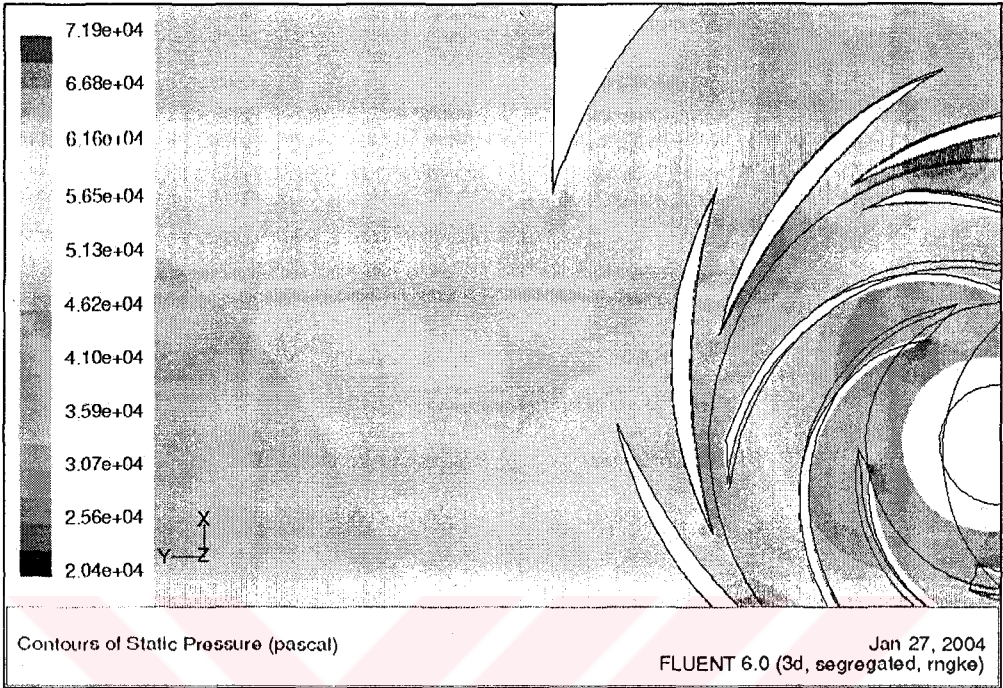


d)

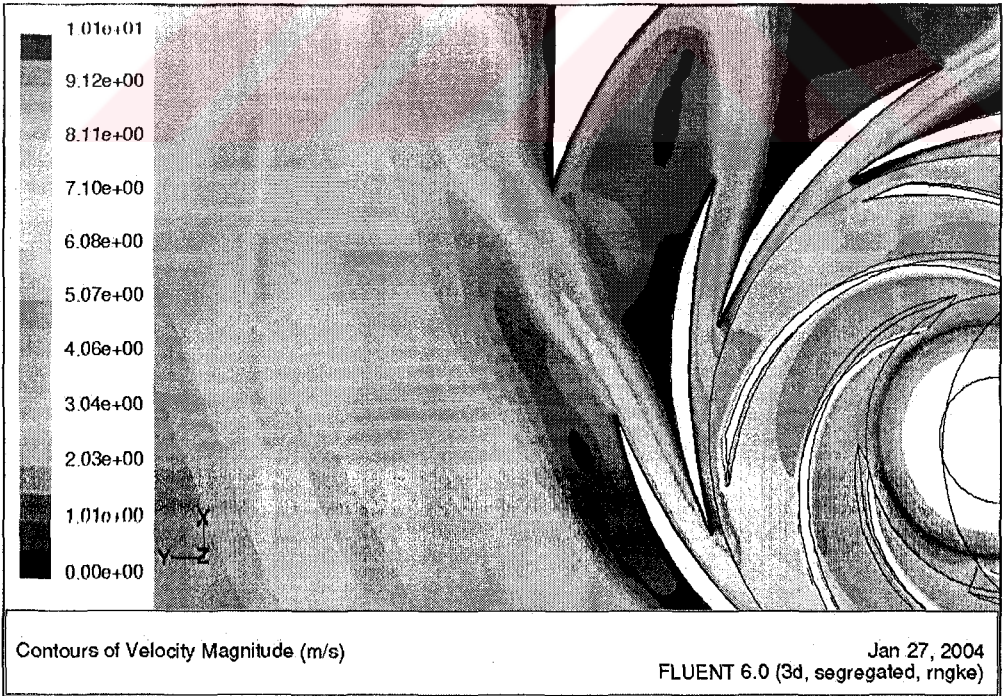


e)

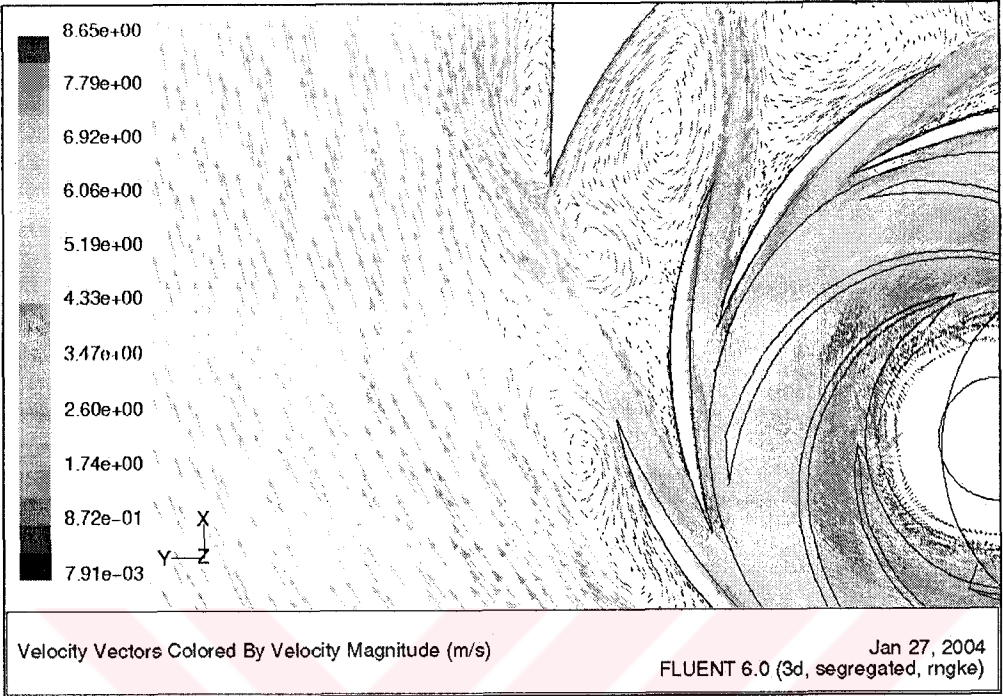
Figure 4.22 a) Contours of static pressure b) Contours of velocity magnitude
c) Velocity vectors d) Contours of vorticity magnitude e) Contours of TKE (for 3.48
l/s) – Case II



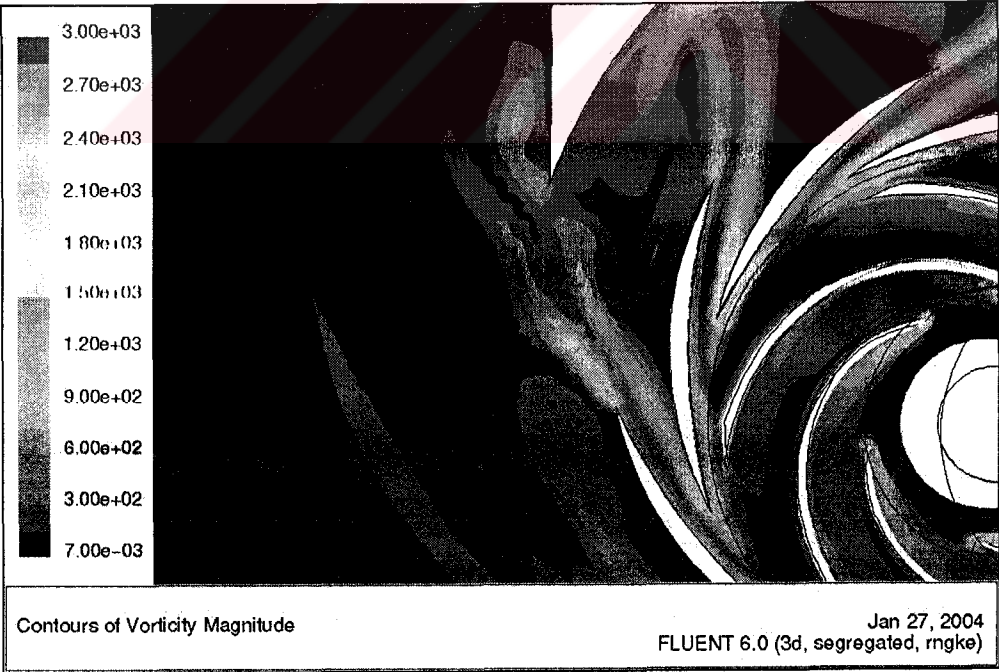
a)



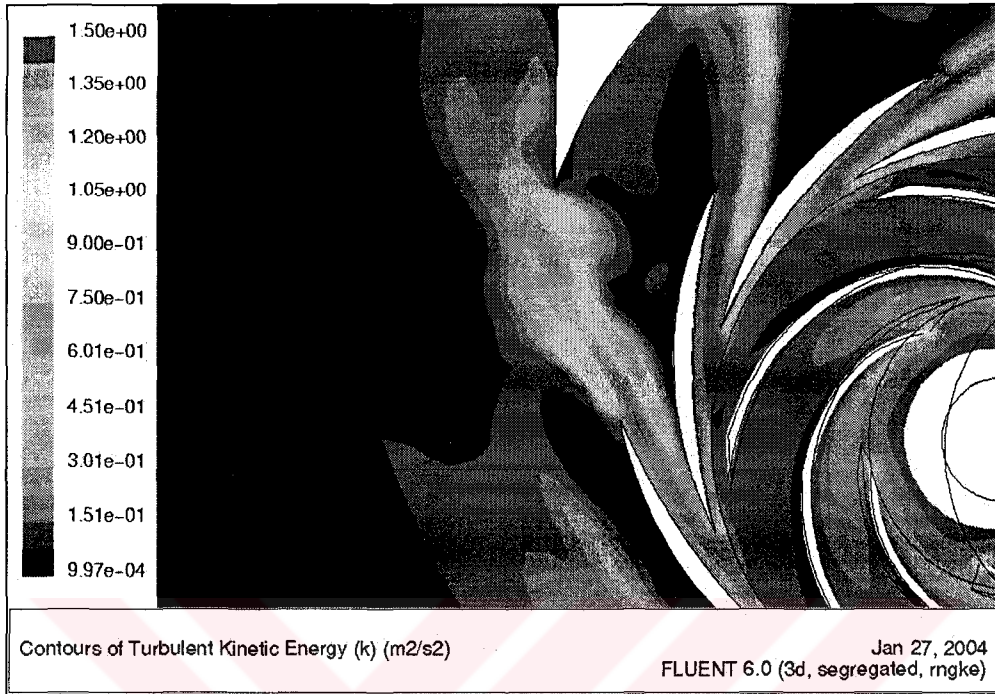
b)



c)

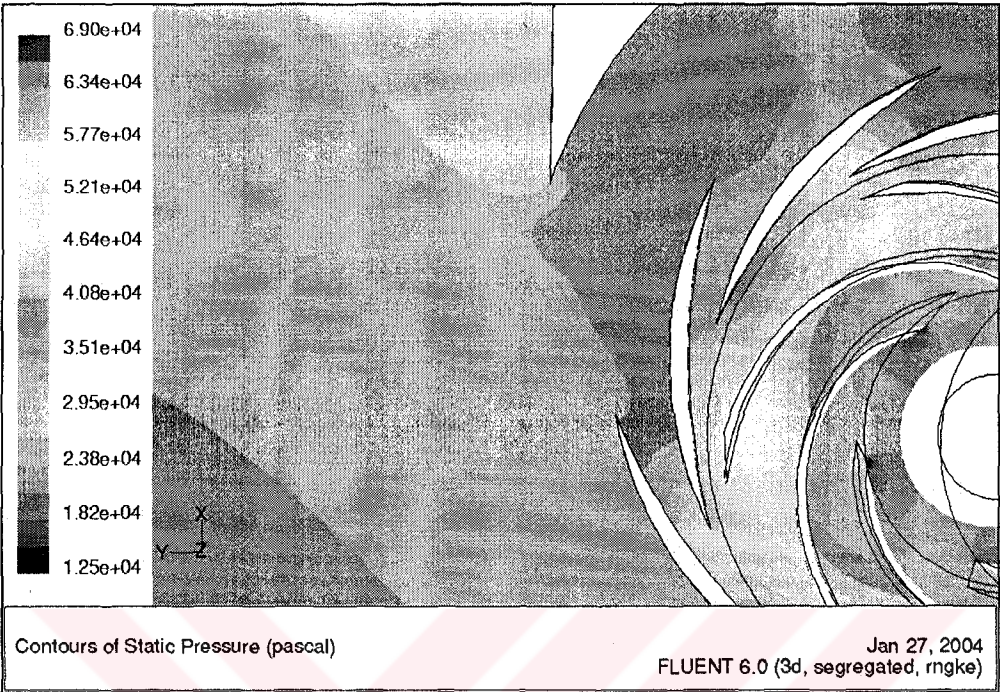


d)

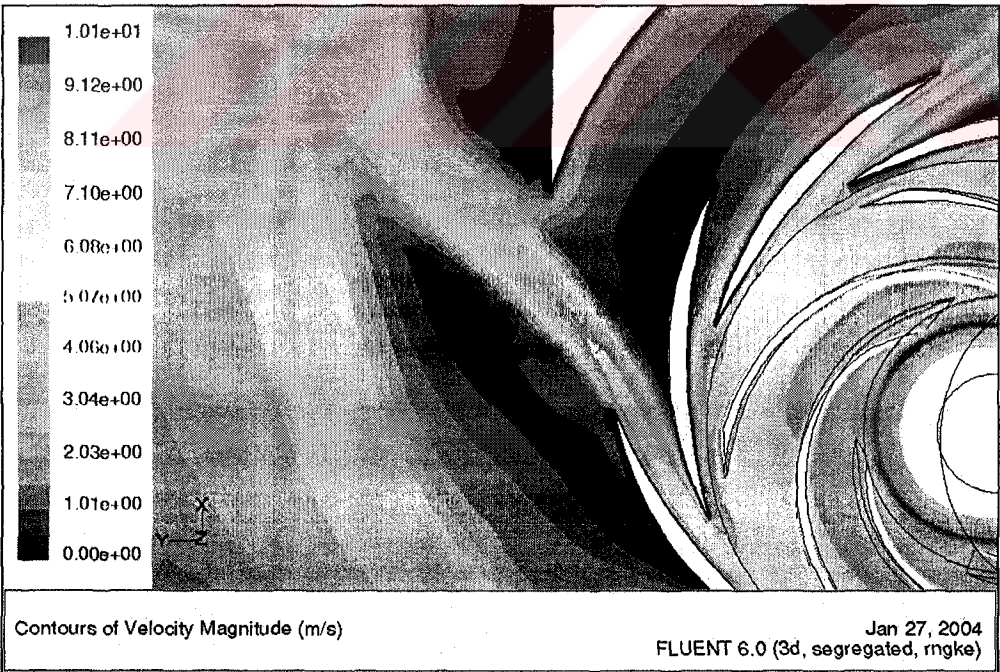


e)

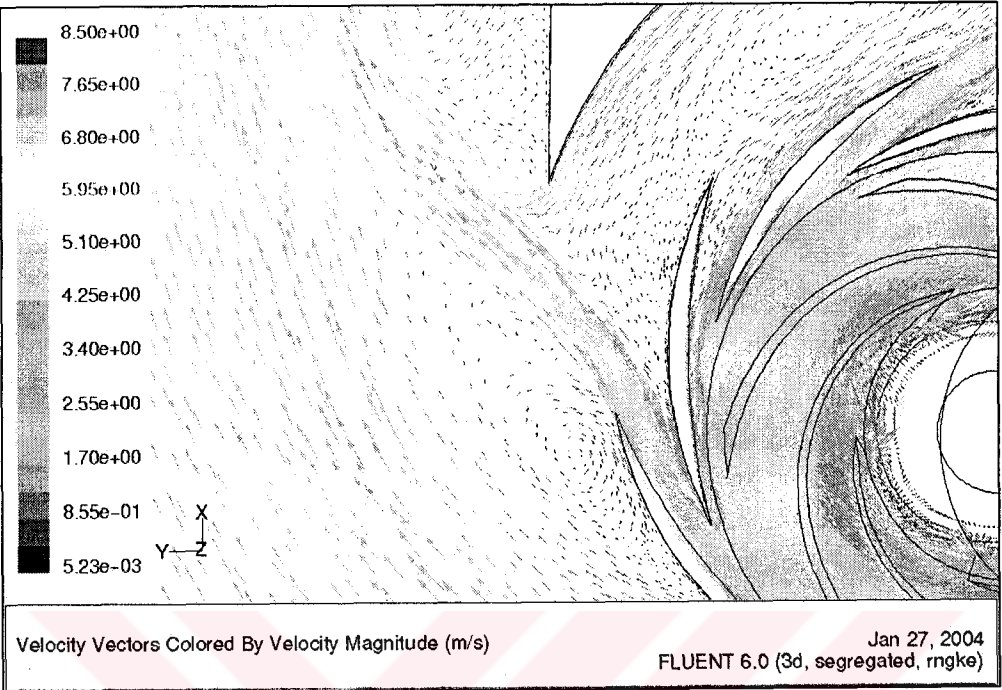
Figure 4.23 a) Contours of static pressure - b) Contours of velocity magnitude
c) Velocity vectors d) Contours of vorticity magnitude e) Contours of TKE (for 4.74
l/s) – Case III



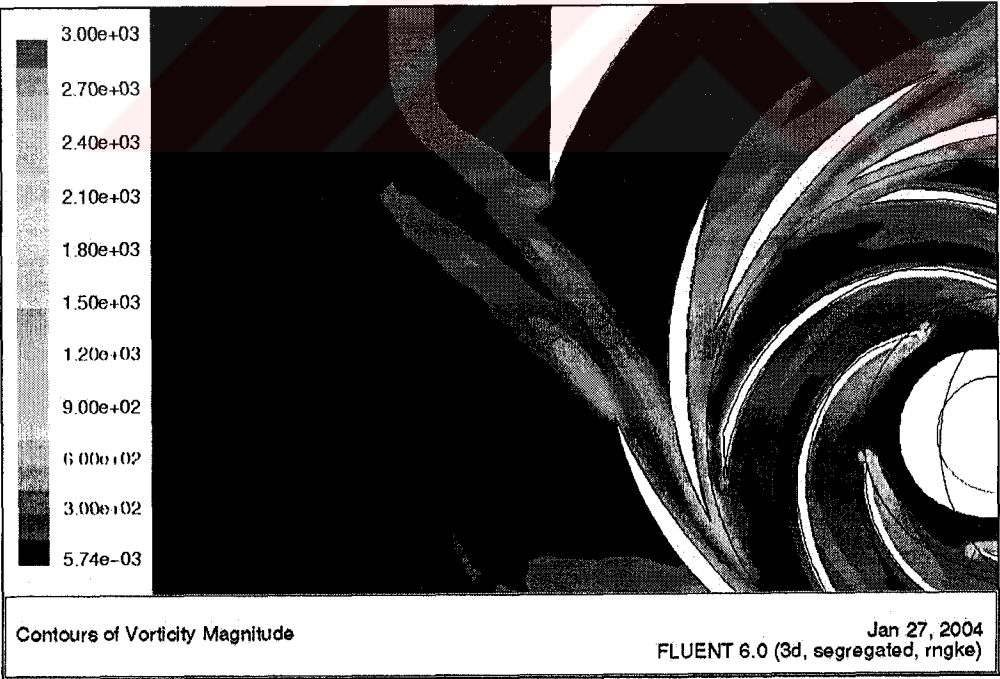
a)



b)



c)

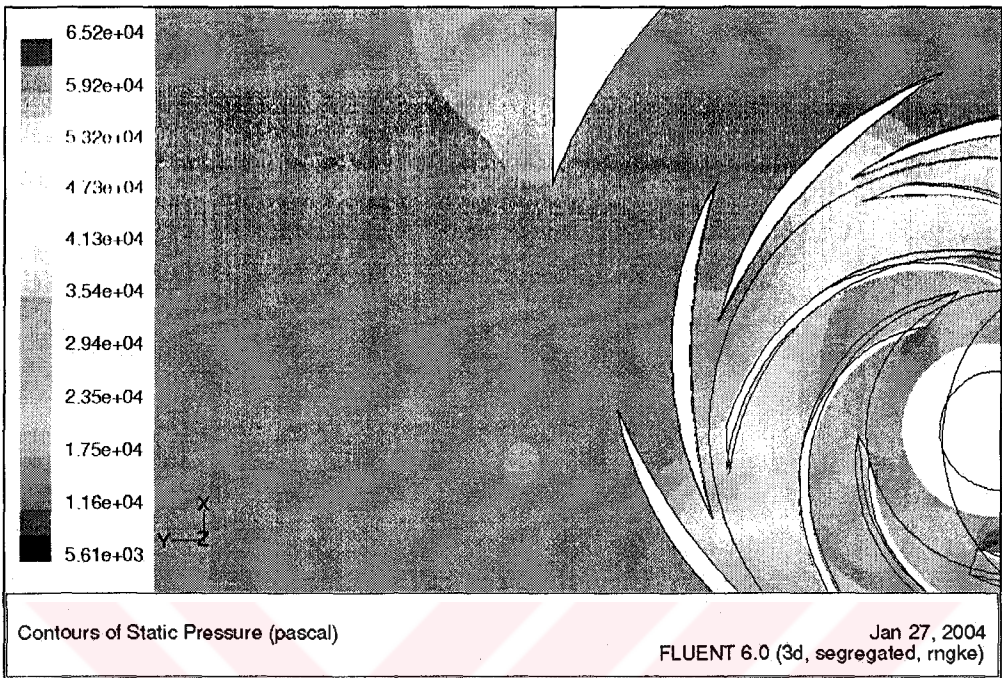


d)

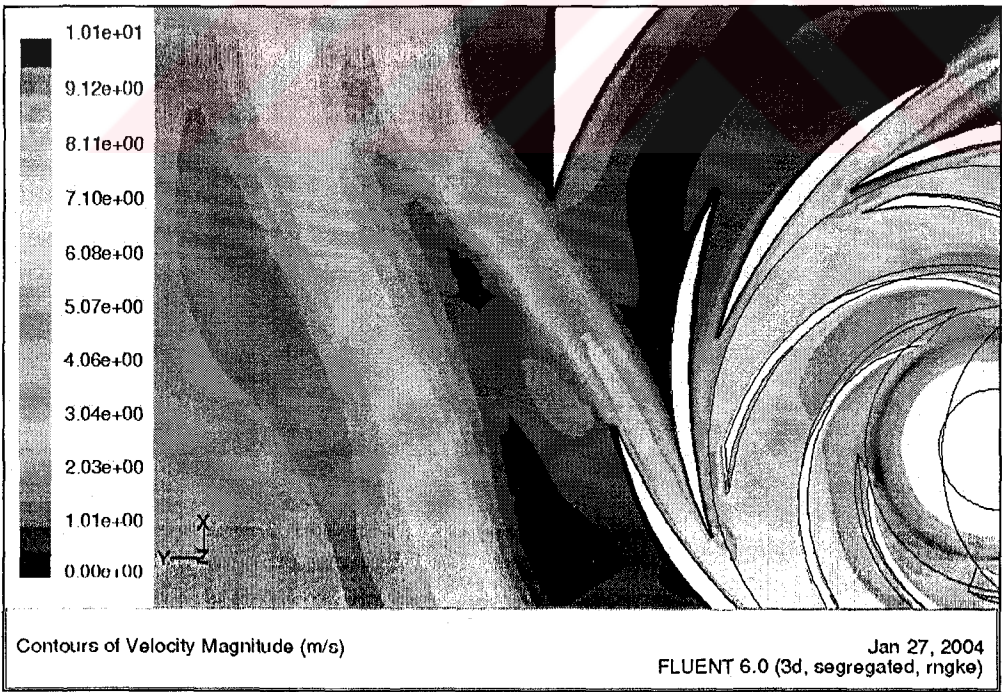


e)

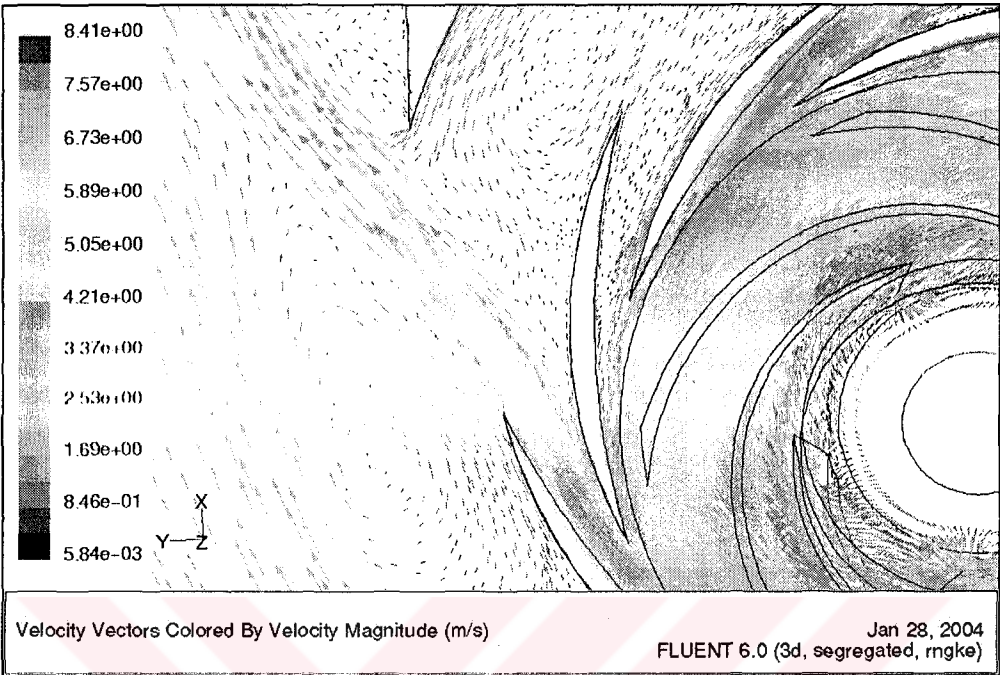
Figure 4.24 a) Contours of static pressure b) Contours of velocity magnitude
c) Velocity vectors d) Contours of vorticity magnitude e) Contours of TKE (for 4.12
l/s) – Case III



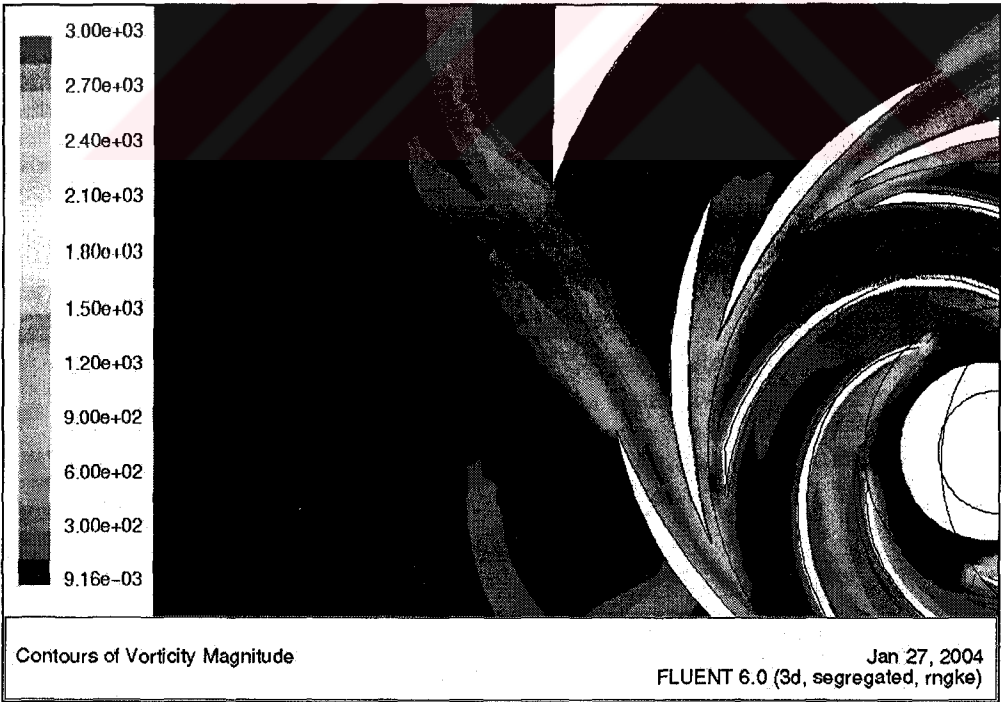
a)



b)



c)

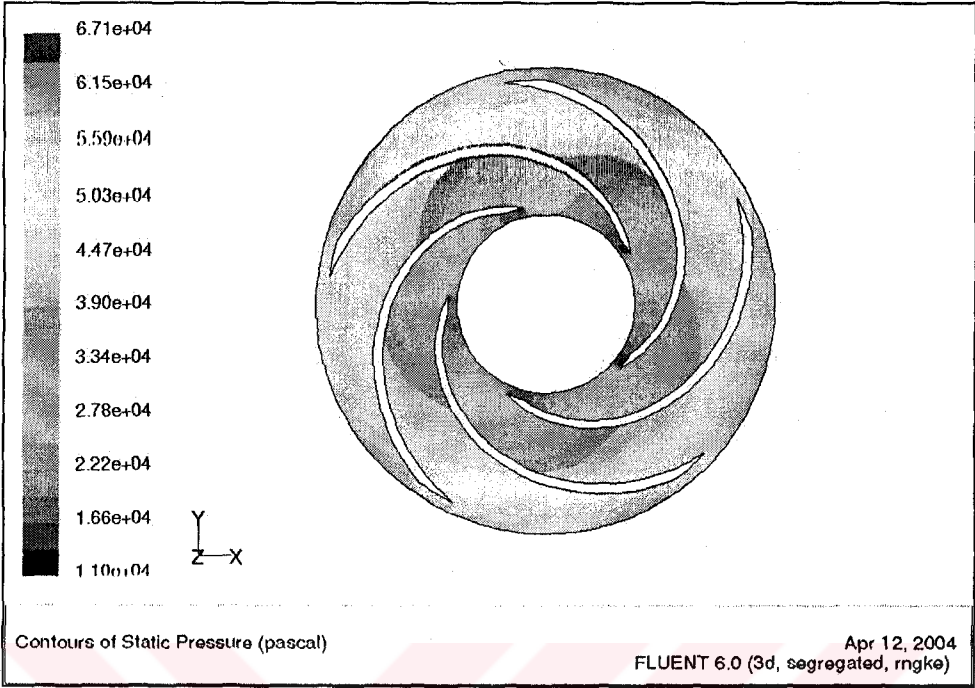


d)

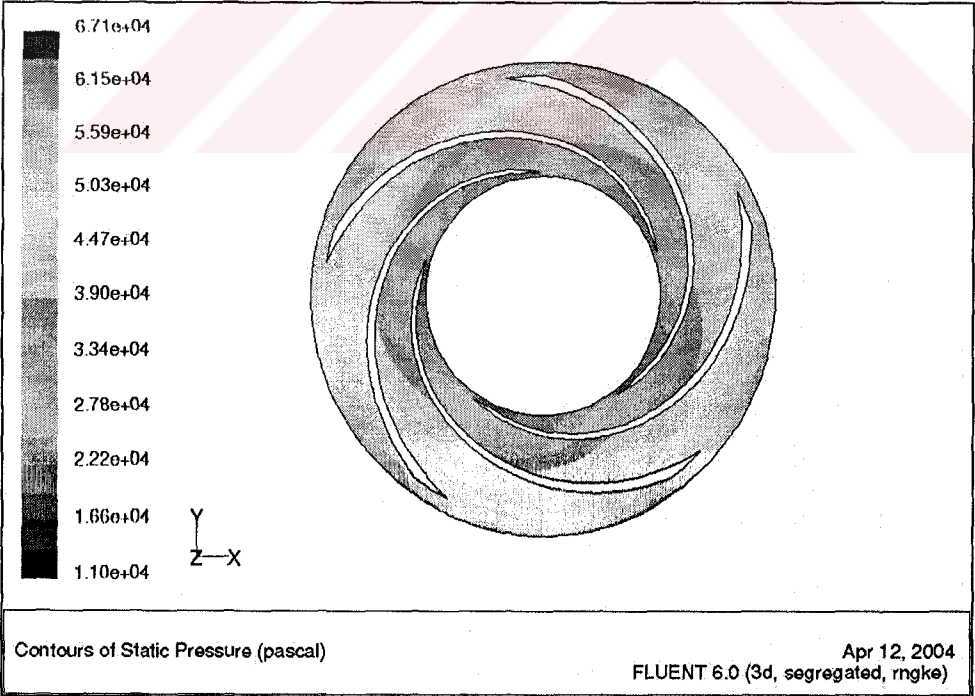


e)

Figure 4.25 a) Contours of static pressure b) Contours of velocity magnitude
c) Velocity vectors d) Contours of vorticity magnitude e) Contours of TKE (for 3.48
1/s) – Case III

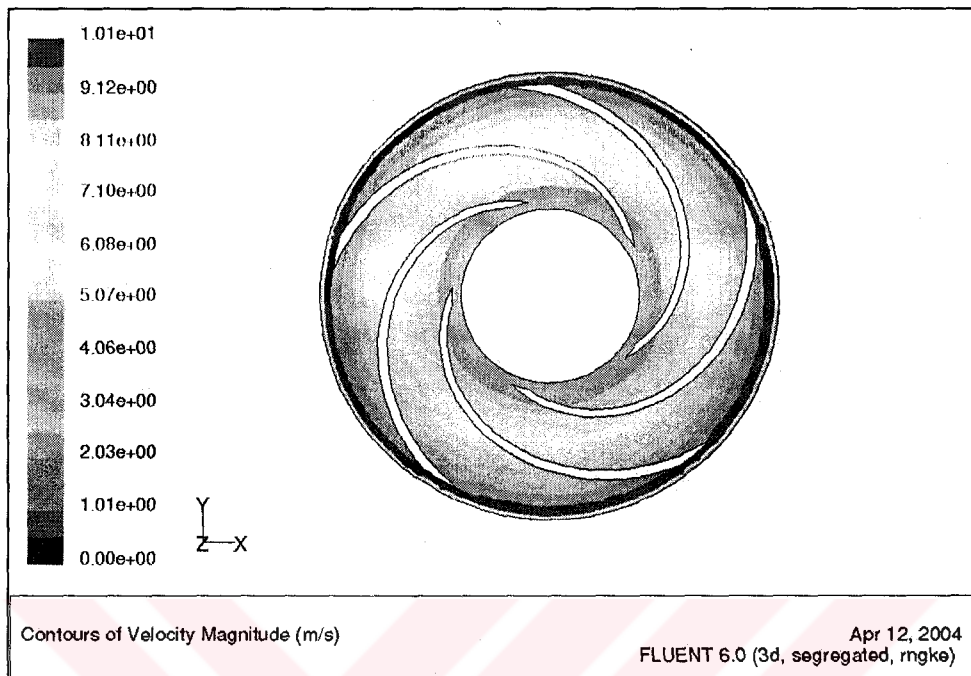


a) Hub

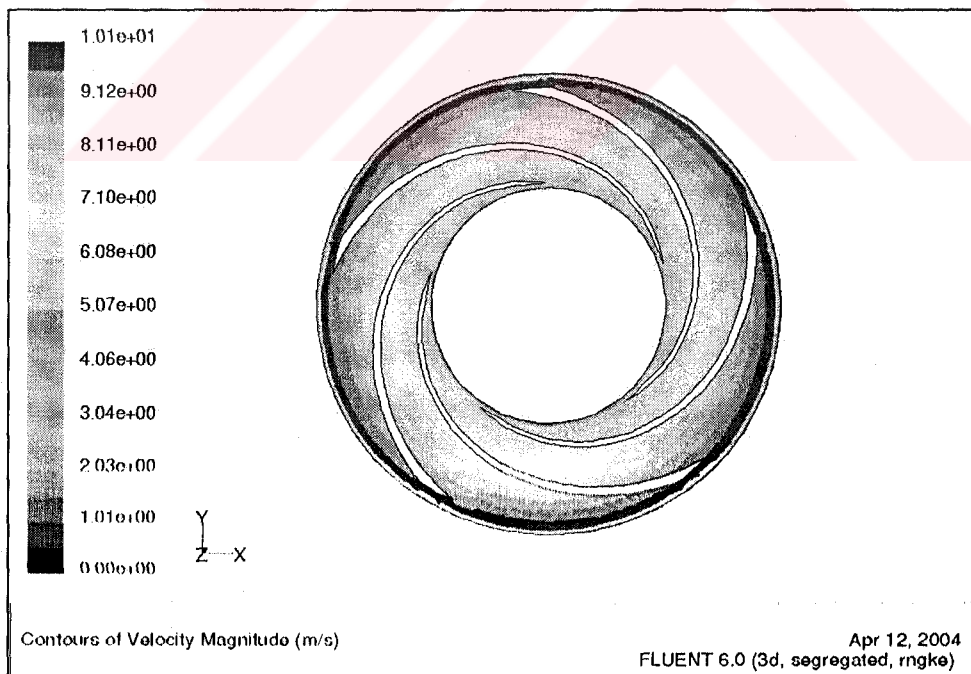


b) Shroud

Figure 4.26 The static pressure contour on hub and shroud for 4.12 l/s

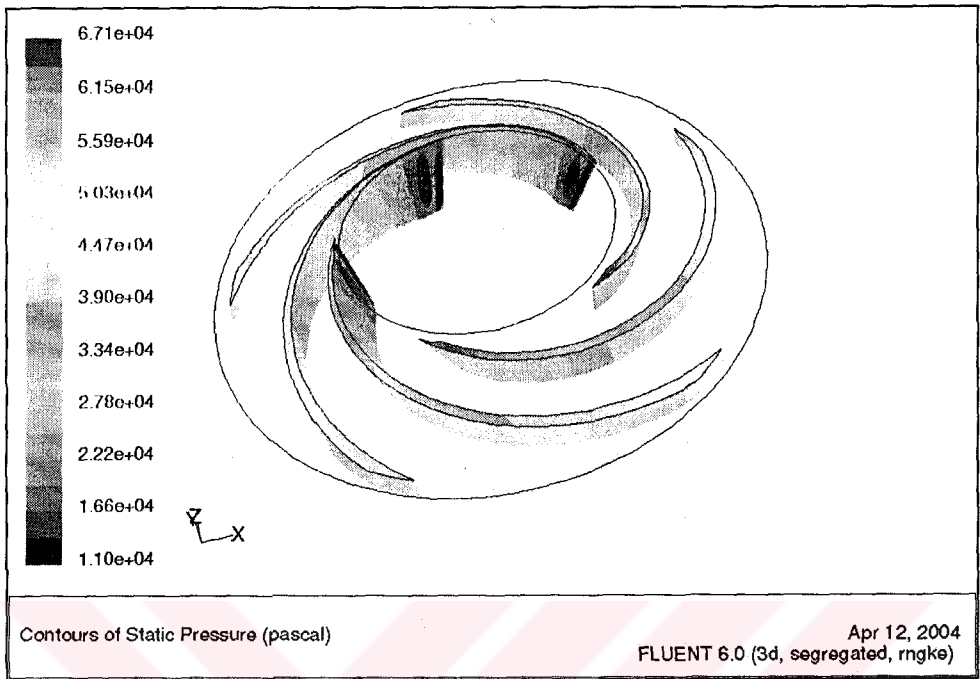


a) Hub

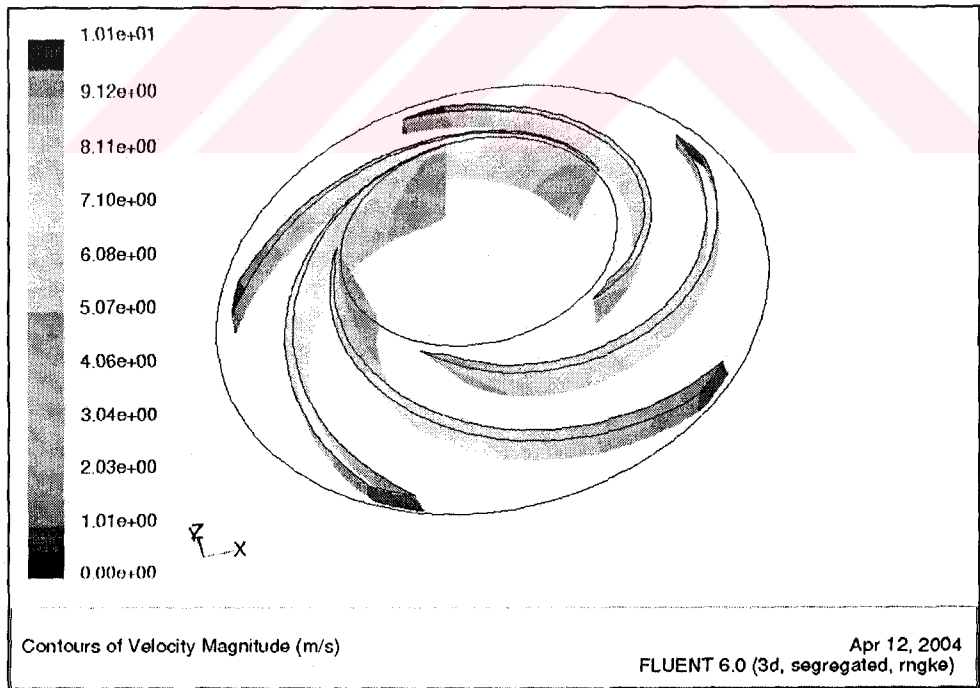


b) Shroud

Figure 4.27 The velocity contour on hub and shroud for 4.12 l/s

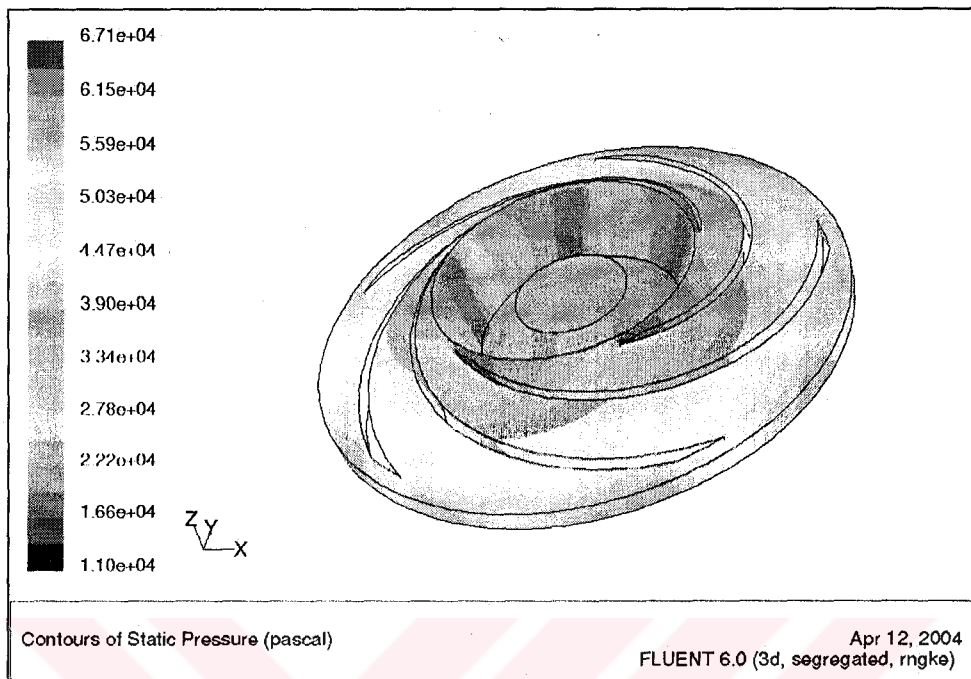


a) Static pressure

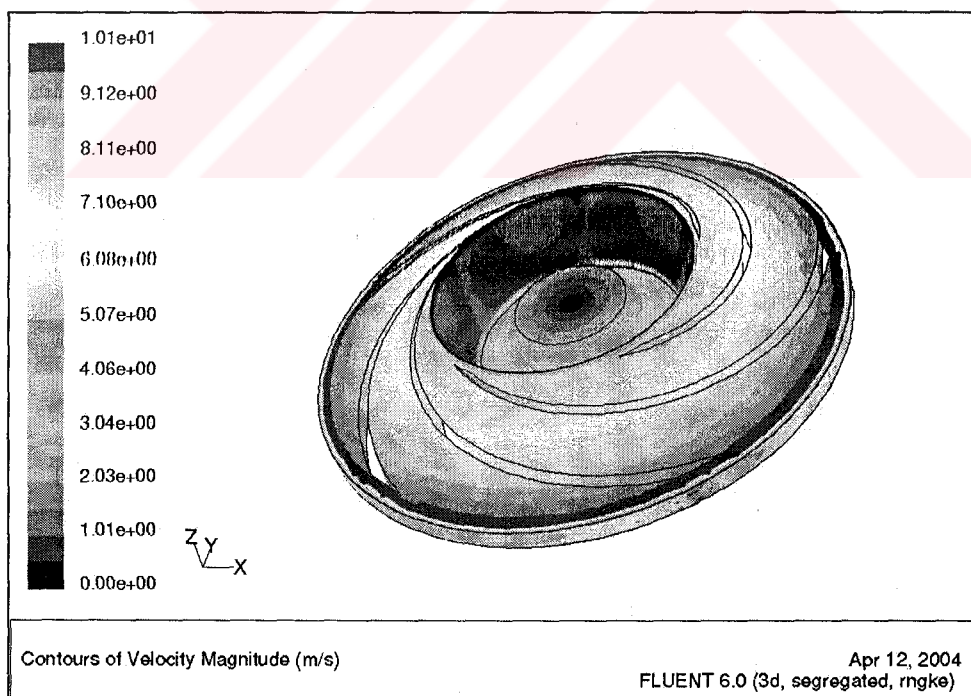


b) Velocity contour

Figure 4.28 The static pressure and velocity contours on impeller blades for 4.12 l/s

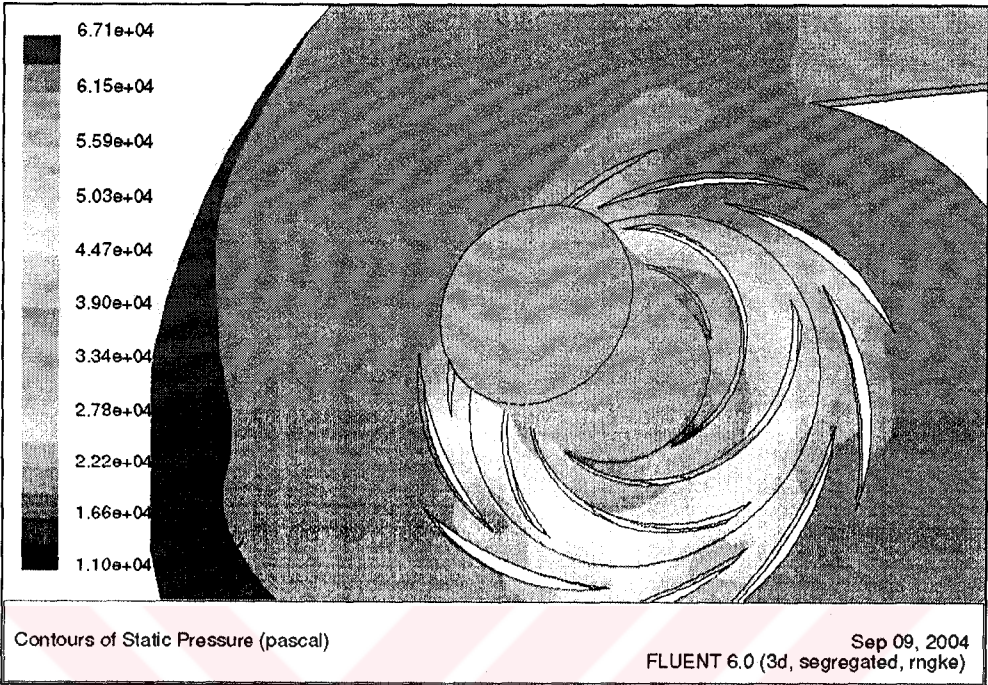


a) Static pressure

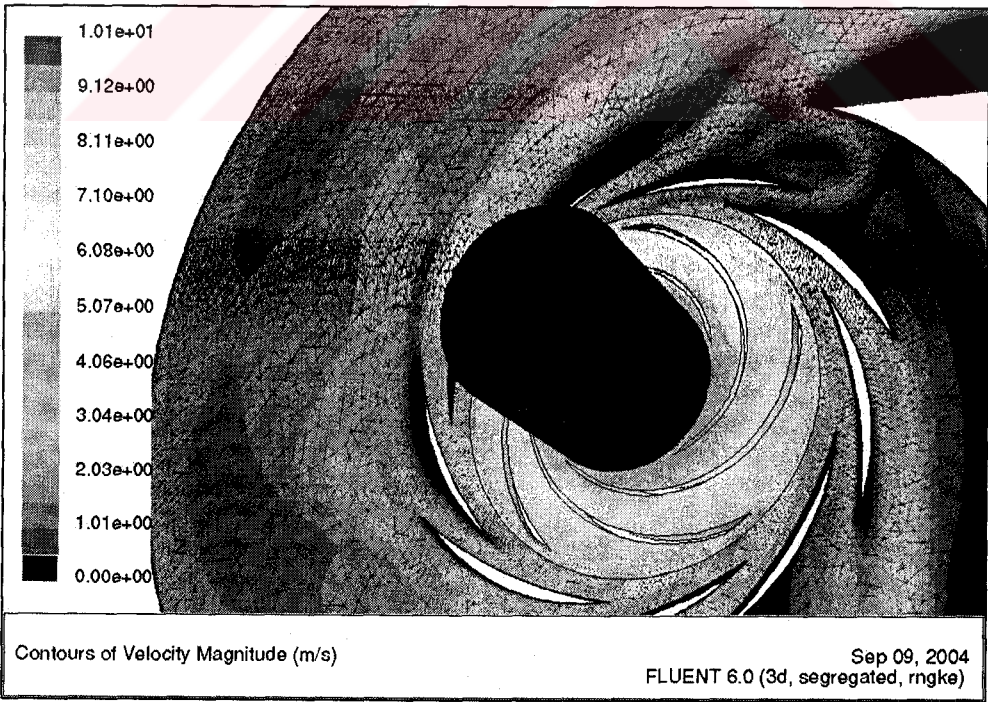


b) Velocity contour

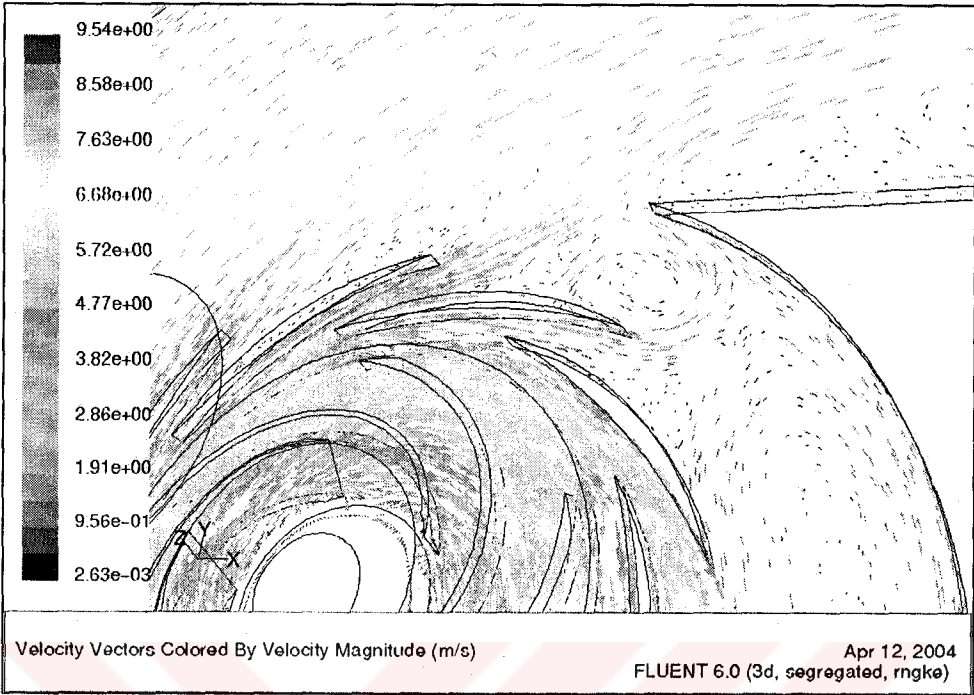
Figure 4.29 The static pressure and velocity contours on impeller for 4.12 l/s



a) Static pressure



b) Velocity contour



c) Velocity vector

Figure 4.30 Three-dimensional view of inside the pump flow for 4.12 l/s

4.5 Pressure Fluctuations and Distribution inside the Pump

Because of non-axisymmetric geometry of volute and the effect of diffuser exit of the impeller, a circumferential pressure variation imposes at the outlet of the impeller. This creates a cyclic variation of the flow in the impeller channels and produces variable blade forces and a radial force on the impeller shaft, which can cause failure of blades. Improvement of the centrifugal pump performance and the control of the circumferential variation of the static pressure caused by the volute require a good understanding of the flow mechanisms defining the three-dimensional structure of the flow in the volute.

Because the radial fluid forces generated from the rotating pressure waves, the whirling direction of the fluid forces will be the same as the rotating direction of the pressure waves. When the rotating direction of the pressure waves is the same as that of the impeller, the fluid forces will be a forward-whirling excitation. This provides a physical explanation for the fact that the fluid forces are a backward-whirling excitation.

The vane pressure fluctuations for comparable radial gaps between impeller blades and diffuser vanes were found to be of similar magnitude.

It can be seen that the magnitude of the fluctuations varies significantly during one impeller revolution. This is due to the varying distance between the impeller blade trailing edge and the diffuser vane leading edges as impeller completes one revolution. The trailing edge pressure fluctuations are largest during the vane passage following the smallest gap between the specific instrumented impeller blade and the diffuser vanes. The minimum pressure at the trailing edge pressure top within each diffuser vane passage is attained after the impeller blade trailing edge has passed the diffuser vane leading edge. The trailing edge then rises sharply to its maximum value, which is attained before the impeller blade trailing edge reaches the next vane leading edge.

The wake causes pressure pulsations, which in turn give rise to vibration and noise. The pressure pulsations are strongly affected by the radial gap between impeller and diffuser.

The largest pressure fluctuation occurs right before the pressure side of the impeller blade trailing edge reaches the diffuser vane leading edge. The pressure decreases sharply and attains its minimum as the suction side of the blade trailing edge reaches the vane leading edge. Then, the pressure rises sharply as the impeller blade moves away from the vane leading edge. Hence, most of pressure fluctuations occur during a rather short time, corresponding to about one third of the impeller blade passage period. The magnitude of the pressure fluctuations is defined as the difference between the maximum and minimum pressure value in the averaging period, which corresponds to one impeller revolution.

It is known that the pressure fluctuations and distribution within the pump are important. Therefore, the pressure fluctuations were investigated by calculating at impeller outlet, between the diffuser vanes, and after diffuser. The points of the calculation pressure distribution are shown in figure 4.1. This figure shows the depended of the radial gap between impeller blades and diffuser vanes, $R_3/R_2 = 1.20, 1.15$ and 1.10 for fifteen flow rates. The impeller-diffuser interaction and the interaction effect of different gaps are investigated by comparison of pressure fluctuations. The results are obtained at mid-vane height (4 mm) like the experimental study to minimize the effect on the flow of hub, shroud and volute; this distance is half of volute thickness. When it is investigated the similar studies, it is seen that this methods is applied.

Similar calculations were made after impeller, within diffuser blades, and after diffuser. The radial gap between the impeller tip and inlet edge of the stator vanes is 20% of the impeller radius (Case I). For all configurations of the calculations, the impeller geometry is the same.

For Case I, peripheral pressure fluctuations are calculated at the impeller and diffuser exit sections where the radius $R_1=107.77$ mm (about 107% of the impeller exit radius) and $R_2=161.25$ mm (about 107% of the diffuser exit radius). For Case II and Case III, R_2 is calculated as 158.024 and 154.463 mm, respectively. The angular origin is at the edge of x-axis and it increases with counterclockwise

orientation and tongue is at 61° (see figure 4.1). Data was recorded on the mid-plane (in 4 mm) of the pump corresponding to flow rate level. The pressure results of fluctuation at the hub and shroud side for the impeller and diffuser exits are observed to show no significant difference due to the small axial width (1.27 cm).

The thirty-six pressures were calculated points are assumed on the central plane at 4 mm. Because the pump has small volute thickness, the pressure fluctuations from hub to shroud were ignored. For each of the thirty-six measurement positions the pressure fluctuations was obtained as a function of the flow rate. The pressure measuring points located of peripheral which is bigger 1.07 times than the impeller radius. This covers a full rotation of the impeller. Figures 4.31, 32 and 33 show the peripheral distribution of the static pressure, for a given volume flow rates. Here the angular position φ is zero at the x-axis of x-y Cartesian coordinate system and increases with anti-clockwise orientation. The static pressure along the volute in close region of diffuser vanes is quite uniform for flow rates in the range of the BEP, but it exhibits either a maximum for low flow rates or a minimum for high flow rates in the other region. These non-uniform pressure distributions are related to the acceleration (high flow rates) or deceleration (low flow rates) of the flow along the volute off-design conditions. The non-uniformity of the pressure in the volute is seen from the flow in the rotating impeller as an unsteady boundary condition, and thus it affects the resulting fluctuations in the pressure field.

These results are presented for the flow rates about 50%, 70%, 85%, 115% and 130% of the best efficiency flow rate Q_N . The comparisons for three radial gap ratios are shown in figures 4.31, 32 and 33., where six different flow rates are plotted. For the flow rates below the nominal one at all gap ratios pressure fluctuations are quite small and limited to a region near the tongue position.

Figure 4.31 shows that the pressure fluctuations in the tongue region is a minimum for the best efficiency flow rate range, and that increases rapidly for both lower and higher flow rates. This is a foreseeable result since only for the best efficiency flow rate there is a good matching between the flow coming out the

impeller and the flow in the volute, with nearly no re-circulation through the gap between impeller and tongue. In off-design conditions however the absolute velocity at the impeller outlet forms a great incidence angle with respect to the mean flow in the volute (behind the tongue region), which leads to intensify flow disturbances accompanying each blade passage.

These results reveal the information about the flow mechanisms in the pump and agreement with the available experimental data in spite of deviation in the magnitude of pressure fluctuations. The magnitude of the pressure fluctuations is defined as the difference between the maximum and minimum pressure values in the averaging period, which corresponds to one impeller revolution. The largest fluctuations, independently of flow coefficient or radial gap, occur at the vane leading edge and on the front half of the vane suction side. Those fluctuations are of the same order of magnitude as the total pressure rise across the pump. The fluctuations are larger on the vane suction than on the vane pressure side. On the vane pressure side, they decrease monotonically from the leading edge to the trailing edge.

Fluctuations are small near the best efficiency point flow rate, and the pressure pulses are upward (pressure increasing) at high flow rates, while downward at partial flow rates. This may be consequence of the flow patterns around the leading edges of the diffuser vanes. Furthermore, the magnitudes of the pressure pulses are not uniform circumferentially during revolution of the impeller. This unevenness depends on flow rates.

At a high flow rate, the pulses are large when the interaction occurs in the front of the volute exit, the opposite situation can be observed at partial flow rates. The pressure pulses are large when the interaction occurs in areas where the static pressure is low. This may be attributed to the high flow velocity in areas where the static pressure is low. The forces are shown to be relatively small near the BEP flow rate, while large at high or partial flow rates.

As compared with the CFD results to the experimental results, CFD approach is capable of predicting global trends in the pressure fluctuations, which are found to be larger on the suction side of diffuser vane than on the pressure side. The

absolute magnitude of the pressure values is lower than high volume flow rate the experimental value at low volume flow rates. At the current stage, the general features of the pressure variations can be captured with accordance to varying radial gap ratios.

This is similar to the diffuser vane pressure measurements presented previously. This component results from the varying radial gap between the impeller blades and the diffuser vanes as the impeller completes one shaft revolution. Figures 4.31, 32 and 33 give some information about the influence of the radial gap on the performance of the entire pump. It can be seen that the pressure is rising continuously with a decreased radial gap. At the three different radial gap ratios, the overall level of pressure fluctuations observed. The results of numerical analysis show that by decreasing radial gap between impeller and diffuser, pressure fluctuations reach to maximum value. But the pressure fluctuations are minimal in the case of maximum radial gap.

The measurements were carried out at 9 positions between two diffuser vanes on the traversing plane. Comparison of the pressure fluctuations in the two diffusers and at five different flow rates is presented in figure 4.32.

It can be seen that (especially for the flow rates $Q=1.96$ and 5.36 l/s) the largest fluctuations occur at 180° . Similar observations were made by Arndt et al. (1989, 1990) based on unsteady wall pressure measurements on the stator and impeller vanes of a centrifugal pump. The fluctuations are the smallest for design flow rate. Compared with the Case I, the maximum pressure fluctuations in the Case II and Case III are significantly smaller. This shows that decreasing of the gap increases the unsteadiness of the flow. The highest fluctuation occurs at high volume flow rates.

The large pressure fluctuations on the front of half of the suction side, at the leading edge, and on the pressure side closest to the leading edge decrease significantly with increasing radial gap. Increasing the radial gap between impeller blades and diffuser vanes resulted in decreasing the fluctuations.

It can be seen that the magnitude of the fluctuations varies significantly during one impeller revolution. This is due to varying distance between the impeller

blade trailing edge and diffuser vane leading edges as impeller completes one revolution. The trailing edge pressure fluctuations are largest during the vane passage following the smallest gap between the impeller blade and the diffuser vanes. The minimum pressure at the trailing edge within each diffuser vane passage is attained after the impeller blade trailing edge has passed the diffuser vane leading edge. The trailing edge pressure then rises sharply to its maximum value, which is attained before the impeller blade trailing edge reaches the next vane leading edge.

The highest pressure fluctuations occur with the vaned diffuser and at the suction side of the stator vanes. Furthermore, the pressure fluctuation increases with the flow rate and the performance of the pump changed in combination with changing radial gap.

For each flow rate, the behavior observed is very different from that of the stationary pressure distribution. As stated above, in general the pressure fluctuations are comparatively small in design flow rate. As can be seen in figure 4.32, the diffuser produces a strong effect, especially at the region after the diffuser. It can be observed how the effect of diffuser (pressure stronger peak and valley) is in pressure fluctuations for high and low flow rates. For the three radial gap ratios, pressure fluctuations are observed to behave similarly at the same flow rate. The pressure fluctuations are increased at low and high flow rates, but nominal flow rate. For off-design conditions, however, the region of the volute close to the tongue presents a simultaneous evolution of the pressure fluctuations, with big amplitude in the whole region. They are the result of the strong interaction between the impeller and diffuser, which produces pressure oscillations. Along the volute after the diffuser, these oscillations are progressively reduced due to divergence and absorption. Comparing the graphs of figure 4.33 for flow rates above and below Q_N , the fluctuations in the region close to the tongue are seen to be shifted from each other. This is related to the shift of the stagnation point on the tongue and the different characteristics of the leakage flow through the radial gap.

In summary, pressure fluctuations in after the impeller, diffuser and between the impeller and diffuser are effective from the point of view of performance and

efficiency. The effects of the impeller-diffuser interaction are particularly important for off-design conditions.

The fluctuating pressure field extends to dynamics forces which can cause fatigue on the pump shaft (especially at off-design conditions). Once the pressure predictions have been proven to be quite accurate, the numerical model developed has shown its worth and can be used to calculate these forces. Such interaction clearly increases the fluctuation levels for off-design conditions, which produces other effects already studied in the bibliography (limiting operating ranges, increase of losses, cavitation etc.).

The pressure fluctuations registered around the tongue and diffuser were very dependent on the flow rate and the radial gap ratios. In the general amplitude of the fluctuations is greater in off-design conditions and small gap ratios in particular the tongue region. Comparison of the present data shows that reducing the radial gap leads to an increase in the pressure fluctuations. The pressure fluctuations change with the flow rates and radial gap ratios.

Comparisons of pressure fluctuations are presented in figures 4.31, 32 and 33. It is evident that near the tongue and the exit the pressure fluctuations are considerably smaller, whereas somewhat far upstream of the tongue they are slightly higher.

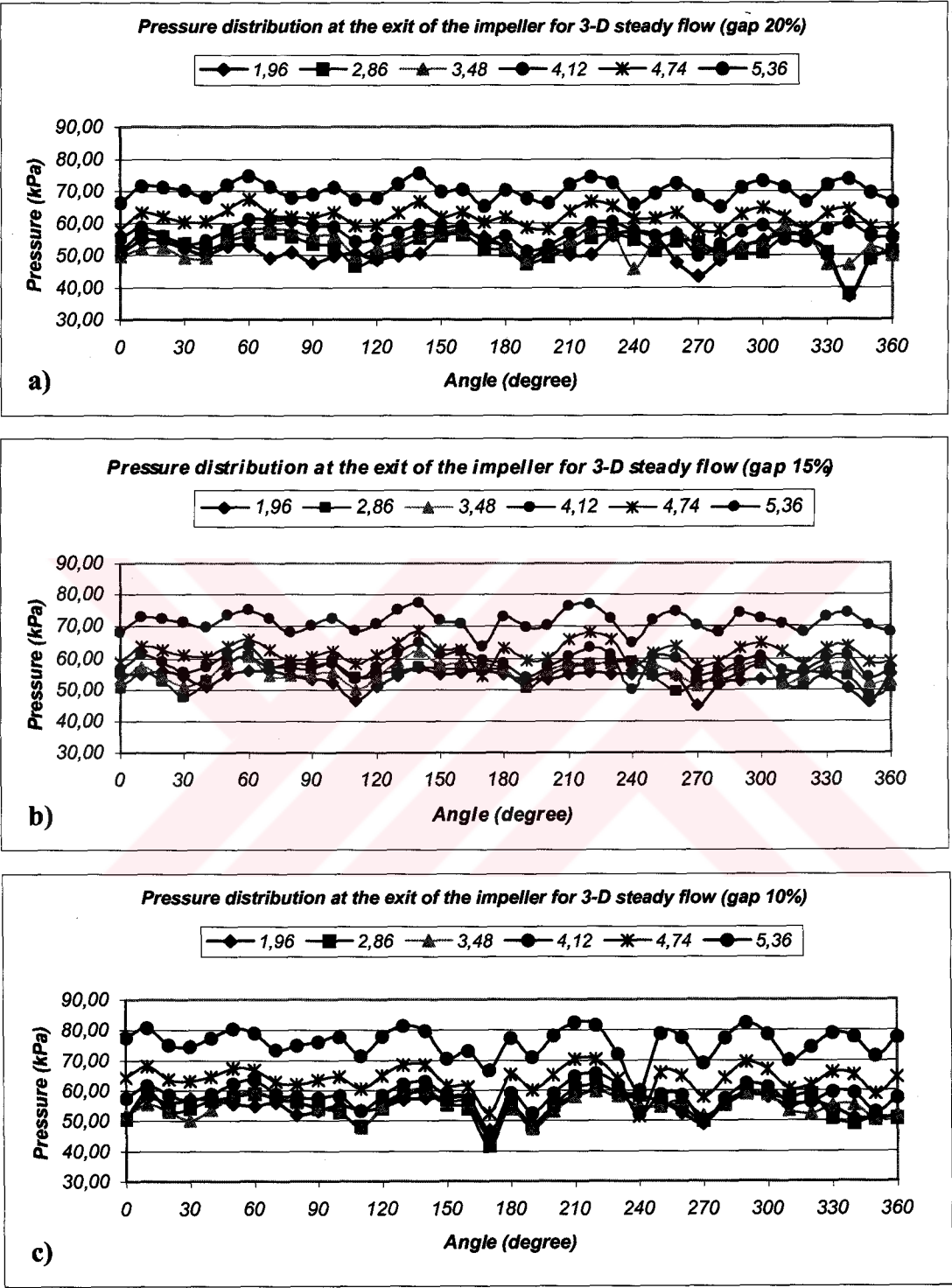


Figure 4.31 Pressure distributions between impeller and diffuser for Case I, II, III at six flow volume flow rates

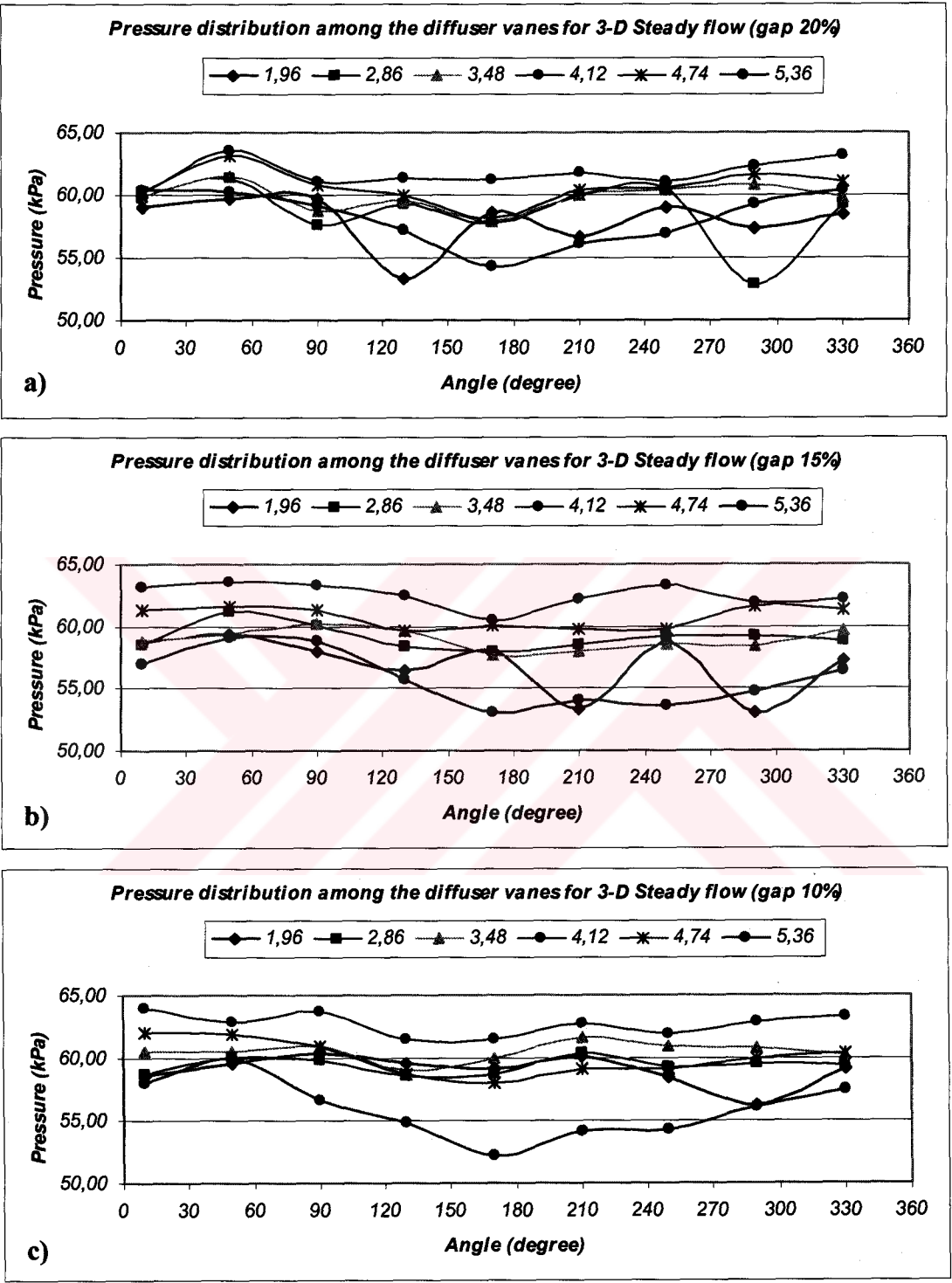


Figure 4.32 Pressure distributions between diffuser vanes for Case I, II, III at six flow volume flow rates

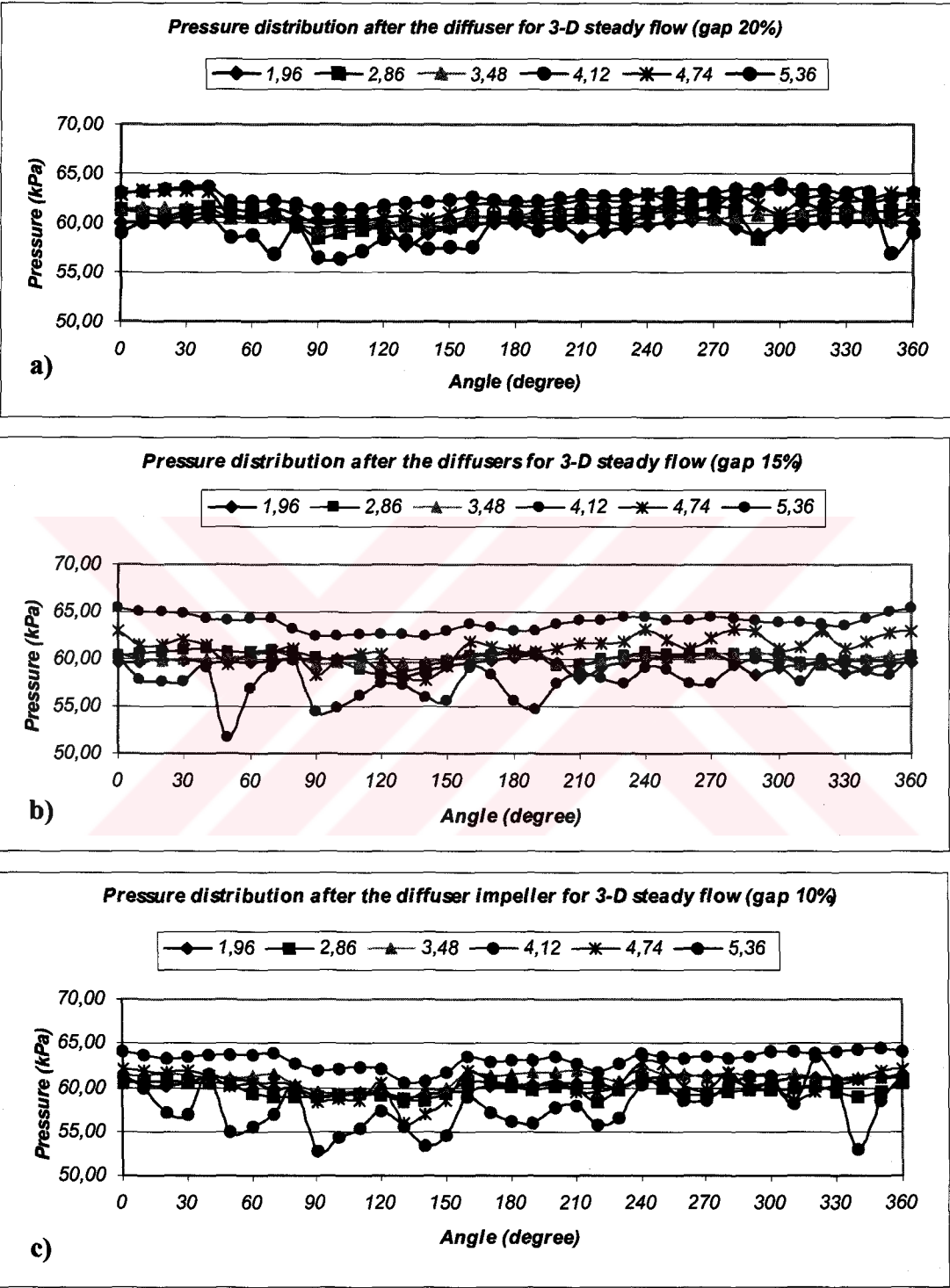


Figure 4.33 Pressure distributions after diffuser for Case I, II, III at six flow volume flow rates

5. CONCLUSIONS

The role of CFD in engineering predictions has become so strong that today it may be viewed as a new third dimension of fluid dynamics, the other two dimensions being the above stated classical cases of pure experiment and pure theory. One of the advantages of the numerical simulation is the ease to carry out changes in the geometry, parametric studies and analysis of anomalous operation conditions.

The ultimate goal of this work was to show the effect of different gaps on the impeller-diffuser interaction, the flow structure in pump, performance and efficiency of pump, by means of CFD. Also, the present work intended to show how the impeller-diffuser interactions are dependent on different gap ratios and its associated flow structures related to performance characteristics like efficiency, stability of a pump, by means of the experimental and numerical results for the flow behavior in the impeller-diffuser interaction zone. Hence, it can be gained a deeper knowledge of the flow variables inside a centrifugal pump (pressure, velocity etc.), which could be used in a design process.

The flow characteristics field inside the pump changes strongly if the centrifugal pumps are run in a wide operating range around their best efficiency point. In this produces wide range flow rates of the positions of the stagnation point varies in between the tongue and the diffuser vanes. Investigations of flow behavior through the impeller and volute are required for establishing the curve of pump efficiency. This may be performed by means of experimental investigation methods, e.g. PIV and by means of numerical methods, CFD. In this contribution, PIV investigations are performed in order to validate the applied CFD tool for pump efficiency optimization. The boundary layer structure in the diffuser and associated turbulence are strongly affected by the unsteadiness generated by the impeller. In fact, the response of boundary layers in the diffuser to the unsteady head (due to the impeller jet-wake phenomenon upstream) is a dominant phenomenon. The cyclic variations are higher than the turbulent fluctuations within the impeller and between the diffuser vanes, the level of turbulence fluctuations decreases when the distance

between downstream of the trailing edge of the diffuser vanes increased. The impact of impeller blade orientation on the flow structure includes direct effects, such as two types of flow separation within the diffuser passage. The impact of the unsteady outflux from the impeller due to potential effects remain near the leading edge of the diffuser vane, despite the relatively large gap between the impeller and diffuser.

CFD calculations were used to resolve the flow structure and turbulence within the impeller, diffuser and volute of a centrifugal pump in the case of different impeller-diffuser gap orientations. The results demonstrate that the entire flow field is dominated by a series of wakes, generated by the impeller blades, diffuser vanes and separation phenomena that extend well into volute. In traditional pumps, the variations are higher than the turbulent fluctuations through the impeller and diffuser vanes, the turbulence level decreases with increasing the distance between downstream of trailing edge of the impeller and the inlet of the vanes. However, in the present study, the flow inside the pump is hardly affected by decreasing the radial gap. Because a centrifugal pump with a vaned diffuser, satisfying this assumption requires a large gap between the impeller and the diffuser so that the mixing process of the flow leaving the impeller can take place. Otherwise, an unsteady flow at the leading edge of the diffuser vanes represents a source of low efficiency and instability. This mixing process does not occur if radial gap between impeller and diffuser is decreased.

CFD analysis was conducted with the commercial software, called FLUENT. The present numerical analysis was conducted for incompressible, turbulent, steady and unsteady flow. In this study, fluid was assumed to be water, Newtonian such as which has constant density and viscosity. The effects of the rotor-stator interaction on the performance of the centrifugal pump for the steady and unsteady flow were investigated numerically for three radial gap ratios. The presence of the diffuser vanes increases the unsteadiness of the flow downstream of impeller outlet. The highest pressure fluctuations occur with the vaned diffuser and at the suction side of the diffuser vanes. Furthermore, the pressure fluctuation increases with the flow rate. This study also shows similar flow feature in the design and off-

design condition. Numerical calculations of 2D-3D study steady and unsteady flow were performed for different 15 volume flow rates.

The present study is shows a numerical simulation of the 3D fluid flow inside a centrifugal pump. This simulation has been performed for the steady and unsteady flow in order to examine the impeller-diffuser interaction. With this numerical method, it is possible to simulate the blade passage in front of the tongue correctly and the flow and pressure fluctuations induced. The numerical results are compared with the experimental result of PIV technique. The obtained results allow the analysis of the main phenomena existent in these pumps, such as: the pressure changes in the volute for different flow rates; the incidence at the leading edge of the blade with different flow conditions; and the secondary flow generated in the volute due to the width change between the impeller and diffuser.

The present study has demonstrated that the CFD gives a general idea about pump performance and the flow in characteristics inside of the centrifugal pump. For the present calculations, the numerical deviations from the experimental data can be attributed to physical modeling errors. These are; *i)* difference between CFD pump modeling and real pump modeling; *ii)* pump geometry deviation; and *iii)* turbulence models.

Once the numerical model has been validated for measured variables, more detailed flow studies are now then possible and the detailed flow structures can be analyzed. Also, the obtained data found to be useful and valuable in the future designs or for geometrical modifications of a given geometry.

The impeller-diffuser interaction in a centrifugal pump is successfully predicted by a numerical model developed using a finite volume commercial code (FLUENT). The pump performance values and performance curves have been calculated using the numerical results. The changes on the pump geometries can be done by considering the obtained numerical results of the model.

Numerical calculations have been used to study the effects of different radial gap ratios between impeller and diffuser on the flow structure and pressure fluctuations. It is demonstrated that increasing the radial gap, reduces the impact of non-uniform outflux from the impeller (mostly the jet-wake phenomenon) on the

flow around the tongue. When the gap is narrow, the primary sources of pressure fluctuations are impingement of the wake on the tip of the tongue. At the large gap, intensity of the wake and its associated vorticity trains are decreased, and the effect of non-uniform outflux from the impeller occurs when the jet impinges on the tongue.

Based on Navier-Stokes equations and RNG $k-\varepsilon$ turbulent model, the SIMPLEC algorithm is used for the prediction of 3D turbulent flow through the impeller. The comparison between the numerical results and experimental data indicate that the present code appears to be able to yield useful information for flow behaviors under the design and off-design operating conditions. And this kind of calculation can be used for predicting the optimization and performance curve of pump.

The investigation of the internal flow, within a centrifugal pump, has permitted the study of the effect of the presence of a vaned diffuser at different operating conditions. The actual results indicate the existence of an impeller-diffuser interaction. However, diffuser effect on the impeller internal flow is limited to the impeller exit and it does not have any significant upstream influence. The frontier between the diffuser and the impeller does not seem to be easily crossed. The mixing process of the flow at the impeller discharge is affected by the presence of the diffuser vanes and the flow rate. At the design point, the first half of the diffuser pitch is characterized by an early mixing of the flow, whereas, in the second half, the flow entering the diffuser channel is still presenting its periodicity due to the impeller. At partial flow, the flow is more homogeneous in the sections neighboring the diffuser vane's leading edge. As a result, the diffuser performance may be affected differently by the complex flow coming out of the impeller. Further investigations in the diffuser and at different operating conditions are projected to better understand the impeller-diffuser interaction.

In summary, it was found that pressure fluctuations can occur on the impeller blades and diffuser vanes if the radial gap between the impeller blades and the diffuser vanes is small, i.e., in order to a small percentage of the impeller discharge radius. The large pressure fluctuations at the impeller blade trailing edge

may be responsible for the cavitation damage occurring at the trailing edges of the impeller blades of high-speed centrifugal pump. This result should be considered in the structural design of a centrifugal pump.

The impact of impeller blade orientation on the flow structure includes direct effect, such as the jetting ahead of the blade and wake trailing behind the tip of blade, as well as indirect effects, such as flow separation within the diffuser passage.

The centrifugal pump with a vaned diffuser was calculated for validation purposes, and the predicted results steady and unsteady flow showed reasonable agreement with the experimental data. Calculations were carried out for design flow rate level, and the results show satisfactory agreement with well established with experimental data.

From the numerical results and discussions, the following conclusions can be summarized;

- The performance curve for the centrifugal pumps shows that there is good approximation with experimental results under FLUENT solver.
- The jet-wake flow pattern clearly discernable inside the impeller and diffuser channels changing magnitude of size by the flow rate level. This phenomena display that how the complex flow dominates in the pump flow by comparing velocity contours and turbulent kinetic energies especially the higher value accumulated around the impeller blade tips and the diffuser inlet region. The jet-wake flow pattern has affected significantly by changing the diffuser gap distance. When the gap ratio decreases, the jet is influenced within the diffuser because of the flow squeezing in the small gap region. It is also observed that flow is roughly stabilized inside the tongue region. Opposite to that, the wake form is expanded by decreasing the gap ratio which resulted in more un-uniformities around the side of the diffuser walls.
- The pump efficiency has also little affected by the impeller-diffuser gap which decreases when the radial gap decreases in the design flow rate conditions.

- The pressure fluctuation has examined at impeller tip, inside the diffuser and diffuser outlet corresponds to the radial gap and the flow rate. Off-design flow rate has significant effect on pressure fluctuations and especially in the region of tongue (point of C and D) destabilized further due to complex flow but effect of the diffuser has clearly discernable by checking the outlet section of pressure fluctuations. In the case of the radial gap, there is not significant influence on the pump flow behavior where the uniformity increases while the gap decreases especially on the design point.
- Flow is meandering within the diffuser which inherits from the impeller due to centrifugal force effect.
- The switching between the reverse and jet flow in the diffuser channel was observed under the rotating stall condition, which mainly depends on the flow rate level.
- The inlet distortion of the diffuser accelerates the development and merge of the boundary layers of the diffuser walls, hub and shroud side of the walls.
- The axial distortion was not taken into account for the diffuser at smaller axial width by checking previous studies.
- By checking the performance curve with experimental data, it is shown that 3D model results are better than 2D model.
- Comparing with pump performance and efficiency curve, it is noticed that the convergence has good agreement with experimental results especially at low mass flow rate.
- The efficiency versus flow rate level is described for the best of efficiency point in the centrifugal pump. Also, this point is known as design point. It is found that the design point of the mass flow rate is 4.12 L/s.
- The largest vane pressure fluctuations occurred on the vane suction side close to the leading edge. For a large volume flow rates the magnitude of those fluctuations reached to a maximum value in Case III.

- Decreasing the radial gap between impeller blades and diffuser vanes a significant increase in the pressure fluctuations along the front half of the vane suction side resulted. The largest blade pressure fluctuations occurred at the impeller blade trailing edge.

5.1 Future Studies

- Displaying time varying results may serve as a real tool for understanding the interaction between impeller and diffuser.
- Further work should be performed on the influence and a general validation of different turbulence models.
- For the purpose of comparison, a dynamic mesh model can be used in order to see the improvement of the results comparing to the present numerical results.
- It might be useful to work on the grid refinement, for the advanced of the turbulence models and the cavitation effect. Cavitation is often associated with inlet recirculation but was not considered in the present study.
- Impeller-diffuser interaction can be investigated relating to the blade passing frequency.
- The effect of vaneless diffuser on the flow inside pump can also be investigated.
- Further more, detailed experimental investigation will be useful tool far further numerical investigations in this sense, the PIV techniques is a convenient measuring method to use for these types of applications.

REFERENCES

- AKIN, O. and ROCKWELL, D., 1994. "Flow Structure in a Radial Flow Pumping System Using High-Image-Density Particle Image Velocimetry" ASME J. Fluids Eng., 116, pp. 538-544.
- ADAMCZYK, J.J., 1985. "Model Equation For Simulating Flows In Multistage Turbomachinery" ASME Paper No., 85-GT-226.
- ADAMCZYK, J.J., CELESTINA, M.L., BEACH, T.A. and BARNETT, M., 1990. "Simulation of Three-Dimensional Viscous Flow within a Multistage Turbine" ASME J. of Turbomachinery, Vol. 112, p. 370.
- ADKINS, D.R. and BRENNEN, C.E., 1988. "Analysis of Hydrodynamic Radial Forces on Centrifugal Pump Impellers" ASME J. of Fluids Eng., Vol. 110, pp. 20-28.
- ADKINS, G.G. and SMITH, L.H., 1982. "Spanwise Mixing In Axial-Flow Turbomachines" ASME J. of Engineering Power, Vol. 104, p. 97.
- ADRIAN, R.J., 1991. "Particle Imaging Techniques for Experimental Fluid Mechanics", Ann Rev Fluid Mech, Vol. 23, pp. 261-304.
- AKHRAS, A., EL HAJEM, M., MOREL, R. and CHAMPAGNE, J.Y., 2000. "The Internal Flow Investigation of a Centrifugal Pump" 10th International Symposium on Applications of Laser Techniques to Fluid Mechanics, Lisbon, Portugal.
- ANDERSON, J.D. JR., 1995. "Computational Fluid Dynamics" McGraw-Hill.
- ANDERSON, J., WOOD, H.G., ALLAIRE, P.E. and OLSEN, D.B., 2000. "Numerical Analysis of Blood Flow in the Clearance Regions of a Continuous-Flow Artificial Heart Pump" Artificial Organs, Vol. 24, pp. 492-500.
- ARNDT, N., ACOSTA, A.J., BRENNEN, C.E. and CAUGHEY, T.K., 1989. "Rotor/Stator Interaction in a Diffuser Pump" ASME J. Turbomach., 111, pp. 213-221.
- ARNDT, N., ACOSTA, A.J., BRENNEN, C.E. and CAUGHEY, T.K., 1990. "Experimental Investigation of Rotor-Stator Interaction in a Centrifugal Pump with Several Vaned Diffusers" ASME J. Turbomach., 112, pp. 98-108.

- BARNHART, D.H., ADRIAN, R.J. and PAPEN, R.J., 1995. "Phase-Conjugate Holographic System for High-Resolution Particle-Image Velocimetry". *Appl. Opt.*, Vol. 33, pp. 7159.
- BAUN, D.O. and FLACK, R.D., 2003. "Effect of Volute Design and Number of Impeller Blades on Lateral Impeller Forces and Hydraulic Performance" *International Journal of Rotating Machinery*, 9 (2), 145-152.
- BAUN, D.O., KOSTNER, L. and FLACK, R.D., 2000. "Effect of Relative Impeller to Volute Position on Hydraulic Efficiency and Static Radial Force Distribution in a Circular Volute Centrifugal Pump" *ASME Journal of Fluids Engineering*, 122, 598-603.
- BERT, P.F., COMBES, J.F. and KUENY, J.L., 1996. "Unsteady Flow Calculation in a Centrifugal Pump Using a Finite Element Method" XVIII IAHR Symposium on Hydraulic Machinery and Cavitation, Valencia, Spain.
- BLANCO-MARIGORTA, E., FERNANDEZ-FRANCOS, J., GONZALEZ-PEREZ, J. and SANTOLARIA-MORROS, C., 2000a. "Numerical Flow Simulation in a Centrifugal Pump with Impeller-Volute Interaction" FEDSM200-11297, ASME Fluids Eng. Conf., Boston, Massachusetts.
- BLANCO-MARIGORTA, E., FERNANDEZ-FRANCOS, J., PARRONDO-GAYO, J. and SANTOLARIA-MORROS, C., 2000b. "Numerical Simulation of Centrifugal Pumps" FEDSM200-11162, ASME Fluids Eng. Conf., Boston, Massachusetts.
- BRENNEN, C.E., 1994. "Hydrodynamics of Pumps" Oxford University Press and CTEI Inc.
- CAO, S., GOULAS, A., WU, Y., TSUKAMOTO, H., PENG, G., LIU, W., ZHAO, L. and CAO, B., 1999. "Three-Dimensional Turbulent Flow in a Centrifugal Pump Impeller Under Design and Off-Design operating Conditions" FEDSM99-6872 ASME Fluids Engineering Division, San Francisco.
- CHEN, SHIH H. and LAU, LU-FU, 1997. "Flow Field Calculations of a Centrifugal Pump with Volute" Proceedings of the 1997 International Gas

- Turbine & Aeroengine Congress & Exposition, Orlando, FL, USA; Sponsored by ASME; Paper 97-GT-49, American Society of Mechanical Engineers.
- CHU, S., DONG, R. and KATZ, J., 1995. "Relationship Between Unsteady Flow, Pressure Fluctuation, and Noise in a Centrifugal Pump-Part B: Effects of Blade-Tongue Interactions" ASME J. Fluids Eng. Vol. 117, pp. 30-35.
- CROBA D.J. and KUENY L., 1996. "Numerical Calculation of 2-D, Unsteady Flow in Centrifugal Pumps: Impeller and Volute Interaction" International Journal for Numerical Methods in Engineering, Vol. 22 (6), pp. 467-481.
- DAWES, W.N., 1986. "A Numerical Method for the Analysis of Three-Dimensional Viscous Compressible Flow in a Turbine Cascade: Application to Secondary Flow Development in a Cascade with and without Dihedral" ASME Paper 86-GT-145. New York: American Society of Mechanical Engineers.
- DEAN, R. and SENOO, Y., 1960. "Rotating Wake in Vaneless Diffusers" ASME J. Basic Eng., 82, p.53.
- DENTON, J.D. and DAWES, W.N., 1999. "Computational Fluid Dynamics For Turbomachinery Design" Proceedings of the Institution Of Mechanical Engineers, Vol. 213, Part C, pp. 107-124.
- DENUS, C.K. and GÖDE, E., 1999. "A Study in Design and CFD Analysis of a Mixed-Flow Pump Impeller" ASME-FEDSM-99-6858.
- DIXON, S.L., 1978. "Fluid Mechanics, Thermodynamics of Turbomachinery" Pergamon Press Ltd. Oxford.
- DONG, R., CHU, S. and KATZ, J., 1992a. "Quantitative Visualization of the Flow within the Volute of a Centrifugal Pump, part A: Technique" Journal of Fluids Engineering, Vol. 114, No. 3, pp. 390-395.
- DONG, R., CHU, S. and KATZ, J., 1992b. "Quantitative Visualization of the Flow Structure within the Volute of a Centrifugal Pump, part B: Results and Analysis" Journal of Fluids Engineering, Vol. 114, No. 3, pp. 396-403.
- DONG, R., CHU, S. and KATZ, J., 1997. "Effect of Modification to Tongue and Impeller Geometry in Study Flow", Pressure Fluctuations, and Noise in a Centrifugal Pump" Transaction of ASME, Vol 119, pp. 506-515.

- DOUGLAS, J.F., GASIOREK J.M.K. and SWAFFIELD J.A., 2000. "Fluid Mechanics" Prentice Hall
- DRING, R.P., JOSLYN, H.D., HARDWIN, L.W. and WAGNER, J.H., 1982. "Turbine Rotor-Stator Interaction" ASME J. Eng. Power, 104, pp. 729–742.
- ECKARDT, D., 1976. "Detailed Flow Investigations within a High-Speed Centrifugal Compressor Impeller" ASME J. Fluids Eng., 88, pp. 390–402.
- EISELE, K., ZHANG, Z. and CASEY, M.V., 1997, "Flow Analysis in a Pump Diffuser- Part 1: LDA and PTV Measurements of the Unsteady Flow", J. of Fluids Eng., 119, pp. 968-977.
- ENGEDA, A., 1998. "From the Crystal Palace to the Pump Room" International Gas Turbine&Aeroengine Congress, Stockholm, Sweden, pp. 98-GT-22.
- FILIPENCO, V.G., DENIZ, S., JOHNSTON, J.M., GREITZER, E.M. and CUMPSTY, N.A., 1998. "Effects of Inlet Flow Field Conditions on the Performance of Centrifugal Compressor Diffusers Part 1: Discrete-Passage Diffuser" ASME Paper No. 98-GT-473.
- FISHER, E.H., 1981. "A study of Diffuser/Rotor Interaction in a Centrifugal Compressor" Journal Mechanical Engineering Science, Vol 23, No 3, pp. 149-156.
- FLUENT User's Guide, 2000.
- FORTES-PATELLA, R., LONGATTE, F., KUENY, J.L. and CROBA, D., 1995. "Numerical Analysis of Unsteady Flow in a Centrifugal Pump" ASME Fluid Machinery, FED-Vol. 222, pp. 41–46.
- GALLIMORE, S.J. and CUMPSTY, N.A., 1986. "Spanwise Mixing In Multistage Axial Flow Compressors: Part I- Experimental Investigations" ASME J. Of Turbomachinery, Vol. 108, pp. 2.
- GALLUS, H.E, LAMBERTZ, J. and WALLMANN, T., 1980, "Blade-Row Interaction in an Axial Flow Subsonic Compressor Stage", ASME J. of Engineering for Power, 102, 169-177.

- GAYO, J.L.P., 2000. "An Experimental Study on the Unsteady Pressure Distribution Around the Impeller Outlet of a Centrifugal Pump" Proceedings of ASME FEDSM'00, ASME 2000 Fluids Engineering Division Summer Meeting.
- GONZALEZ, J., BLANCO, E., SANTOLARIA, C. and FERNANDEZ, J., 2002 (a). "Unsteady Flow Structure on a Centrifugal Pump: Experimental and Numerical Approaches" Proceeding of the ASME FEDSM'02 2002 Joint US- European Fluids Engineering Summer Conference, Montreal, Canada, pp. 1-8.
- GONZALEZ, J., FERNANDEZ, J., BLANCO, E. and SANTOLARIA, C., 2002 (b). "Numerical Simulation of the Dynamic Effects Due to Impeller-Volute Interaction in a Centrifugal Pump" Journal of Fluids Engineering, Vol. 124, pp. 348-355.
- GONZALEZ, J., FRANCISCO, C., SANTOLARIA, C. and PARRA, M. T., 2003 (a). "Numerical Model for the Unsteady Flow Behavior Inside a Double Suction Pump" Proceeding of the ASME FEDSM'03 4th ASME_JSME Joint Fluids Engineering Conference, Honolulu, Hawaii, USA.
- GONZALEZ, J., SANTOLARIA, PARRONDO, J. L., FERNANDEZ, J. and BLANCO, E., 2003 (b). "Unsteady Radial Forces on the Impeller of a Centrifugal Pump with Radial Gap Variation" Proceeding of the ASME FEDSM'03 4th ASME_JSME Joint Fluids Engineering Conference, Honolulu, Hawaii, USA.
- GOTO, A., 1992. "Study of Internal Flows in a Mixed-Flow Pump Impeller at Various Tip Clearances Using Three-Dimensional Viscous Flow Computations" ASME Journal of Turbomachinery, Vol. 114, pp. 373-382.
- GULEREN K. M. and PINARBAŞI A., 2004. "Numerical Simulation of the Stalled flow within a Vaned Centrifugal Pump", Proc. Inst. Mech. Engrs. J. Mechanical Engineering Science, Vol. 18, Part C, pp1-11.
- GUNZBURGER, M.D. and NICOLAIDES, R.A., 1993. Incompressible Computational Fluid Dynamics. Trends and Advances, Cambridge University Press.

- HAUPT U., SEIDEL U., ABDEL-HAMID A.N. and RAUTENBERG M., 1988. "Unsteady Flow in a Centrifugal Compressor With Different Types of Vaned Diffuser" *Journal of Turbomachinery*, Vol 110, pp. 293-302.
- HEILMANN, C.C. and SIEKMANN, H.E., 2002. "Particle Image Velocimetry as CFD-Validation Tool for Flow Field Investigations in Centrifugal Pumps" *The 9th of International Symposium on Transport Phenomena and Dynamics of Rotating Machinery*, Honolulu, Hawaii, pp. 1-10.
- HERGT, P., 1999. "Pump Research and Development: Past, Present, and Future" *ASME Transactions, Journal of Fluids Engineering*, 121, pp. 248-253.
- IINO, T. and KASAI, K., 1985. "An Analysis of Unsteady Flow Induced by Interaction Between a Centrifugal Impeller and a Vaned Diffuser (in Japanese). *Trans. Japan Soc. of Mech. Eng.*, 51, No.471, pp. 154-159.
- INOUE, M. and CUMPTSY, N.A., 1984. "Experimental Study of Centrifugal Impeller Discharge Flow in Vaneless and Vaned Diffusers" *ASME J. Eng. Gas Turbines Power*, 106, pp. 455-467.
- IVERSEN, H.W., 1960. "Volute Pressure Distribution, Radial Forces on the Impeller, and Volute Mixing Losses of a Radial Flow Centrifugal Pump" *ASME J. of Engineering for Power*, pp. 136-144.
- JOHNSTON J.P. and DEAN R.C., 1966. "Losses in Vaneless Diffusers of Centrifugal Compressors and Pumps" *ASME J. Eng. For Power*, 88, pp. 49-62.
- KAUPERT, K.A., HOLBEIN, P. and STAUBLI, T., 1996. "A First Analysis of Flow Field Hysteresis in a Pump Impeller" *Journal of Fluids Engineering*, Vol. 118, pp. 685-691
- KELECY F.J., "Numerical Prediction of Cavitation in a Centrifugal Pump" *QNET-CFD Network News Letter* Vol.2, No.3, pp. 14-16
- KRAIN, H., 1988. "Swirling Impeller Flow" *ASME J. Turbomach.*, 110, pp.122-128.
- LAKSHMINARAYANA, B., 1991. "An Assessment of Computational Fluid Dynamic Techniques in the Analysis and Design of Turbomachinery- The 1990 Freeman Scholar Lecture" *ASME J. Fluid Eng.*, Vol. 113, pp. 315-352.

- MARX, Y.P., 1997. "Unsteady Simulations of Impeller-Diffuser Interactions in a Centrifugal Pump" EPFL Supercomputing Review No 9, pp. 24-32.
- MENEVEAU, C. and KATZ, J., 2002. "A Deterministic Stress Model for Rotor-Stator Interactions in Simulations of Average-Passage Flow" *Journal of Fluids Engineering*, Vol. 124, pp. 550-554.
- MIYAZOE, Y., TOSHIO, S., ITO, K., KONISHI, Y., YAMANE, T., NISHIDA, M., ASZTALOS, B., MASUZAWA, T., TSUKIYA, T., ENDO, S. and TAENAKA, Y., 1999. "Computational Fluid Dynamics Analysis to Establish the Design Process of a Centrifugal Blood Pump: Second Report" *Artificial Organs* Vol. 23, pp. 762-768.
- MUGGLI, F.A., EISELE, K., CASEY, M.V., GULICH, J.F. and SCHACHENMANN, A., 1997. "Flow Analysis in a pump diffuser- Part 2: Validation and limitations of CFD for diffuser flows" *J. of Fluids Eng.*, Vol. 119, pp. 978-984.
- NEUMANN, B., 1991. "The Interaction Between Geometry and Performance of a Centrifugal Pump" Mechanical Engineering Publications Ltd.
- OGATA, M. and ICHIRO A., 1995. "An experimental study of Rotating Stall in a Radial Vaned diffuser" *Unsteady Aerodynamics and Aeroelasticity of Turbomachines*, pp. 625-641.
- OH, J.S. and RO, S.H., 2000. "Application of Time Marching Method to Incompressible Centrifugal Pump Flow" *Proceedings of the 2nd International Symposium on Fluid Machinery and Fluid Engineering*. Beijing: Tsinghua University Press, pp. 219-225.
- POTTS, I. and NEWTON, T.M., 1998. "Use of a Commercial CFD Package to Predict Shut-Off Behavior of a Model Centrifugal Pump: An Appraisal" IMechE Seminar Publication: *CFD in Fluid Machinery Design*, London: Institute of Mechanical Engineers.
- QIAN, Y., ARAKAWA, C., KUBOTA, T. and SUZUKI, R., 1994. "Numerical Flow Simulation on Channel Vortex in Francis Runner" *Proceedings of 17th IAHR Symp.*, Beijing, pp. 45-57.

- QIN, W. and TSUKAMOTO, H., 1997. "Theoretical Study of Pressure Fluctuations Downstream of a Diffuser Pump Impeller-Part 1: Fundamental Analysis on Rotor-Stator Interaction" ASME J. Fluids Eng., 119, pp. 647-652.
- RAI, M.M., 1987. "Navier-Stokes Simulation Of Rotor/Stator Interaction Using Patched And Overlaid Grids" J. Of Propulsion And Power, Vol. 3, pp. 387.
- RHIE, C.M., GLEIXNER, A.J., SPEAR, D.A., FISCHBERG, C.J., and ZACHARIAS, R.M., 1995. "Development and Application Of A Multistage Navier-Stokes Solver. Part A: Multistage Modeling Using Body Forces and Deterministic Stresses" ASME Int. Gas Turbine and Aeroengine Congress and Exposition, 95-GT-342.
- RIBI, B., 1996. "Instability Phenomena in Centrifugal Compressors in Flow in Radial Turbomachines" Von Karman Institute Lecture series No.1996-01, editor: R.A. Van den Braembussche.
- ROTH G. and KATZ J., 1999. "Parallel Truncated Multiplication and other Methods for Improving the Speed and Accuracy of PIV Calculations" FEDSM-6998 Proc. ASME Fluids Eng. Conf., San Francisco.
- ROTH, G. and KATZ, J., 2001. "Five Techniques for Increasing the Speed and Accuracy of PIV Interrogation" Measurement Science and Technology Journal, Vol. 12, pp. 238-245.
- SANO T., YOSHIDA Y., TSUJIMOTO Y., NAKAMURA Y. and MATHUSHIMA T., 2002. "Numerical Study of Rotating Stall in a Pump Vaned Diffuser" ASME Journal of Fluids Engineering, Vol. 124, pp. 363-370.
- SATO K. and HE, L., 2001. "Numerical Investigation into the Effects of a Radial Gap on Hydraulic Turbine Performance" Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy, Vol. 215 (1), pp. 99-108.
- SCHULZ, H., GREIM, R. and VOLGMANN, W., 1996. "Calculation of Three-Dimensional Viscous Flow in Hydrodynamic Torque Converters" ASME J. Turbomach., 118, pp. 578-589.

- SHAW, C.T., 1992. "Using Computational Fluid Dynamics" Prentice Hall International (UK) Ltd.
- SHI, F. and TSUKAMOTO, H., 2001. "Numerical Study of Pressure Fluctuations Caused by Impeller-Diffuser Interaction in a diffuser Pump Stage" 123, pp. 466-474.
- SINHA, M. and KATZ, J., 1998. "Flow Structure and Turbulence in a Centrifugal Pump with a Vaned Diffuser" ASME Proceedings of Fluids Engineering Division Summer Meeting 1998.
- SINHA, M., 1999. "Rotor-Stator Interactions, Turbulence Modeling and Rotating Stall in a Centrifugal Pump with Diffuser Vanes" Ph.D. Dissertation, The Johns Hopkins University, Baltimore, MD.
- SINHA, M., PINARBAŞI, A. and KATZ, J., 2000. "On the Onset and Development of Rotating Stall within a Vaned Diffuser of a Centrifugal Pump" ASME/JSME Fluids Engineering Division Summer Meeting, FEDSM2000-11241.
- SINHA, M., PINARBAŞI, A. and KATZ, J., 2001. "The Flow Structure During Onset and Developed States of Rotating Stall Within a Vaned Diffuser of a Centrifugal Pump" Journal of Fluids Engineering, Vol. 123, pp. 490-499.
- SINHA, M. and KATZ, J., 2000. "Quantitative Visualization of the Flow in a Centrifugal Pump with Diffuser Vanes, part A: On Flow Structure and Turbulence" Journal of Fluids Engineering, Vol. 122, No. 1, pp. 97-107.
- SINHA, M., KATZ, J. and MENEVEAU, C., 2000. "Quantitative Visualization of the Flow in a Centrifugal Pump with Diffuser Vanes, part B: Addressing Passage Averaged and LES Modeling Issues in Turbomachinery flows" Journal of Fluids Engineering, Vol. 122, No. 1, pp. 108-116.
- SPEZIALE, C., 1991. "Analytical Methods for the Development of Reynolds-Stress Closures in Turbulence" Ann. Rev. Fluid Mech., Vol. 23, pp. 107.
- STEPHANOFF, A.J., 1957. "Centrifugal and Axial Flow Pumps" J. Wiley and Sons, New York.

- SUN, J. and TSUKAMATO, H., 2001. "Off-Design Performance Prediction for Diffuser Pumps" *Journal of Power and Energy, Proceedings of I. Mech. E., Part A*, Vol. 215, pp. 191-201.
- TATEBAYASHI, Y., TANAKA, K., HAN, H. and KOBAYASHI, T., 2000. "A 3-D Simulation of Flow in a Screw-Type Centrifugal Pump with Tip Clearance" *Proceedings of the 2nd International Symposium on Fluid Machinery and Fluid Engineering*. Beijing: Tsinghua University Press, pp. 608-615.
- TSUKAMOTO, H., UNO, M., HAMAFUKU, H. and OKAMURA, T., 1995. "Pressure Fluctuation Downstream of a Diffuser Pump Impeller" *The 2nd Joint ASME/JSME Fluids Engineering Conference, Forum of Unsteady Flow, FED-Vol.216*, pp. 133-138.
- TSUKAMOTO, H., UNO, M., HAMAFUKU, N. and OKAMURA, T., 1995. "Pressure Fluctuation Downstream of a Diffuser Pump Impeller" *ASME FED*, Vol. 216, pp. 133-138.
- TSURUSAKI, H. and KINOSHITA T., 1999. "Flow Control of Rotating Stall in a Radial Vaneless Diffuser" *FEDSM99-7199*, *ASME Fluids Eng. Conf.*, San Francisco.
- TUZSON, J., 2000. "Centrifugal Pump Design" J. Wiley and Sons, New York.
- UBALDI, M., ZUNINO, P., BARIGOZZI, G. and CATTANEI, A., 1996. "An Experimental Investigation of Stator Induced Unsteadiness on Centrifugal Impeller Out-flow" *ASME J. Turbomach.*, 118, pp. 41-54.
- VALENTI, M., 1996. "CFD Software Improves Pump Design" *Mechanical Engineering*, Vol. 118, pp. 82.
- VERSTEEG, H.K. and MALALASEKERA, W., 1995. "An Introduction to Computational Fluid Dynamics" Longman Scientific and Technical, Essex.
- VISSER, F.C., BROUWERS, J.J.H. and JONKER, J.B., 1999. "Fluid Flow in a Rotating Low-Specific-Speed Centrifugal Impeller Passage" *Fluid Dynamics Research*, Vol. 24, pp 275-292.
- WUIBAUT, G., DUPONT, P., BOIS, G., CAIGNAERT, G. and STANISLAS, M., 2001. "Particle Image Velocimetry Applied to the Measurement of Flow Fields

both in the Impeller and the Diffuser of a Centrifugal Pump” *Houille Blanche*, Issue 2, pp. 75-80.

YAMANE, T. and NAGASHIMA, T., 1997. “High Speed Centrifugal Impeller and Diffuser Interaction near Stall Conditions” *Proceedings of the 8th International Symposium*, Stockholm, Sweden.

ZHANG, J., TAO, B. and KATZ, J., 1997. “Turbulent Flow Measurement in a Square Duct with Hybrid Holographic PIV” *Experiments In Fluids*, Vol. 23, p. 373.



CURRICULUM VITAE

Personal Details

Name : Adnan ÖZTÜRK
Date Place of Birth : 12 December 1965, Sivas, TURKEY
Nationality : Turkish
Marital Status : Married, two child
Military Service : 1988-1989 (LFC-Reserve Officer)
Present Position : Research Assistant, Mechanical Engineering
Department, University of Çukurova, Adana, TURKEY
Address : University of Çukurova, Mechanical Engineering
Department, 01330, Adana, TURKEY
Telephone : +90-322-338 60 84 / 2722
Fax : +90-322-338 61 26
GSM : +90-535-582 47 20
E-mail : adozturk@cu.edu.tr, aozturk@cumhuriyet.edu.tr

Educational Background

1976 : Ülkü Primary School, Sivas, TURKEY
1979 : Atatürk High School, Sivas, TURKEY
1982 : Technical and Industrial Vocational High School, Sivas,
TURKEY
1988 : Bsc, Department of Mechanical Engineering, Zonguldak
Engineering Faculty, University of Hacettepe, Ankara, TURKEY
1997 : Msc, Department of Mechanical Engineering, University of
Cumhuriyet, Sivas, TURKEY
1999-present : Phd, University of Çukurova, Mechanical Engineering
Department, Adana, TURKEY

Research Activity

Fluid Dynamics, CFD (Computational Fluid Dynamics), CFD Package Programs,
Turbomachines, Centrifugal Pumps

Employment

1990-1991 : Mechanical Engineer, Nurol A.Ş. (Sivas Demir Çelik Fabrikası
A.Ş.), Ankara, TURKEY
1992-1996 : Lecturer, Vocational School of Divriği, Cumhuriyet University,
Sivas, TURKEY
1997-1999 : Research Assist., Department of Mechanical Engineering,
University of Cumhuriyet, Sivas, TURKEY
1999-present : Research Assist., University of Çukurova, Mechanical Engineering
Department, Adana, TURKEY

Membership of Institutions

Union of Chambers of Turkish Engineers and Architects, Chamber of Mechanical
Engineers (TMMOB Makina Mühendisleri Odası)