BORANG PENGESAHAN STATUS TESIS

JUDUL: **MODELLING AND SIMULATION OF AN ASYMMETRIC PLANE DIFFUSER**

SESI PENGAJIAN: 2007/2008

Saya **SITI QHADIJAH BT ABDUL RASHID**

(HURUF BESAR)

mengaku membenarkan tesis (PSM/Sarjana/Doktor Falsafah)* ini disimpan di Knowledge Management Center dengan syarat-syarat kegunaan seperti berikut:

1. Tesis adalah hakmilik Kolej Univeristi Teknologi Malaysia.
2. Knowledge Management Center dibenarkan membuat salinan untuk tujuan pengajian sahaja.
3. Perpustakaan dibenarkan membuat salinan tesis ini sebagai bahan pertukaran antara institusi pengajian tinggi.
4. **Sila tandakan (4)

☐	SULIT	(Mengandungi maklumat yang berdarjah keselamatan atau kepentingan Malaysia seperti yang termaktub di dalam AKTA RAHSIA RASMI 1972)
☐	TERHAD	(Mengandungi maklumat TERHAD yang telah ditentukan oleh organisasi/badan di mana penyelidikan dijalankan)
√	TIDAK TERHAD	

Disahkan oleh

(TANDATANGAN PENULIS)	(TANDATANGAN PENYELIA)

Alamat Tetap:
459 A,JALAN KAMPUNG MASJID,
11920, BAYAN LEPAS
PULAU PINANG

EN. MOHD FAIRUSHAM BIN GHAZALI
Nama Penyelia

Tarikh: **23 NOVEMBER 2007**

Tarikh: **23 NOVEMBER 2007**

CATATAN:
* Potong yang tidak berkenaan.
** Jika tesis ini SULIT atau TERHAD, sila lampirkan surat daripada pihak berkuasa/organisasiberkenaan dengan menyatakan sekali sebab dan tempoh tesis ini perlu dikelaskan sebagai SULIT atau TERHAD.
υ Tesis dimaksudkan sebagai tesis bagi Ijazah Doktor Falsafah dan Sarjana secarapenyelidikan, atau disertasi bagi pengajian secara

MODELING AND SIMULATION OF AN ASYMMETRIC PLANE DIFFUSER

SITI QHADIJAH BT ABDUL RASHID

A report submitted in partial fulfilment of the
requirements for the award of the degree of
Bachelor of Mechanical Engineering

Faculty of Mechanical Engineering
Universiti Malaysia Pahang

NOVEMBER 2007

Supervisor's Declaration

"I hereby declare that I have read this project report and in my opinion this project report is sufficient in terms of scope and quality for the award of the Degree of Mechanical Engineering"

Signature	:	....
Name of Supervisor	:	Mr Mohd Fairusham B Ghazali
Date	:	23 November 2007

I declare that this thesis entitled " Modeling and Simulation of An Asymmetric Plane Diffuser is the result of my own research except as cited in the references. This thesis has not been accepted of any degree and is not concurrently submitted in candidature of any other degree.

Signature : ....

Name : SITI QHADIJAH ABD RASHID

Date : 23/11/07

To my beloved Father and Mother,

Specially for my siblings and also other family members

Thanks for your love and supports

ACKNOWLEDGEMENT

In preparing this thesis, I am indeed indebted and wish to express my appreciation to Mr Fairusham Bin Ghazali who are my supervisor. He encouraged and guided me in doing this project. He has contributed my understanding and thoughts regarding this project. He is willing to spend extra time in order to help me completed this project.

I am also indebted with Universiti Malaysia Pahang for my bachelor study, librarians at UMP. I felt thankful to have supportive friends, who always share the difficulties and solutions to the certain problems.

I am grateful to have understanding family members who always give full support in completing my project especially my parent and members of the family.

ABSTRACT

This thesis deals with modeling and simulation of an asymmetric plane diffuser which is the most interesting test case in flow field by using Fluent software. The objective of this thesis are to understand the physical of the flow field in the plane diffuser, to evaluate turbulence model performance in predicting flow field using CFD commercial code, FLUENT and to compare with experimental data. The flow behavior includes pressure distribution and velocity profiles in asymmetric plane diffuser was investigated using three turbulence modeling which are realizable k-ε model, standard k- ω model and SST k- ω model. The results from the simulation was compared to the experimental data (Buice & Eaton 1997) in order to produce accurate result. The acquired results of pressure distribution shows that both k- ω model provide a fairly accurate in beginning stage while realizable k- ε shows the worst result. For velocity profiles within the diffuser, the simulation results are much different from the experiment except the bottom half of the diffuser. The realizable k- ε model being the worst followed by standard k- ω model while SST k- ω model gives better result. For the recirculation region, SST k- ω model predicts the biggest region while realizable k-ε model shows the smallest. The CFD commercial codes can solve problems faster and the best way to determined the fluid flow in this project.

ABSTRAK

Tesis ini memperkatakan tentang pemodelan dan simulasi pada penyebar kapal tidak simetri dimana ia adalah kes ujian yang paling menarik dalam bidang aliran dengan menggunakan perisian Fluent. Objektif tesis ini adalah untuk memahami aliran fizikal di dalam penyebar, menilai perlaksanaan model kegeloraan dalam meramalkan aliran menggunakan perisisan Fluent dan membandingkan dengan data eksperimen. Aliran yang bertindak termasuklah penyebaran tekanan dan profil halaju dalam penyebar diuji dengan menggunakan tiga model turbulent iaitu model k-ε , model standard k- ω dan model SST k- ω. Keputusan dari simulasi di bandingkan dengan data eksperimen (Buice & Eaton 1997) untuk menghasilkan keputusan yang tepat. Keputusan yang diperolehi daripada penyebaran tekanan menunjukkan kedua-dua model k- ω menghasilkan keputusan yang agak tepat pada peringkat awal manakala model k-ε menunjukkan keputusan yang tidak memuaskan. Untuk profil halaju pula, keputusan simulasi sedikit berlainan daripada data eksperimen kecuali pada dasar penyebar. Model k- ε menunujukkan keputusan terburuk diikuti dengan model k- ω manakala model SST k- ω menunjukkan keputusan yang terbaik. Untuk kawasan pengaliran balik, model SST k- ω menunjukkan kawasan yang terbesar manakala model k- ω menunujukkan kawasan pegaliran balik yang terkecil. Perisian aliran (CFD) dapat menyelesaikan masalah aliran dengan cepat dan adalah cara yang terbaik untuk menentukan bidang aliran cecair dalam projek ini.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF ABBREVIATIONS

CFD	-	Computational Fluid Dynamics
DNS	-	Direct Numerical Solution
LES	-	Large Eddy Simulation
RANS	-	Reynolds Average Navier-Stokes Equation
RSM	-	Reynolds Stress Model
SST k- ϖ	-	Shear Stress Transport k- ϖ Model

LIST OF SYMBOLS

P	-	Static Pressure
U,u	-	Fluid Velocity
U_b	-	Fluid Bulk Velocity
u	-	Mean Fluid Velocity Component
$u(t)$	-	Fluctuating Fluid Velocity Component
3-D	-	Three Dimensional
2-D	-	Two Dimensional
H	-	Height of Inlet of Diffuser
Re	-	Reynolds Number
ρ	-	Fluid Density
τ_w	-	Wall Shear Stress
μ	-	Kinematic Viscosity
μ_t	-	Turbulence Viscosity
k	-	Turbulent Kinetic Energy
ε	-	Dissipation of Turbulent Kinetic Energy
P_k	-	Production of Turbulent Kinetic Energy
C_μ	-	Turbulent Viscosity Constant
$C_{\varepsilon 1}$	-	Standard k- ε constant
$C_{\varepsilon 2}$	-	Standard k- ε constant
σ_ε	-	Standard k- ε constant
ϖ	-	Inverse Time Scale
β	-	Standard k- ϖ constant
σ_k^*	-	Standard k- ϖ constant
α	-	Standard k- ϖ constant
β	-	Standard k- ϖ constant

σ_{ϖ}^{*}	-	Standard k- ϖ constant
y	-	Distance normal to the wall
C_f	-	Skin Friction Coefficient

LIST OF APPENDICES

CHAPTER 1

INTRODUCTION

1.1 Project Background

Flow in the asymmetric plane diffuser is considered by researchers as one of the most interesting test cases due to the flow characteristics that occur in the flow. These characteristics include flow separation of a fully developed turbulent flow, due to an adverse pressure gradient generated by the channel expansion. Flow reattachment, and redevelopment of the flow also occurs downstream of the expansion. The problem is taken as a general case of situations where separation occurs either on a flat plane or a gently curved surface. The problem is similar in nature to what happens on the suction side of an airfoil as it reaches stall conditions (separation), yet the geometry is confined and thus requires fewer nodes than the stalling airfoil problem. The results from this test case can thus be use to validate and verify turbulence models that can then be applied to these problems with confidence in the mechanical engineering sector. The project is entirely based upon the use of FLUENT.

1.2 Problem Statement

The experimental test case presented in this project is a separated flow in an asymmetric plane diffuser. A diffuser is the mechanical device that is designed to control the characteristics of a fluid at the entrance to a thermodynamic open system. Diffusers are used to slow the fluid's velocity and to enhance its mixing into the surrounding fluid. In this plane diffuser, the critical part that to be analyze is at the inclined wall where the circulation of the fluid flow occurs. The degree of the tangential and the inclined wall is only about 10^{o}. So, it is very difficult to determine the velocity and pressure of the fluid flow inside the plane diffuser by experimental way. It takes so much time and need to build the real plane diffuser. It is also very highly cost because if the error occurs, we need to do it all over again. Nowadays, the high technology of Computational Fluid Dynamics (CFD) can easily be used in order to simulate the fluid flow of the plane diffuser. It gives more advantages such as, not wasting so much time and low cost in doing this research. For example, if the errors occurs, we can easily change the method or some value until we get the satisfied value. Besides, we solve bigger problems faster, and has been proven on the widest possible variety of platforms in the industry. In this high technology world, people usually search for the effective, faster and accurate method. CFD commercial codes are the best way to determined the fluid flow in this project.

1.3 Objectives of Project

To measure the extent of one project should go; some objectives need to set in order to ensure the success of the project. Therefore, the objectives of this project are :

i. To understand the physic of the flow field in the plane diffuser.
ii. To evaluate turbulence model performance in predicting flow field using CFD commercial code, FLUENT.
iii. To compare with available experimental data.

1.4 Scopes of Project

Scopes are important steps or procedures to be applied in achieving objectives. Based on objectives above, the scope of this project are defines as follow:

i. Simulation studies using CFD commercial code in asymmetric plane diffuser.
ii. Use the k-ε, k-ω and SST k-ω turbulence model in FLUENT
iii. Geometry setup and mesh models for test case using GAMBIT

1.5 Flow Chart

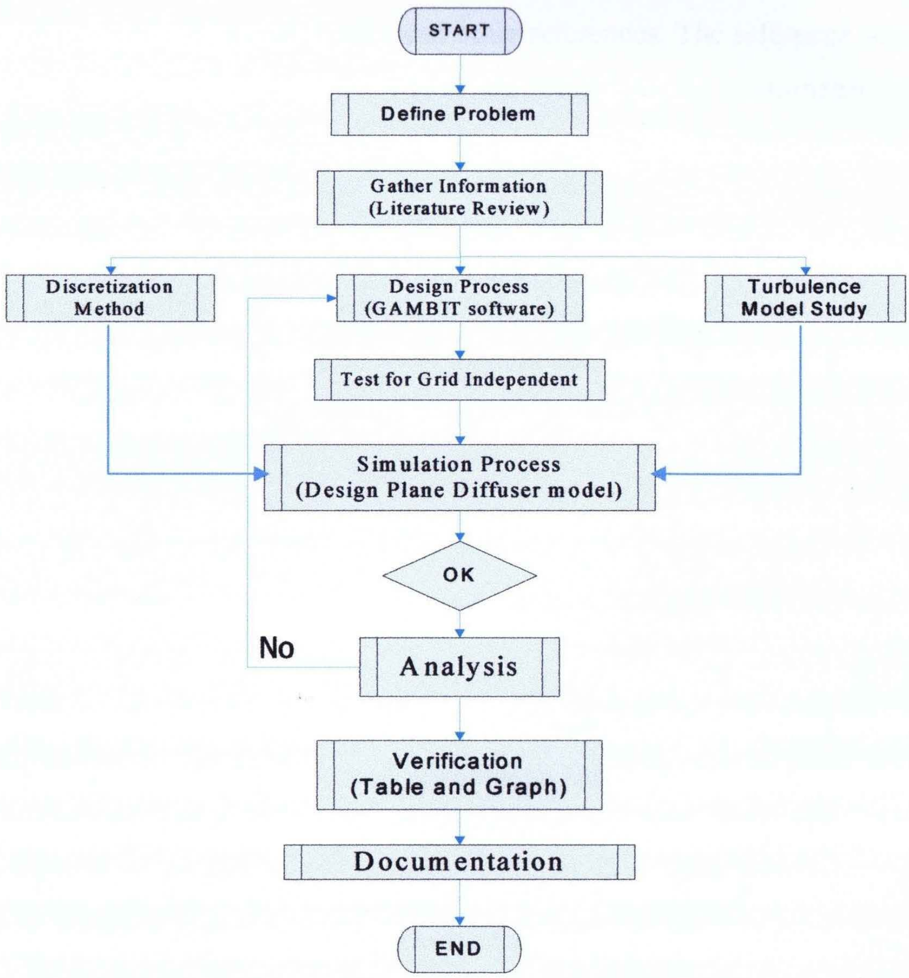


Figure 1.1 Project Flow Chart

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

In order to perform this project, literature review has been made from various sources likewise journal, books and other references. The reference sources emphasize on few important aspects which are related to the asymmetric plane diffuser.

This chapter will described about asymmetric plane diffuser, turbulence, the near wall consideration in turbulence model, Computational Fluid Dynamics (CFD), turbulence modeling, discretization method and will be discussed about the test case of the plane diffuser.

2.2 Computational Fluid Dynamics (CFD)

Computational Fluid Dynamics or CFD is the analysis of systems involving fluid flow, heat transfer and associated phenomena such as chemical reactions by means of computer based simulation. Computers are used to perform the millions of calculations required to simulate the interaction of fluids and gases with the complex surfaces used in engineering. However, even with simplified equations and high-speed supercomputers, only approximate solutions can be achieved in many cases. More accurate codes that can accurately and quickly simulate even complex scenarios such as supersonic or turbulent flows are an ongoing area of research. This technique is very powerful and spans a wide range of industrial and non-industrial applications areas. For examples in aerodynamics of aircraft and vehicles, hydrodynamics of ships, power plant, turbomachinery and many more. This software actually economical and sufficiently complete. There are several unique advantages of CFD over experiment-based approaches to fluid design :

- Substantial reduction of lead times and costs of new designs
- Ability to study systems where controlled experiments are difficult or impossible to perform.
- Ability to study systems under hazardous conditions at and beyond their normal performance limits.
- Practically unlimited level of detail of results.

The fundamental basis of any CFD problem is the Navier-Stokes equations, which define any single-phase fluid flow.

2.3 Fluent

FLUENT is a renowned computer programme of modeling and simulating fluid flows and heat transfer problems in order to solve the fluid equations and produce the required solutions. FLUENT's pre-processor is another program called GAMBIT, which provides the tool of building and applying the mesh on the geometrical representation of the problem concerned. Since meshing is an important process of the CFD analysis, as too coarse a grid since would have very high errors in the solution and too fine a grid would only waste computational time, GAMBIT provides the user with easy commands of modifying the mesh as required. The processing and post-processing stages are done by FLUENT. This program enables the user to choose and set the discretization method, model material and convergence criterions, among others, used for the analysis. Among commercial CFD software products, FLUENT has the largest array of industrially tested capabilities - some 1,000 physical models. These models are remarkably robust, with associated features to accelerate convergence every time. Because FLUENT uses unstructured, hybrid modeling technology, models can be built that conform to arbitrary geometric shapes and other complex surfaces. As a result, your CFD model will have accuracy it needs, where it is needed. These are the advantages of using FLUENT software:

i) Ease-of-Use

- FLUENT allows you to make changes to the analysis at any time during the setup, solution, or post processing phase. This saves time and enables you to refine your designs efficiently. The intuitive interface makes learning easy. Smart panels show only the modeling options that are appropriate for the problem setup at hand. CAD geometries are easily imported and adapted for CFD solutions.

ii) Speed

- Solver enhancements and numerical algorithms that decrease the time to solution are included in every new release of FLUENT software. Our mature, robust, and flexible parallel processing capability enables you to solve bigger problems faster, and has been proven on the widest possible variety of platforms in the industry

iii) Powerful Visualation

- FLUENT's post processing provides several levels of reporting, so you can satisfy the needs and interests of all audiences. Quantitative data analysis can be as rigorous as you require. High resolution images and animations allow you to communicate your results with impact. Numerous data export options are available for integration with structural analysis and other CAE software programs.

2.4 Gambit

GAMBIT is Fluent's geometry and mesh generation software. GAMBIT's single interface for geometry creation and meshing brings together most of Fluent's preprocessing technologies in one environment. Advanced tools for journaling let you edit and conveniently replay model building sessions for parametric studies. GAMBIT's combination of CAD interoperability, geometry cleanup, decomposition and meshing tools results in one of the easiest, fastest, and most straightforward preprocessing paths from CAD to quality CFD meshes.

As a state-of-the-art preprocessor for engineering analysis, GAMBIT has several geometry and meshing tools in a powerful, flexible, tightly-integrated, and easy-to use interface. GAMBIT can dramatically reduce preprocessing times for many applications. Most models can be built directly within GAMBIT's solid geometry modeler, or imported from any major CAD/CAE system. Using a virtual geometry overlay and advanced cleanup tools, imported geometries are quickly converted into suitable flow domains. A comprehensive set of highly automated and size function driven meshing tools ensures that the best mesh can be generated, whether structured, multiblock, unstructured, or hybrid. GAMBIT's range of CAD readers allow you to bring in any geometry, error free, into its meshing environment. GAMBIT also has an excellent boundary layer mesher for growing optimum grid cells off wall surfaces in the geometries for fluid flow simulation purposes.

2.5 Turbulence Modelling

A turbulence model is a computational procedure to close the system of mean flow equations so that a more or less wide variety of the problems can be calculated. For most engineering purposes, it is unnecessary to resolve the details of turbulence fluctuations. Only the effects of the turbulence the mean flow are usually sought. For a turbulence model to be useful in a general purpose CFD code it must have wide applicability, be accurate, simple and economical to run. The most common turbulence models are classified as below:

1. Spalart-Almaras Model
2. Types of the k-ε model (standard, RNG, realizable)
3. Types of the k-ω. model (standard, SST)
4. Reynolds Stress Models (RSM)
5. Large Eddy Simulation (LES)

Fluid flows in practice generally can be divided into three types, i.e. creep, laminar and turbulent flow, with turbulent flow being the most common, complex and unpredictable of the three. This is due to the characteristics of these types of flows, which are listed below:

1. The inertia of fluid particles are more dominant than the viscous effects of the
2. Flow, causing it to be unstable
3. Instantaneous ("snapshots") of turbulent flows are very unpredictable and 'chaotic' with a fluctuating velocity field, but there is hope to predict the mean
4. Values.
5. Almost all turbulent flows are three dimensional (3-D) flow as the fluctuations occurs rapidly in all three spatial dimensions

6. Occurrence of turbulent diffusion, which is the process of parcels of fluid with differing concentration of one or more of the fluid's transported quantities, mixes together as they are brought into contact
7. Mixing causes loss of mean potential or kinetic energy due to the action of viscosity, which is an irreversible process known as dissipation
8. The flows possess coherent structures, consist of repeatable and deterministic, often large scale events. However, these events occur randomly and differ each time, thus making it very unpredictable.

Thus numerical methods of solving these flows are essential and play a very important role in the design process. There are many approaches to solving turbulent flows, which include the use of correlations, the use of integral equations, Direct Numerical Solutions (DNS) and Large-Eddy-Simulation (LES), where the latter two attempt to produce a collection of "snapshots" of instantaneous flows, which can then be averaged similarly to experimental data. The one method that will be discuss at length is the Reynolds-Average Navier-Stokes (RANS) equation, which directly attempts to solve model equations for the mean quantities since this is the approach taken for this project.

Turbulent flows are characterized by fluctuating velocity fields. This causes the transported quantities such as momentum, energy, and species concentration to mix, and cause these quantities to fluctuate as well. Since the fluctuations can be of small scale and high frequency, they are too computationally expensive to simulate directly in practical engineering calculations. Instead, the fundamental equations can be time-averaged, 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. This is demonstrated below. The instantaneous velocity, U, consists of a mean velocity, U, and a fluctuating velocity, u (t), given by

$$U = \overline{U} + u(t) \qquad (2.1)$$

The Reynolds-averaged Navier-Stokes (RANS) equations are time-averaged. Equations of motion for fluid flow. They are primarily used while dealing with turbulent flows. These equations can be used with approximations based on knowledge of the properties of flow turbulence to give approximate averaged solutions to the Navier-Stokes equations.

For incompressible flow of Newtonian fluid, these equations can be written as,

$$\rho\frac{\partial\overline{u_i}}{\partial t}+\rho\frac{\partial\overline{u_j u_i}}{\partial x_j}=\rho\overline{f_i}+\frac{\partial}{\partial x_j}\left[-\overline{p}\delta_{ij}+\mu\left(\frac{\partial\overline{u_i}}{\partial x_j}+\frac{\partial u_i}{\partial x_i}\right)-\rho\overline{u_i^{'}u_j^{'}}\right] \quad (2.2)$$

The left hand side of this equation represents the change in mean momentum of fluid element due to the unsteadiness in the mean flow and the convection by the mean flow. This change is balanced by the mean body force, the isotropic stress due to the mean pressure field, the viscous stresses, and apparent stress $\rho\overline{u_i^{'}u_j^{'}}$ due to the fluctuating velocity field, generally referred to as Reynolds stresses. The presence of these terms causes the equation to be unsolvable since there are too many unknowns. Thus it is required to model these stresses, and this is where the turbulence models come in. The Reynolds stresses are modeled using a model termed the eddy-viscosity model, given by:

$$-\rho\overline{u_i u_j}=\mu_t\left(\frac{\partial\overline{u_i}}{\partial x_j}+\frac{\partial\overline{u_j}}{\partial x_i}\right)-\frac{2}{3}\rho\delta_y k \quad (2.3)$$

μ_t is known as the turbulence viscosity, which can be characterized by two parameters, the turbulent kinetic energy, k, and a length scale, L. Thus, by modeling an equation for the kinetic energy, k, and the length scale, L, can be solved, and having done that, the RANS equation can also be solved.

2.5.1 The k-ε model

Since the realizable k-ε model and k-ω model will be used in this project, the basic equations for both models will be described in detail:

The k-ε model is the most commonly used of all the turbulence models. It is classified as a two equation model. This denotes the fact that the transport equation is solved for two turbulent quantities k and ε. Within the model the properties k and ε are defined through equation below:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \tag{2.4}$$

C_μ represents the dimensionless parameter or a constant, k is the turbulent kinetic energy, and ε, is the turbulent dissipation. The turbulent kinetic energy is modelled by the transport equation given below:

$$\frac{\partial(\rho k)}{\partial t} + \partial\frac{(\rho \overline{u_j} k)}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\mu\frac{\partial k}{\partial x_j}\right) - \frac{\partial}{\partial x_j}\left(\frac{\rho}{2}\overline{u_j u_i u_i} + \overline{p u_j}\right) - \rho\overline{u_i u_j}\frac{\partial \overline{u_i}}{\partial x_k}\frac{\partial \overline{u_i}}{\partial x_k} \tag{2.5}$$

The second term on the left hand side of the equation and the first term on the right can be solved easily. The second term on the right hand side is known as turbulent diffusion and another model is needed to solve this term. The third term on the right hand side represents the rate of production of kinetic energy by the mean flow, P_k, while the last term represents the rate of dissipation of kinetic energy into internal energy, ε. These two terms also needs to be modelled, with ε needing its own transport equation. This is given by the equation below:

$$\frac{\partial(\rho\varepsilon)}{\partial t} + \frac{\partial(\rho \overline{u_j}\varepsilon)}{\partial x_j} = C_{\varepsilon 1} P_k \frac{\varepsilon}{k} - \rho C_{\varepsilon 2}\frac{\varepsilon^2}{k} + \frac{\partial}{\partial x_j}\left(\frac{\mu_t}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial x_j}\right) \tag{2.6}$$

where $C_{\varepsilon 1}$, $C_{\varepsilon 2}$, and σ_ε, are again dimensionless parameters and constants. These dimensionless parameters are set depending the performance of the model solving known simple test cases, and can be changed whenever necessary.

2.5.2 The k-ω model

For the k-ω model, the turbulent viscosity is defined as

$$\mu_t = \rho \frac{k^2}{\omega} \tag{2.7}$$

with the k equation being slightly modified, such that

$$\frac{\partial(\rho k)}{\partial t} + \partial \frac{(\rho \overline{u_j} k)}{\partial x_j} = P_k - \rho \beta^* k \omega + \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k^*}\right)\frac{\partial \omega}{\partial x_j}\right] \tag{2.8}$$

which contains the production, diffusion and dissipation term as before. The equation given by:

$$\frac{\partial(\rho \omega)}{\partial t} + \partial \frac{(\rho \overline{u_j} \omega)}{\partial x_j} = \alpha \frac{\omega}{k} P_k - \rho \beta \omega^2 + \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k^*}\right)\frac{\partial \omega}{\partial x_j}\right] \tag{2.9}$$

where $\alpha, \beta, \beta^*, \alpha_k^*, \alpha_\varpi^*$ which can be changed whenever necessary. From practice, it can be seen that each of these models give better results at different flow conditions, thus it is important to choose the best model depending on the flow problem that is needs to be solved.

2.5.3 Near Wall Treatment Condition

An important issue in turbulence modeling is the numerical treatment of the equations in regions close to the walls. When there is a wall, the turbulence becomes more complex because of the no slip condition at the wall where the flow is reduced to laminar flow or the molecular viscosity dominates. The near wall formulation determines the accuracy of the wall shear stress and heat transfer predictions.

The walls affect the turbulent flow by:

1. No-slip condition at the wall which is v (wall) =0.
2. Viscous damping and kinematics blocking near the wall reduce velocity fluctuations.
3. Large gradients in temperature and velocity field occur near the wall because walls are the main source of turbulence.
4. The near wall treatment determines how the solver analyses this near wall region. In general there are 3 near wall treatments available, using

- standard wall functions
- non-linear wall functions

A wall function is a distribution function that describes the variation of *Ui*, *T*, u_iu_j, k, and ε between a wall and the turbulent zone near it. It is often used to bypass the necessity of detailed numerical treatment and the uncertain validity of a turbulence model.

2.5.4 Discretization Method

Discretization involves the substitution of a variety of finite-difference-type approximations for the terms in the integrated equations representing flow processes such as convection, diffusion and sources. This converts the integral equations into the system of algebraic equations. This process is usually carried out as a first step toward making them suitable for numerical evaluation and implementation on digital computers. In order to be processed on a digital computer another process named quantization is essential.

The stability of the chosen discretization is generally established numerically rather than analytically as with simple linear problems. Special care must also be taken to ensure that the discretization handles discontinuous solutions gracefully. The Euler equations and Navier-Stokes equations both admit shocks, and contact surfaces.

Some of the discretization methods being used are:

- Finite volume method.

  This is the "classical" or standard approach used most often in commercial software and research codes. The governing equations are solved on discrete control volumes. This integral approach yields a method that is inherently conservative (i.e., quantities such as density remain physically meaningful):

$$\frac{\partial}{\partial t}\iiint Q dV + \iint F dA = 0 \qquad (2.10)$$

  Where Q is the vector of conserved variables, F is the vector of fluxes (see Euler equations or Navier-Stokes equations), V is the cell volume, and A is the cell surface area.

- Finite element method.

  This method is popular for structural analysis of solids, but is also applicable to fluids. The FEM formulation requires, however, special care to ensure a conservative solution. The FEM formulation has been adapted for use with the Navier-Stokes equations. In this method, a weighted residual equation is formed:

$$R_i = \iiint W_i Q dV^e \tag{2.11}$$

where R_i is the equation residual at an element vertex i , Q is the conservation equation expressed on an element basis, W_i is the weight factor and V^e is the volume of the element.

- Finite difference method.

  This method has historical importance and is simple to program. It is currently only used in few specialized codes. Modern finite difference codes make use of an embedded boundary for handling complex geometries making these codes highly efficient and accurate. Other ways to handle geometries are using overlapping-grids, where the solution is interpolated across each grid.

$$\frac{\partial Q}{\partial t} + \frac{\partial F}{\partial x} + \frac{\partial G}{\partial y} + \frac{\partial H}{\partial z} = 0 \tag{2.12}$$

  Where Q is the vector of conserved variables, and F, G, and H are the fluxes in the x, y, and z directions respectively.

- Boundary element method.

  The boundary occupied by the fluid is divided into surface mesh.

- High-resolution schemes are used where shocks or discontinuities are present. To capture sharp changes in the solution requires the use of second or higher order numerical schemes that do not introduce spurious oscillations. This usually necessitates the application of flux limiters to ensure that the solution is total variation diminishing.

2.5.5 Navier-Stokes Equation

The Navier-Stokes equations, named after Claude-Louis Navier and George Gabriel Stokes, are a set of equations that describe the motion of fluid substances such as liquids and gases. These equations establish that changes in momentum in infinitesimal volumes of fluid are simply the product of changes in pressure and dissipative viscous forces (similar to friction) acting inside the fluid. These viscous forces originate in molecular interactions and dictate how viscous a fluid is. Thus, the Navier-Stokes equations are a dynamical statement of the balance of forces acting at any given region of the fluid.

The Navier-Stokes equations are differential equations which describe the motion of a fluid. These equations, unlike algebraic equations, do not seek to establish a relation among the variables of interest (e.g. velocity and pressure), rather they establish relations among the rates of change or fluxes of these quantities. In mathematical terms these rates correspond to their derivatives. Thus, the Navier-Stokes equations for the most simple case of an ideal fluid with zero viscosity states that acceleration (the rate of change of velocity) is proportional to the derivative of internal pressure. The Navier-Stokes equations are derived from conservation principles of:

- Mass
- Energy
- Momentum
- Angular momentum

The complete Navier-Stokes equations are :

$$\frac{\partial(\rho u)}{\partial t}+div(\rho uU)=-\frac{\partial p}{\partial x}+div(\mu\, grad\, u)+S_{M_x} \tag{2.13}$$

$$\frac{\partial(\rho w)}{\partial t}+div(\rho wU)=-\frac{\partial p}{\partial z}+div(\mu\, grad\, w)+S_{M_z} \tag{2.14}$$

$$\frac{\partial(\rho v)}{\partial t}+div(\rho vU)=-\frac{\partial p}{\partial y}+div(\mu\, grad\, v)+S_{M_y} \tag{2.15}$$

$$\frac{\partial(\rho i)}{\partial t}+div(\rho iU)=-p\; div\,U+div(k\, grad\, T)+\Phi+S_i \tag{2.16}$$

$$\frac{\partial(\rho m_j)}{\partial t}+div(\rho m_j \mathrm{u})=div(\Gamma_j\, grad\, m_j)+S_j \tag{2.17}$$

2.5.6 Conservation of Mass

Conservation of mass is written:

$$\begin{aligned}&\frac{\partial \rho}{\partial t}+\nabla.(\rho v)=0\\&\frac{\partial \rho}{\partial t}+\rho\nabla.v+v.\nabla\rho=0\\&\frac{D\rho}{Dt}+\rho\nabla.v=0\end{aligned} \tag{2.18}$$

where ρ is the mass density (mass per unit volume), and v is the velocity of the fluid.

In the case of an incompressible fluid, ρ does not vary along a path line and the equation reduces to $\nabla.v = 0$

2.5.7 Conservation of Energy

The first law of thermodynamics asserts simply that energy is conserved plus perhaps that heat is included as a form of energy. A commonly-used corollary of the first law is:

$$dE = TdS - PdV \tag{2.19}$$

2.5.8 Conservation of Momentum

Conservation of momentum is expressed in a manner similar to the continuity equation, with vector components of the momentum replacing density, and with a source term to represent forces acting on the fluid. We replace ρ in the continuity equation with the net momentum per unit volume along a particular direction, ρv_i, where v_i is the i^{th} component of the velocity, i.e. the velocity along the x, y, or z direction.

$$\frac{\partial}{\partial t}(\rho v_i) + \nabla.(\rho v_i v) = \rho f_i \tag{2.20}$$

ρf_i is the i^{th} component of the force acting on the fluid (actually the force per unit volume). Common forces encountered include gravity and pressure gradients. This can also be expressed as:

$$\frac{\partial}{\partial t}(\rho v) + \nabla.(\rho v \otimes v) = \rho f \tag{2.21}$$

$v \otimes v$ is a tensor, the $\otimes$ representing the tensor product.

Simplifying it further using the continuity equation, this becomes:

$$\rho \frac{Dv_i}{Dt} = \rho f_i \tag{2.22}$$

which is often written as

$$\rho \frac{Dv}{Dt} = \rho f \tag{2.23}$$

In which recognize the usual F=ma.

2.5.9 Conservation of Angular Momentum

The time derivative of angular momentum is called torque:

$$\tau = \frac{dL}{dt} = r \times \frac{dp}{dt} = r \times F \tag{2.24}$$

So requiring the system to be "closed" here is mathematically equivalent to zero external torque acting on the system:

$$L_{system} = cons\tan t \leftrightarrow \sum \tau_{ext} = 0 \tag{2.25}$$

where τ_{ext} is any torque applied to the system of particles

2.6 The Asymmetric Plane Diffuser

A diffuser is the mechanical device that is designed to control the characteristics of a fluid at the entrance to a thermodynamic open system. Diffusers are used to slow the fluid's velocity and to enhance its mixing into the surrounding fluid. In contrast, a nozzle is often intended to increase the discharge velocity and to direct the flow in one particular direction.

Flow through nozzles and diffusers may or may not be assumed to be adiabatic. Frictional effects may sometimes be important, but usually they are neglected. However, the external work transfer is always assumed to be zero. It is also assumed that changes in thermal energy are significantly greater than changes in potential energy and therefore the latter can usually be neglected for the purpose of analysis.

Diffusers are very common in heating, ventilating, and air-conditioning systems. Diffusers are used on both all-air and air-water HVAC systems, as part of room air distribution subsystems, and serve several purposes:

- To deliver both conditioning and ventilating air
- Evenly distribute the flow of air, in the desired directions
- To enhance mixing of room air into the primary air being discharged
- Often to cause the air jet(s) to attach to a ceiling or other surface (Coanda effect)
- To create low-velocity air movement in the occupied portion of room
- Accomplish the above while producing the minimum amount of noise

When possible, dampers, extractors, and other flow control devices should not be placed near diffusers' inlets (necks), but instead not be used at all or be placed far upstream. They have been shown to dramatically increase noise production. For as-cataloged diffuser performance, a straight section of duct needs serve a diffuser. An elbow, or kinked flex duct, just before a diffuser often leads to poor air distribution and increased noise.

Diffusers may be round, rectangular, or linear slot diffusers (LSDs), for example. This last type takes the form of one or several long, narrow slots (hence the name), often semi-concealed in a fixed or suspended ceiling.

Occasionally, diffusers are used in reverse fashion, as air inlets or 'returns'. This is especially true for LSDs and 'perf' diffusers. But more commonly, grilles are used as return or exhaust air inlets. The first investigation of the flow characteristics in a plane asymmetric diffuser was done by Obi et al (1993). In the investigation the opening angle of the diffuser was 10°.

2.7 Test Case

The experimental test case presented in this thesis is a separated flow in an asymmetric plane diffuser. This flow is of considerable interest to turbulence modelers due to it is simple geometry and the availability of experimental data. The flow has several desirable features which make it good test case for validation. According to Kaltenbrach the most important features are:

- The flow experiences pressure-driven separation from a smooth wall. Many technical devices are designed to operate under these conditions.
- The flow exhibits rich flow physics, such as the combined effects of adverse pressure gradient and convex curvature near the diffuser inlet.
- The inlet duct has a length of more than 100 duct heights, thereby guaranteeing that the inlet flow is fully developed turbulent channel flow.

In such flow, determining the separation line depends on the correct modeling of the shear stress and normal stresses, while the reattachment point mainly depends on the magnitude of the shear stress in a free shear layer. The major challenges of turbulence models in such a flow are:

- To predict the correct time scale.
- To predict the correct anisotropy of the Reynolds stress tensor.

The geometry of the diffuser can be seen in figure 2.1. The geometry is divided into three regions:

- the inlet
- the diffusing section
- the outlet

H is the inlet channel height, and an overall expansion ratio of 4.7. The inlet plane is located at x/H = -5 and the exit plane is located at x/H = 75.The Reynolds number based on the bulk velocity U_b at the inlet section and H is $Re_b = 18000$. The corresponding Reynolds number based on the friction velocity is u_* is $Re_t = 500$

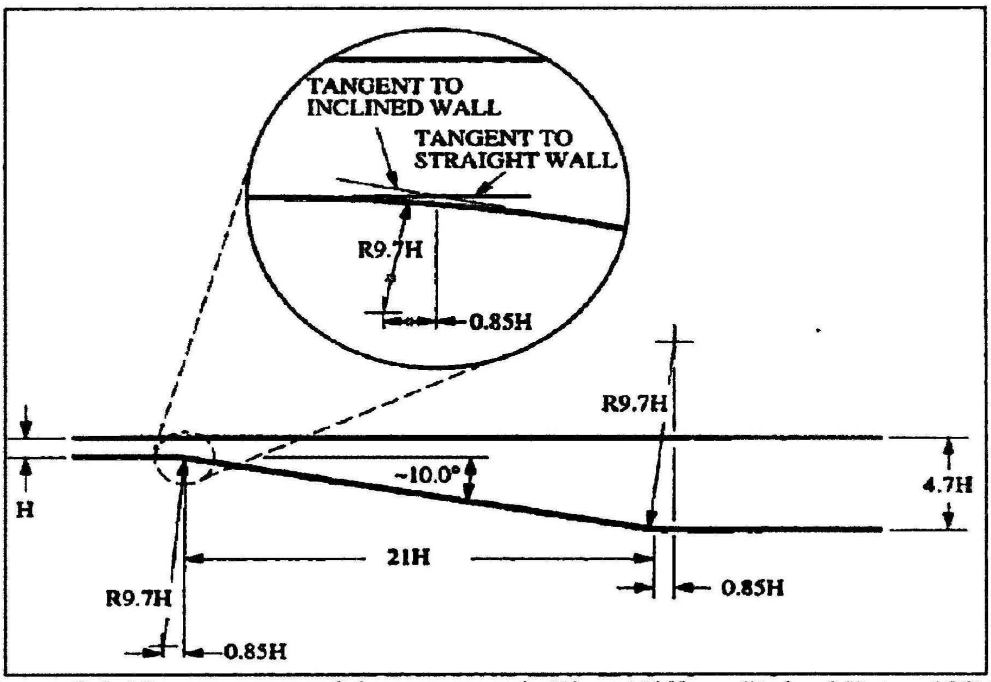


Figure 2.1 The geometry of the asymmetric Plane Diffuser(Buice&Eaton,1997)

The diffusing section has a length 21H where . the inlet channel height, and an overall expansion ratio of 4.7:1. The inclination is at 10° to the horizontal plane. The length of the expansion is 21H. Fillet radius of the inclined wall and the horizontal wall at both ends are 9.7H. The x and y direction are defined in the stream wise direction and in the direction normal to the horizontal walls respectively. The x-axis originates at the intersection of the tangent of both inclined and horizontal wall, as identified in Figure 2.1 and the y-axis originates from the bottom wall downstream of the channel.

2.8 Turbulence

Turbulence or turbulent flow is a flow regime characterized by chaotic, stochastic property changes. This includes low momentum diffusion, high momentum convection, and rapid variation of pressure and velocity in space and time. It is not an easy task to and a proper definition of turbulence. However, in 1937 Taylor and von Karman proposed the following definition:

"Turbulence is an irregular motion which is general makes it is appearance in fluids, gaseous or liquid, when they flow past solid or surface or even when neighboring streams of the same fluid flow past over one another"

Almost all fluid flow which we encounter in daily life is turbulent. Typical examples are flow around (as well as *in*) cars, aero planes and buildings. The boundary layers and the wakes around and after bluff bodies such as cars, aero planes and buildings are turbulent. Also the flow and combustion in engines, both in piston engines and gas turbines and combustors, are highly turbulent. Hence, when we compute fluid flow it will most likely be turbulent.

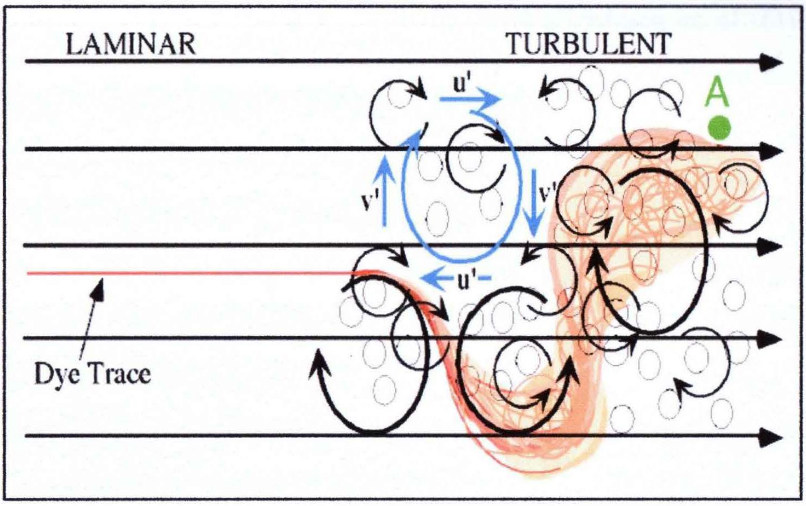


Figure 2.2 Tracer transport in laminar and turbulent flow

The Figure 2.2 shows tracer transport in laminar and turbulent flow. The straight, parallel black lines are streamlines, which are everywhere parallel to the mean flow. In laminar flow the fluid particles follow the streamlines exactly, as shown by the linear dye trace in the laminar region. In turbulent flow eddies of many sizes are superimposed onto the mean flow. When dye enters the turbulent region it traces a path dictated by both the mean flow (streamlines) and the eddies. Larger eddies carry the dye laterally across streamlines. Smaller eddies create smaller scale stirring that causes the dye filament to spread (diffuse). Turbulence is rotational and three-dimensional motion. It is highly dissipative and needs a source of energy to be maintained. It is also highly diffusive and rapid mixing significantly increases momentum, heat, and mass transfer.

2.9 Previous Study

Flows in plane asymmetric diffusers have previously been studied experimentally by Obi et al (1993) and Buice and Eaton (1997) and Buice and Eaton (2000). In all these studies the angle of the inclined wall was 10^o. Simulations and model prediction studies on the geometry with 10^o opening angle have been performed in a number of previous investigations. An extensive numerical study of the plane asymmetric diffuser flow was made by Kaltenbach et al. (1999), who perform LES (Large Eddy Simulations) at a Reynolds number, based on half inlet channel height and inlet channel friction velocity, of 1000. Their data showed good agreement with the experiment data by Buice and Eaton (2000) for mean velocity profile. For instance, Apsley and Leschziner (2000) tested both linear and non linear eddy viscosity models as well as differential stress-transport model. They found that strain dependent coefficients and anisotropy resolving closures are needed. However, no models tested were capable of resolving all flow features in the diffuser. Apsley and Leschziner (2000) also point out the possibility to encounter problems related to 'fapping' motion of the unsteady separation. In an ERCOFTAC workshop(Hellsten and Rautaheimo, 1999), different numerical approaches with varying turbulence models were tested and compared to the Buice and Eaton (2000) data-set. Models used comprised $k-\varepsilon, k-\omega$,RSM and LES. The agreement was, for the more simple models, in general fairly poor indicating that more complex models are needed to capture the flow physics. The plane asymmetric diffuser has also been used as a test case for commercial codes. The investigation performed by Iaccarino (2000) aimed at finding the limits of the versatile commercial codes in this complex flow. The codes tested were CFX, Fluent and Star-CD. Two turbulence models were tested in this 3 codes. The results was compared to Obi et tal (1993) and Buice and Eaton (2000) data sets. The $k-\varepsilon$ model was unable to capture the recirculation zone but the $v^2 - f$ model (Durbin 1995) did so with the 6% accuracy in separation length.

CHAPTER 3

METHODOLOGY

3.1 Introduction

In engineering field, commercial codes software definitely plays important parts in simulation process. For example, in this project, the type of software that was used is FLUENT. This software had been invented to ease the simulating of fluid flow, heat transfer and many more. In FLUENT software, we used GAMBIT . GAMBIT is Fluent's geometry and mesh generation software. GAMBIT has an excellent boundary layer mesher for growing optimum grid cells in the geometries.

3.2 Methodology Flow Chart

To achieve the objectives of the project, a methodology had been developed as shown in Figure 3.1. The methodology flow chart purposely used to give guideline and direction to make the project work successfully.

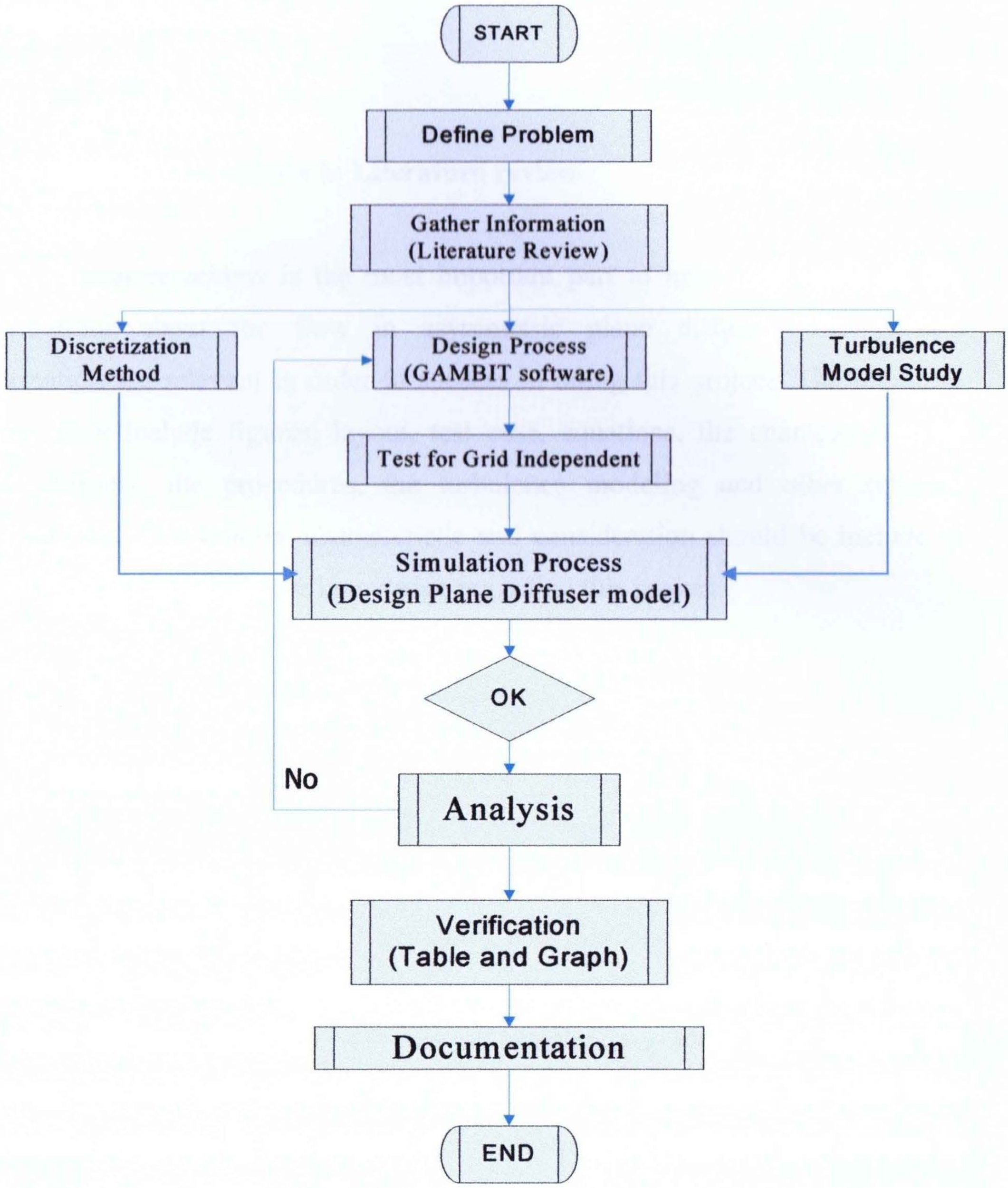


Figure 3.1 Project Flow Chart

3.3 Define Problem

The problem of this project already been defined in the problem statement at chapter 1.

3.4 Gather Information in Literature review

Literature review is the most important part to know and study all the informations about the flow in asymmetric plane diffuser. The gathers information are relevant in order to success in doing this project. The literature review may include figures, layout, test case, equations, the characteristics of plane diffuser, the procedures, the turbulence modeling and other relevant informations. This criteria, characteristic and consideration should be include in this section for gaining more knowledge regarding this project.

3.5 Design Process

In this project, the asymmetric plane diffuser will be design using GAMBIT. GAMBIT is Fluent's geometry and mesh generation software. This software is choose as it is possible for researchers to analyze the fluid flow along the plane diffuser. GAMBIT also has an excellent boundary layer mesher for growing optimum grid cells off wall surfaces in the geometries for fluid flow simulation purposes. In this project, the 2 dimensional functions was chosen in designing this plane diffuser. The geometry of the plane diffuser must be setup by using this software. In this stage, the geometry of the problem is described and a mesh is generated. This includes identifying the different regions of the geometry such as the inlet and outlet, describing the characteristics and dimensions of the geometry, and defining the mesh characteristics such as size, and the cell geometry being either a triangle or quadrilateral for 2-D mesh and being either a tetrahedron, hexahedron, prism or pyramid for 3-D mesh.

The important elements in meshing are:

- Cell = control volume into which domain is broken up computational domain is defined by mesh that represents the fluid and solid regions of interest.
- Face = boundary of a cell
- Edge = boundary of a face
- Node = grid point
- Zone = grouping of nodes, faces, and/or cells.
- Boundary data assigned to face zones. Material data and source terms assigned to cell zones.

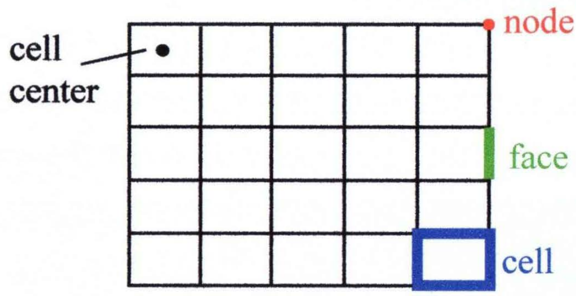


Figure 3.2 Simple 2-D mesh

3.6 Test for grid independence

Grid independency is tested by increasing and decreasing the number of grid cells. Grid-independency is achieved when the mesh used is fine enough, so that when the mesh is refined and the simulation is run on the new mesh, the results achieved are the same as the one before. The mesh that been used must be tested first. A fully converged solution is achieved when the convergence criterion is small enough, so that when the criterion is reduced and the simulation is run again, the results achieved would be the same as before. Once both of these factors are satisfied, then the results achieved can be acceptable. The standard mesh is of the size 300x100, with the first node is at a Y+ value of 1, which is well in the laminar sub-layer region. This means that the mesh will be suitable for all three models and the adapted near wall treatments for each model, used in the simulation.

3.7 Simulation Processes

There are a few steps that need to be done to run CFD analysis for turbulent flows, and these are the pre-processing, processing/solver and post processing procedures. Generally, the procedures are similar with any CFD analysis on any problem.

3.7.1 Pre-processing

Pre-processing consists of the input of a flow problem to CFD programs by means of an operator-friendly interface and the subsequent transformation of this input into a form suitable for use by the solver. The pre-processing stage involved :

- definition of the geometry of the region of interest such as : the computational domain
- grid generation – the sub division of the domain into a number of smaller, non overlapping sub-domains : a grid (or mesh) of cells (or control volumes or elements)
- selection of the physical and chemical phenomena that need to be modeled

- definition of fluid properties.
- specification of appropriate boundary conditions at cells which coincide with or touch the domain boundary

The solution to a flow problem (velocity, pressure, temperature etc) is defined at nodes inside each cell. The accuracy of a CFD solution is governed by the number of cells in the grid. In general, the larger the number of cells the better the solution accuracy. Both of the accuracy and solution and its cost in terms of necessary computer hardware and calculation time are dependent on the fineness of the grid. Optimal meshes are often non-uniform : finer in areas when large variations occur from point to point and coarser in regions with relatively little change. The mesh is applied to the geometry of the problem. Since the near wall region of the flow behaves differently with the outer flow, the mesh in this area needs to be of a certain way, depending on the near wall treatments adapted with the chosen model. The near wall treatment determines how the solver analyses this near wall region. In general there are 3 near wall treatments available, using standard wall functions, non-linear wall functions, and enhanced near wall treatments that had already discussed in chapter 2. After sorted out the mesh, there are different regions of geometry that need to be identified. There are the inlet, outlet or walls for the geometry.

3.7.2 Processing/Solver

The processing stage start after imported it to FLUENT. In this section, there are three numerical solution techniques which are finite difference, finite element and spectral methods. In outline the numerical methods that form the basis of the solver perform the following steps :

- Approximation of the unknown flow variables by means of simple functions.

- Discretisation by substitution of the approximations into the governing flow equations and subsequent mathematical manipulations.
- Solution of the algebraic equations.

In simple explanation, this process solves governing equations with the additional turbulent models, based on the inlet and boundary conditions across each control volume produced in the mesh. This is an iteration process that takes place until the result converges.

3.7.3 Post-processing

As in pre-processing a huge amount of development work has recently taken place in the post-processing field. Owing to the increased popularity of engineering workstations, many of which have outstanding graphics capabilities, the leading CFD packages are now equipped with versatile data visualizing tools. These include:

- Domain geometry and grid display
- Vector plots
- Line and shaded contour plots
- 2D and 3D surface plots
- Particle tracking
- View manipulation (translation, rotation, scaling, etc)
- Color postscript output

Post-Processing functions typically operate on surfaces. The surfaces are automatically created from zones or the additional surfaces can be created. These facilities may also include animation for dynamic result display and in addition to graphics all codes produce trusty alphanumeric output and have data export facilities for further manipulation external to the code. As in many order branches of CAE the graphics output capabilities of CFD codes have revolutionized the communication of ideas to the non specialist.

Post-processing is not only making sure the results are acceptable. The 'error free' results needs to be analyzed, as the whole reason of doing CFD analysis is to understand the behaviour of the flow. The software used, FLUENT, provides the user with graphical representation to visualize the flow in terms of the flow parameters. This includes the velocity profile, pressure distribution, and the stresses within the flow. Having available information on the parameters, then the results can be used to understand the flow behaviour in the flow region.

3.8 Analyze

After running the simulation, the results from the simulation will be compared with the experimental results to see the accuracy of the models that predict the solution of the flow. The convergence also must be checked after running the simulation. At convergence condition:

- All discrete conservation equations (momentum, energy, etc.) are obeyed in all cells to a specified tolerance.
- Solution no longer changes with more iterations.
- Overall mass, momentum, energy, and scalar balances are obtained.

Monitoring convergence with residuals:

- Generally, a decrease in residuals by 3 orders of magnitude indicates at least qualitative convergence.
- Major flow features established.
- Scaled energy residual must decrease to 10-6 for segregated solver.
- Scaled species residual may need to decrease to 10-5 to achieve species balance.
- Monitoring quantitative convergence:
- Monitor other variables for changes.

- Ensure that property conservation is satisfied.

Numerical instabilities can arise with an ill-posed problem, poor quality mesh, and/or inappropriate solver settings.

- Exhibited as increasing (diverging) or "stuck" residuals.
- Diverging residuals imply increasing imbalance in conservation equations.
- Unconverged results can be misleading

Troubleshooting:

- Ensure problem is well posed.
- Compute an initial solution with a first-order discretization scheme.
- Decrease under-relaxation for equations having convergence trouble (segregated).
- Reduce Courant number (coupled).
- Re-mesh or refine grid with high aspect ratio or highly skewed cells

3.9 Verification and documentation

The table and graph of comparison between the simulation and experimental results is produce in this section. After that, all the results need to be compiled and discussed.

3.10 Conclusion

FLUENT generated the graph for pressure, velocity profile, and can shows us the flow separation, reattachment and redevelopment of flow occurring within the diffuser. Afterwards, the comparison results between the simulation and the experimental data will be discuss. These results would be further being interpret in Chapter 4 – Results and Discussion.

CHAPTER 4

RESULTS AND DISCUSSONS

4.1 Introduction

This project majorly explains about the flow in asymmetric plane diffuser which is the most interesting test case in fluid flow investigating field. The literature review of this project has been made from the beginning and from various sources. Then, the simulation using Fluent software is being used to run this project. The software is use to defined the pressure distribution and velocity profiles occurs in the plane diffuser.

The converged simulation result which is in Figure 4.1 are analyzed and the flow of the diffuser had been examined by using the software. The results of the simulation will be explain in this chapter.

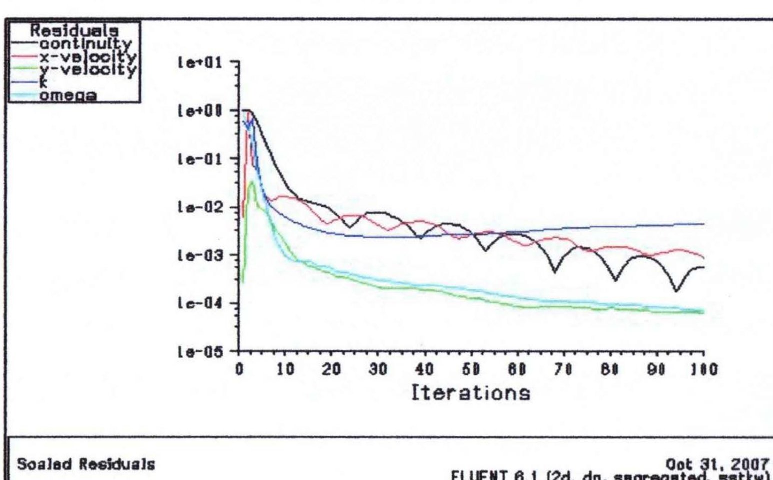


Figure 4.1 Residual convergence

4.2 Pressure Distribution

The plane diffuser shows an interesting flow behavior when the separation occur due to the adverse pressure distribution that occurs when the area of the channel increase. From the Figure 4.2, there is a reduction in pressure from the inlet of diffuser towards the outlet. The expansion ratio of the inlet and outlet of the diffuser is 1:4.7. Eventhough there is an expansion, the pressure in the region above the inclined wall shows the lowest value. This value was affected by the highest pressure value in the downstream region.

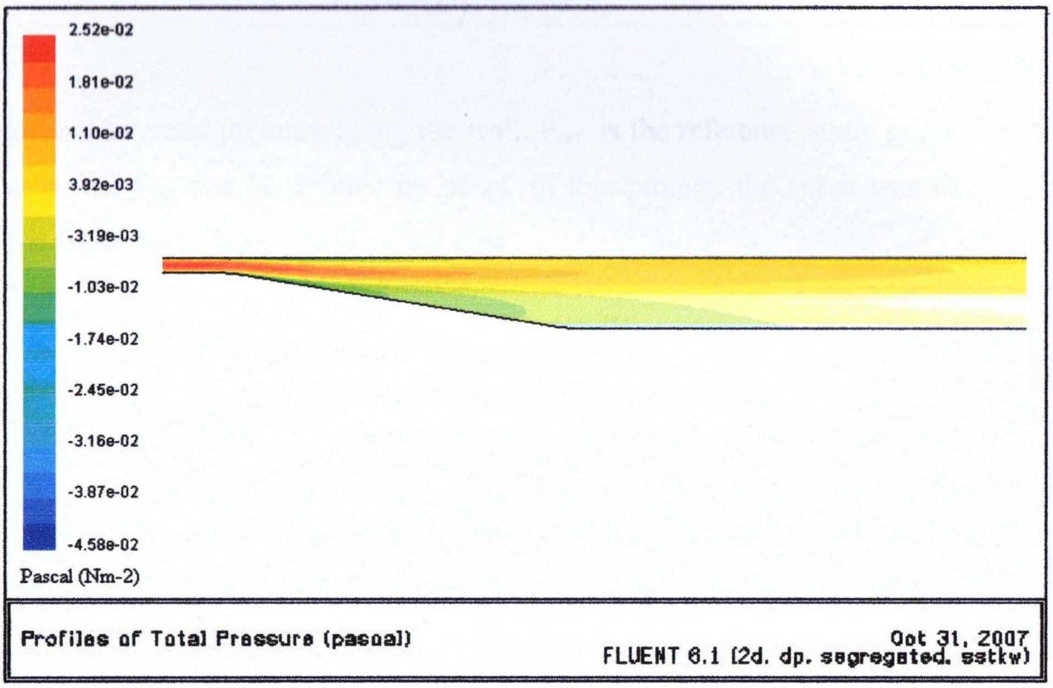


Figure 4.2 Contours Profiles of Total Pressure

The Figure 4.2 shows the total pressure distributions within the plane diffuser. The SST $k - \omega$ model was used in order to get the pressure distributions profile as figure above. The maximum and minimum value of pressure can be determined by referring to the value and the color on the left side of the figure. The pressure at the inlet of the diffuser gives the maximum value which is 0.0252 Pa and the value at the downstream region is around 0.003 Pa. The minimum pressure occurs in the region above the inclined wall. The value is about 0.02 Pa. The separation of flow occurs and produces an adverse pressure gradient.

In order to differentiate the pressure between the region above the inclined wall and the downstream region, the graph of the pressure coefficient, C_p was plotted along the bottom wall. The value of C_p was defined by using the formula below :

$$C_p = \frac{(P - P_{ref})}{\left(\frac{1}{2}\rho U_b^2\right)} = \frac{Static\ \Pr essure}{Dynamic\ \Pr essure} \tag{4.1}$$

P is defined as static pressure along the wall, P_{ref} is the reference static pressure. The value of P_{ref} can be defined by users. In this project, the value was taken from the Buice and Eaton experimental data.

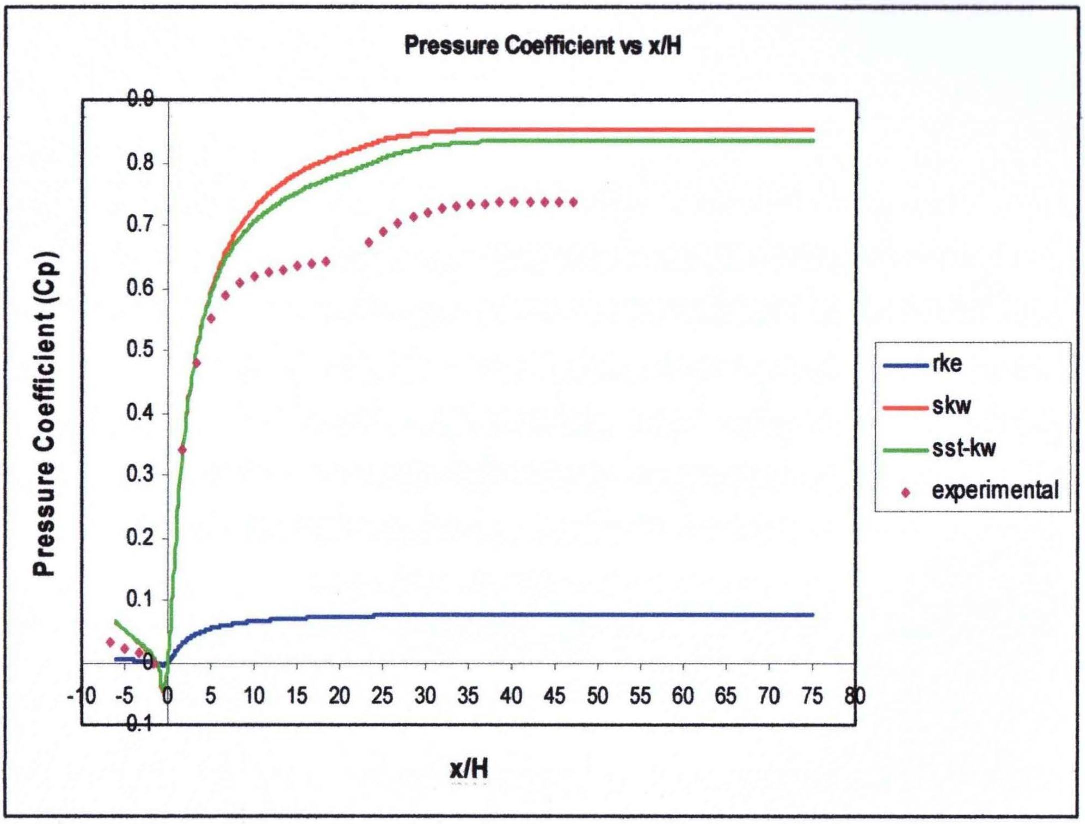


Figure 4.3 Graph Pressure Coefficient vs x/H

The Figure 4.3 shows the pressure coefficient along the bottom wall of the plane diffuser. The comparison between three turbulence model and the experimental data had been plotted above. It shows that, almost three of the models do not give very good results compared to experimental values. The realizable k-ε model shows the worst result. The standard k-ω model and the SST k-ω show accurate results but only in the beginning of the expansion. The value is over predict starts at x/H=5. From the observation, the SST k- ω model provides the most accurate result followed by standard k- ω model and lastly realizable k- ε model.

From the graph, the lowest pressure value is at the wall before the expansion. The pressure suddenly starts to increase above the inclined wall and at the downstream region, the pressure becomes consistent. The sudden pressure at the above inclined wall causes the wall separation. The realizable k- ε model maybe use to predict the most minimal pressure gradient value.

4.3 Velocity Profile

After evaluate the pressure distribution within the diffuser, the velocity profile was investigated by using the same software which is FLUENT. The velocity flow in the diffuser was evaluated by using that previous three turbulence models. After that, the value or results are compared with the experimental data by Buice & Eaton 1997. According to the size of the diffuser, it is inconvenient to predict the flow behavior along the diffuser. Thus, the values are scaled by a factor 5 in order to get accurate result and can provide more detail result.

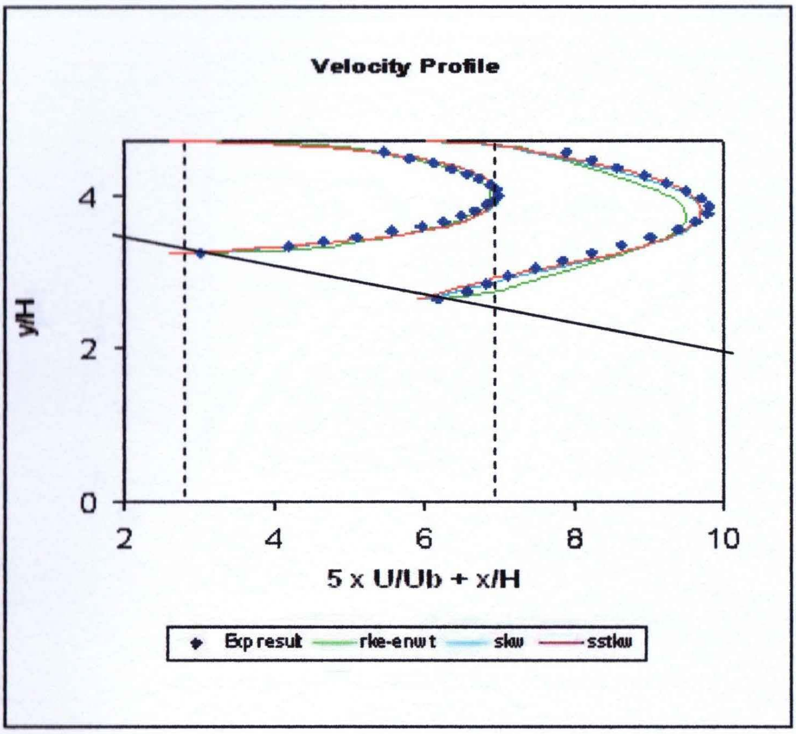


Figure 4.4 Velocity Profile at x/H=03 and x/H=06

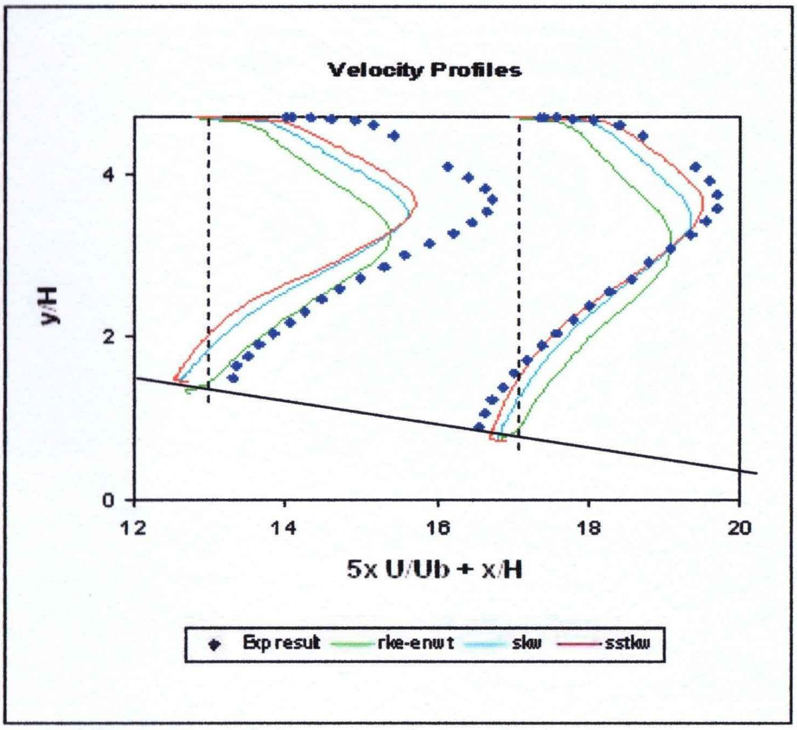


Figure 4.5 Velocity Profile at x/H=13 and x/H=17

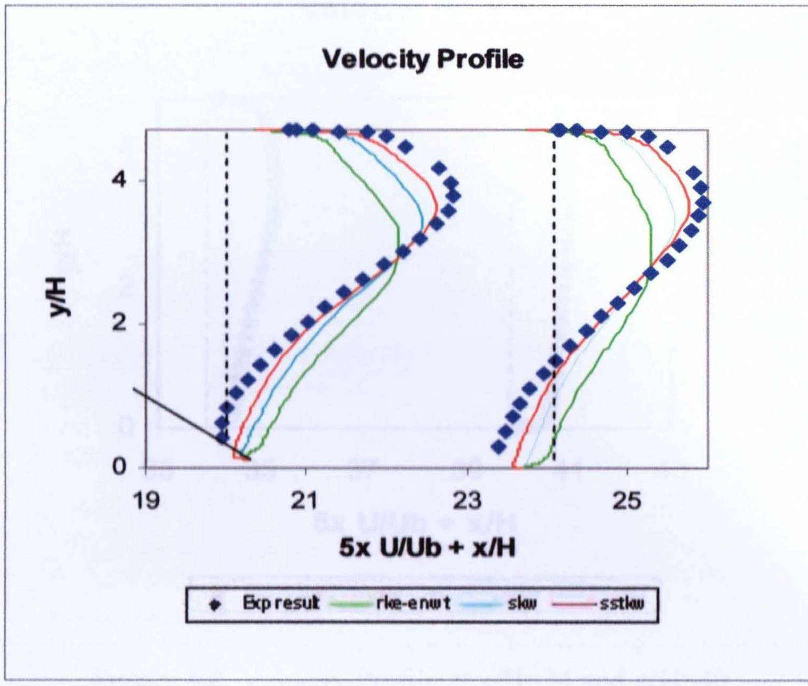


Figure 4.6 Velocity Profile at x/H=20 and x/H=24

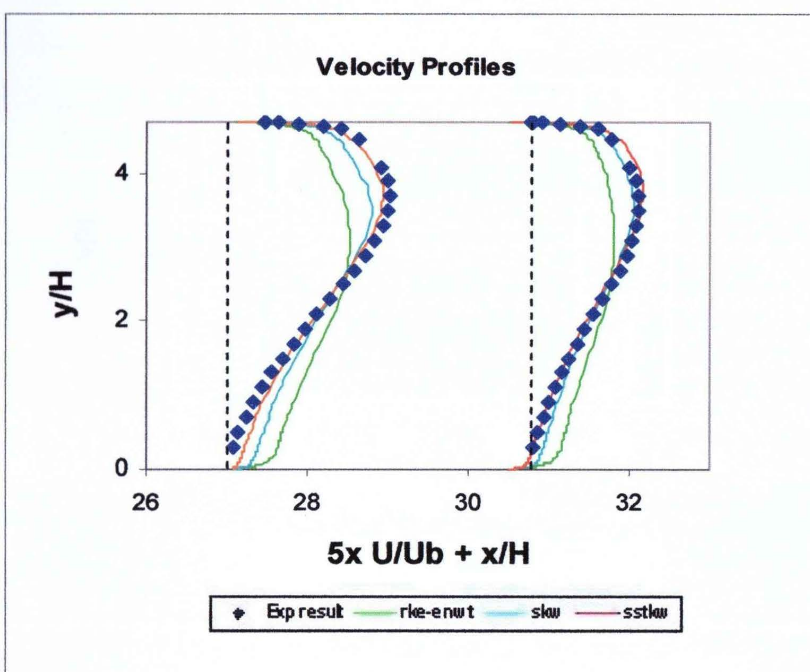


Figure 4.7 Velocity Profile at x/H=27 and x/H=30

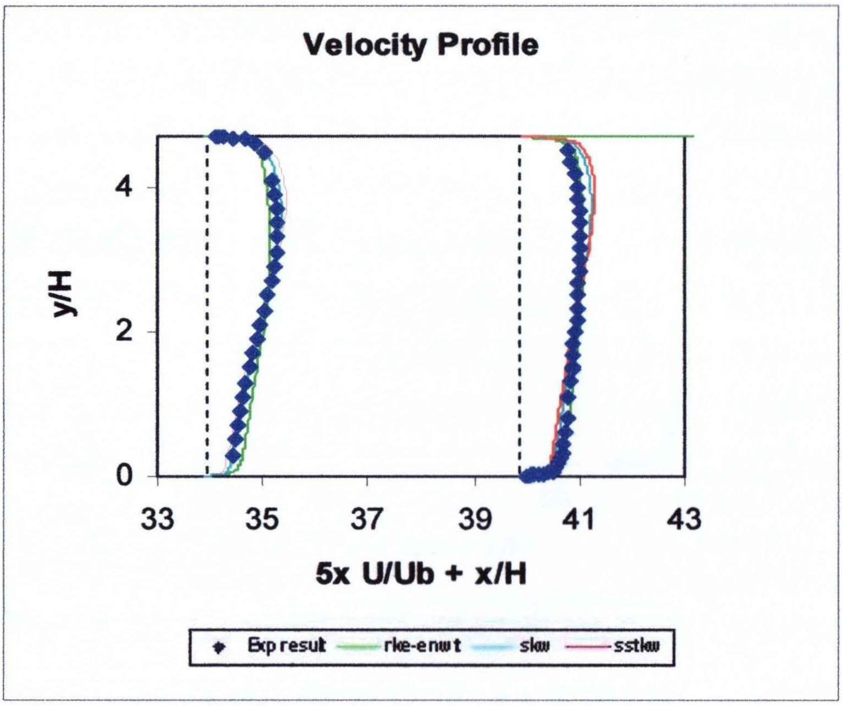


Figure 4.8 Velocity Profile at x/H=34 and x/H=40

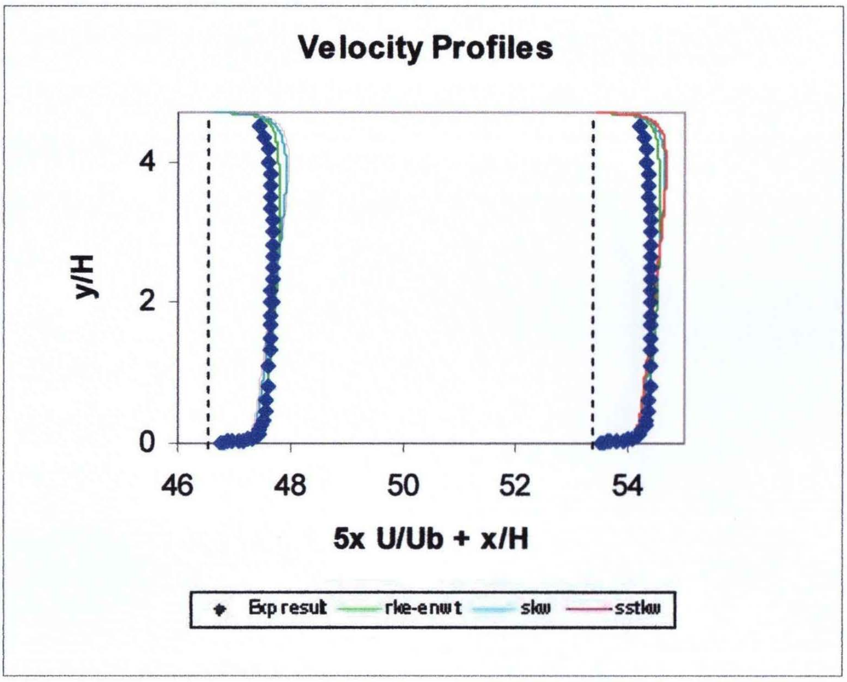


Figure 4.9 Velocity Profile at x/H=47 and x/H=53

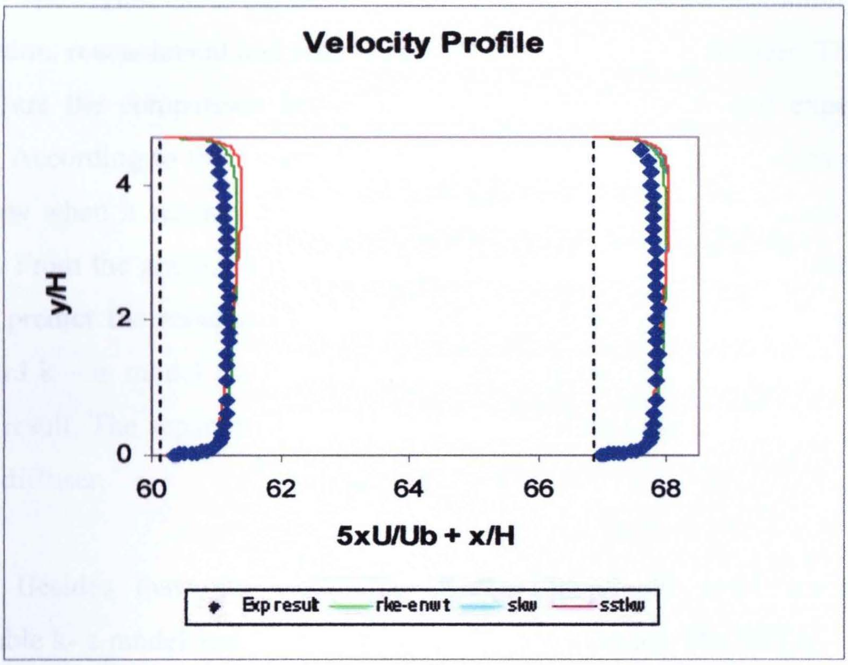


Figure 4.10 Velocity Profile at x/H=60 and x/H=67

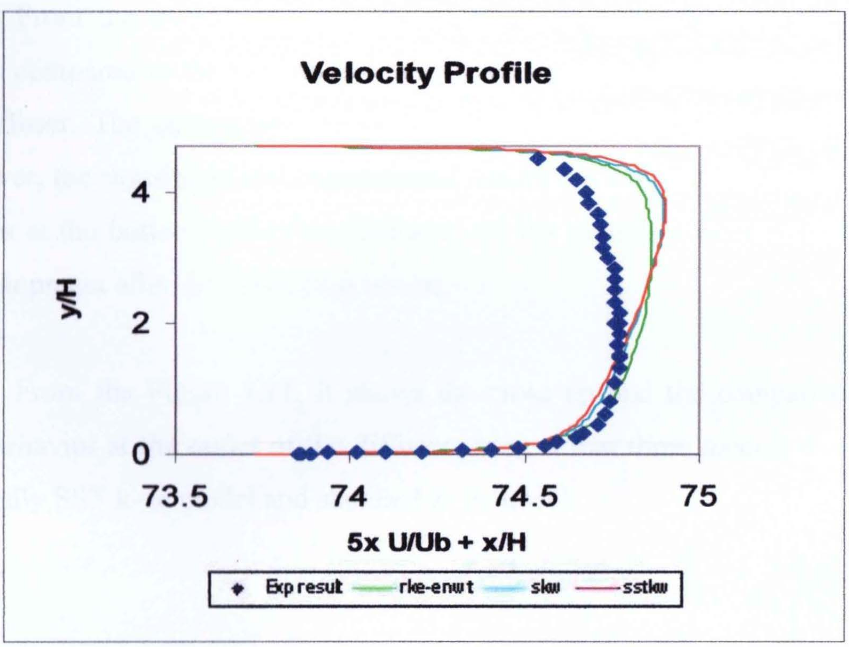


Figure 4.11 Velocity Profile at the outlet

The Figure 4.4 to Figure 4.11 shows the velocity profile including the flow separation, reattachment and redevelopment occur within the diffuser. The results above are the comparison between three turbulence models and experimental result. According to the Figure 4.5, the adverse pressure gradient starts affecting the flow when it reaches the inclined wall region. The flow separation occur at x=14 . From the graph, by comparing with experimental results, all these models under predict the separation region. The models that give good prediction are standard k – ω model and SST k-ω model. While the realizable k-ε provide the worst result. The separation region keep continues until the end of the expansion of the diffuser.

Besides that, the reattachment occur at around x=24 starting with realizable k- ε model and followed by standard k- ω model. The SST k – ω model reattach a bit late at x=27. The reattachment shows that the flow within the diffuser starts to redevelop.

From the overall observation, all the three models shows the different results compared to the experimental result starting from the beginning region in the diffuser. The comparison between the simulation still have large different. However, the simulation and experimental results are more similar when the flow reaches at the bottom half of the diffuser. All the models almost provide slower redevelopment after the separation occurs.

From the Figure 4.11, it shows the close up and the comparison of the flow behavior at the outlet of the diffuser. Almost that three models over predict especially SST k- ω model and standard k- ω model.

4.4 Position of Recirculation Region

Previously, overall investigation of velocity profile within the diffuser had been done but the profiles do not show the exactly part when the separation and reattachment occurs. Thus, the graph of Skin Friction Coefficient was plotted in order to know the exactly recirculation region occur. To plot the figures, the value of Skin Friction Coefficient C_f, need to be calculated by using this formula:

$$C_f = \frac{\tau_\omega}{\frac{1}{2}\rho U_b^2} \tag{4.2}$$

The τ_w is the value of wall shear stress:

$$\tau_\omega = \mu \left.\frac{dU}{dy}\right|_w \tag{4.3}$$

From the Figure 4.12, the recirculation region can be determined by examined the curve that crosses with the x axis by plotting the Skin Friction Coefficient values along the bottom wall. Then, the starting and the end points of that recirculation region can be detected. The experimental results and the three turbulence models provide different area of the recirculation region. Referring to the experimental plot, that region occurs at around x/H=8 and x/H=28. While the realizable $k-\varepsilon$ and standard k-ω models under predicts the recirculation region. The reattachment occurs at x=24. For the SST k- ω model over predicts the separation point and occur at x=4. It actually gives a better point of reattachment.

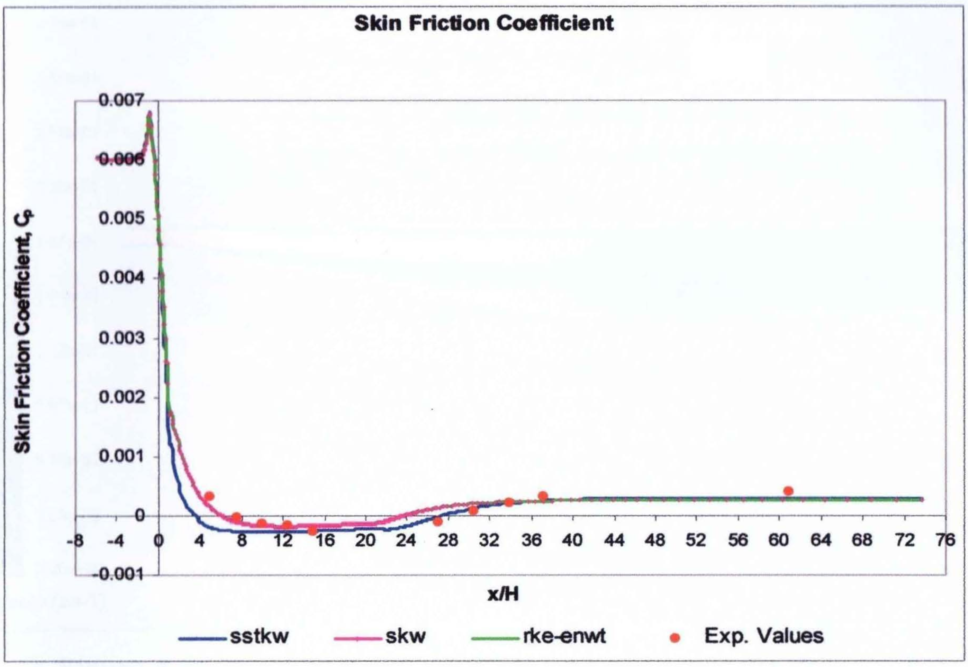


Figure 4.12 Graph Skin Fraction Coefficient vs x/H

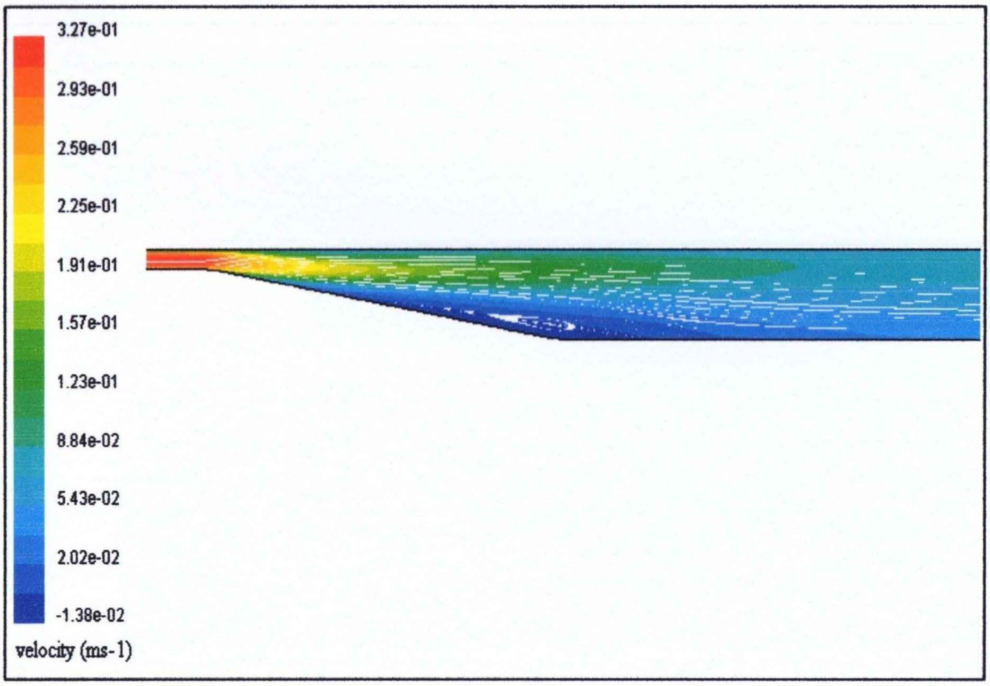


Figure 4.13 Recirculation region in the diffuser (SST k-ω model)

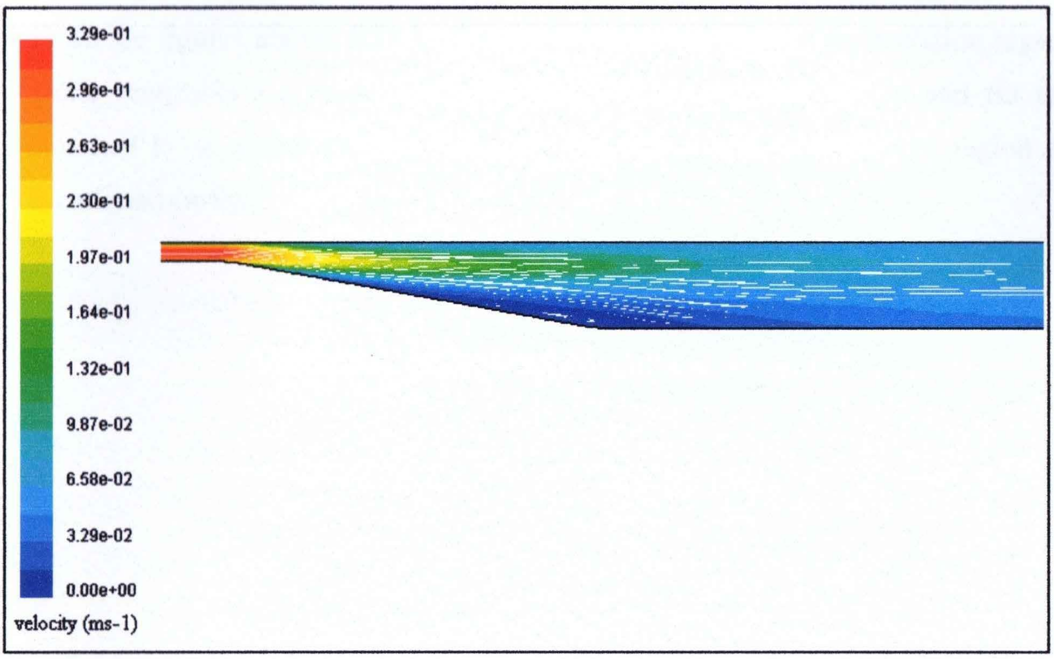


Figure 4.14 Recirculation region in the diffuser (Standard k-ω model)

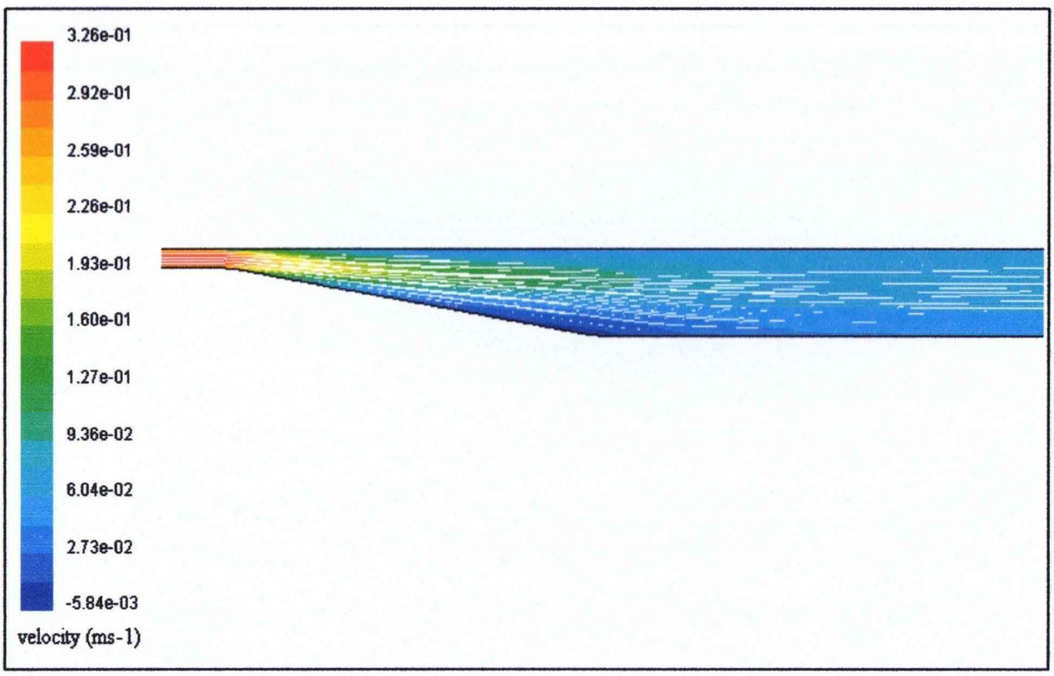


Figure 4.15 Recirculation region in the diffuser (Realizable k-ε model)

Figure 4.13 to Figure 4.15 are the visualations of the recirculation region from the three turbulence model. This visualation was simulated by using Fluent software. From the figures above, SST k-ω model provides the biggest recirculation region while realizable k-ε model gives the smallest recirculation region and for the standard k- ω model provide the same separation and reattachment region as mentioned earlier.

CHAPTER 5

CONCLUSION & RECOMMENDATIONS

5.1 Conclusion

The case flow in the asymmetric plane diffuser had been investigated and had provided the pressure distribution and velocity profile of the characteristics that occur in the flow. The comparison between experimental data and three turbulence models also been done. The mesh of the test case is 300 x 100 nodes. That was the standard mesh used in order to observe the flow behavior in the diffuser.

This project was run to compare all the models that been used which are realizable k- ε with enhanced near wall treatment model, the standard k-ω model and lastly, the SST k-ω model. The model that predicts the most similar value with the experimental data was SST k- ω model. In the earlier prediction, this model was over predicting the size of the recirculation region. While for the realizable k - ε models only predicts very small recirculation region

In conclusion, from the simulation, the SST k- ω model provides the most better result compared to the other two turbulence model. However, CFD commercial codes software need to improve from time to time in order to produce very accurate result.

5.2 Recommendations

CFD commercial codes is very useful in investigating fluid flow field for example in this project, FLUENT software had been used. The recommendations in improving this project are:

- Generate new turbulence model equations by researcher.
- CFD commercial codes software need to improve from time to time in order to produce very accurate result.
- All the results can be compared with the other software such as CFX or cosmos flow

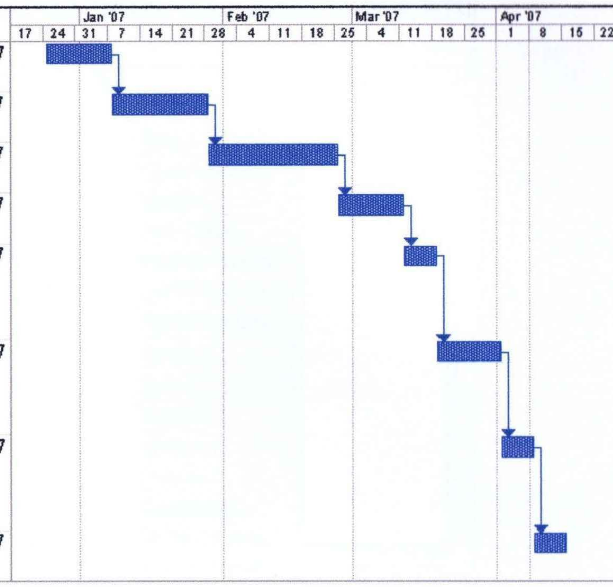
REFERENCES

1. Ching-Jen Chen, Sheng Yuh Jaw. *Fundamental of turbulence Modelling* 1998
2. Peter S.Bernard, James M Wallace *Turbulent Flow : Analysis, Measurement and Prediction* 2002.
3. Apsley D.D.,. Leschziner M.A *Advanced Turbulence Modelling of Separated Flow in a Diffuser.* Department of Civil & Structural Engineering, UMIST, Manchester M60 1QD, U.K. Department of Engineering, Queen Mary and Westfield College, University of London, London E1 4NS, U.K. .1999
4. John Finnemore E., Joseph Franzini B.. *Fluid Mechanics with Engineering Applications.* 10th edition. Mc Graw Hill. 2002
5. Gianluca Iaccarino. *Prediction of the turbulent flow in a diffuser with commercial CFD codes.* 2000
6. Versteeg H.K., Malalasekera W.. *An introduction to computational fluid dynamics The Finite Volume Method.* England : Pearson Education Limited 1995
7. Abdul Rashid M. Z.,. *Modelling and Simulation of an Asymmetric Plane Diffuser.* UMIST. Unpublished. 2003
8. Obai Younis Taha *Formulation,Implementation and Testing of* $k-\omega-v^2-f$ Chalmers University Of Technology,Sweden. 2004.
9. Olle Tornblom,Astrid Herbst & Arne V. Johansson. *Separation control in a plane asymmetric diffuser by means of streamwise votices – experiment, modeling and simulation.* Department of Mechanics,KTH.
10 Vance Dippold. *Buice-Eaton 2D Diffuser : Wind Wall Function Validation Study.* NASA Glenn Research Center.

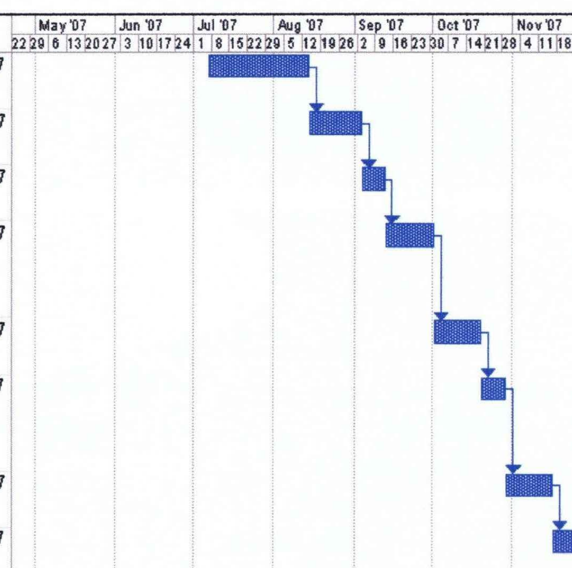
APPENDIX A

Gantt Chart for FYP 1 and FYP 2

No	PROJECT ACTIVITIES	Duration	Start	Finish
1	Proposal	14 days	Mon 12/25/06	Mon 1/8/07
2	Introduction	21 days	Mon 1/8/07	Mon 1/29/07
3	Literature Review	28 days	Mon 1/29/07	Mon 2/26/07
4	Methodology	14 days	Mon 2/26/07	Mon 3/12/07
5	Presentation(with supervisor)	7 days	Mon 3/12/07	Mon 3/19/07
6	Compile and repairing project report	14 days	Mon 3/19/07	Mon 4/2/07
7	Project & Log book submission	7 days	Mon 4/2/07	Mon 4/9/07
8	Presentation (PSM 1)	7 days	Mon 4/9/07	Mon 4/16/07

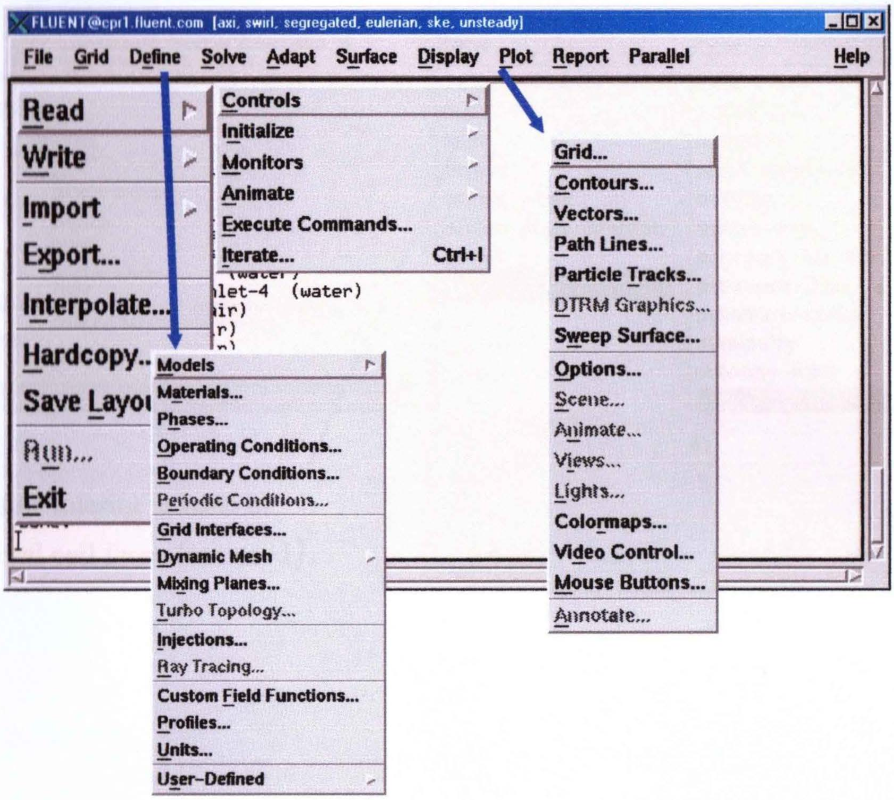


No	PROJECT ACTIVITIES	Duration	Start	Finish
1	Performing Fluent Simulation	28 days	Sat 7/7/07	Tue 8/14/07
2	Analyze result from simulation	14 days	Wed 8/15/07	Mon 9/3/07
3	Discussion	7 days	Tue 9/4/07	Wed 9/12/07
4	Compare with experimental data	14 days	Thu 9/13/07	Mon 10/1/07
5	Result and discussion	14 days	Tue 10/2/07	Fri 10/19/07
6	Draft Report	7 days	Sat 10/20/07	Mon 10/29/07
7	Pre-presentation & final presentation	14 days	Tue 10/30/07	Fri 11/16/07
8	Final Report corrections and submission	7 days	Sat 11/17/07	Mon 11/26/07



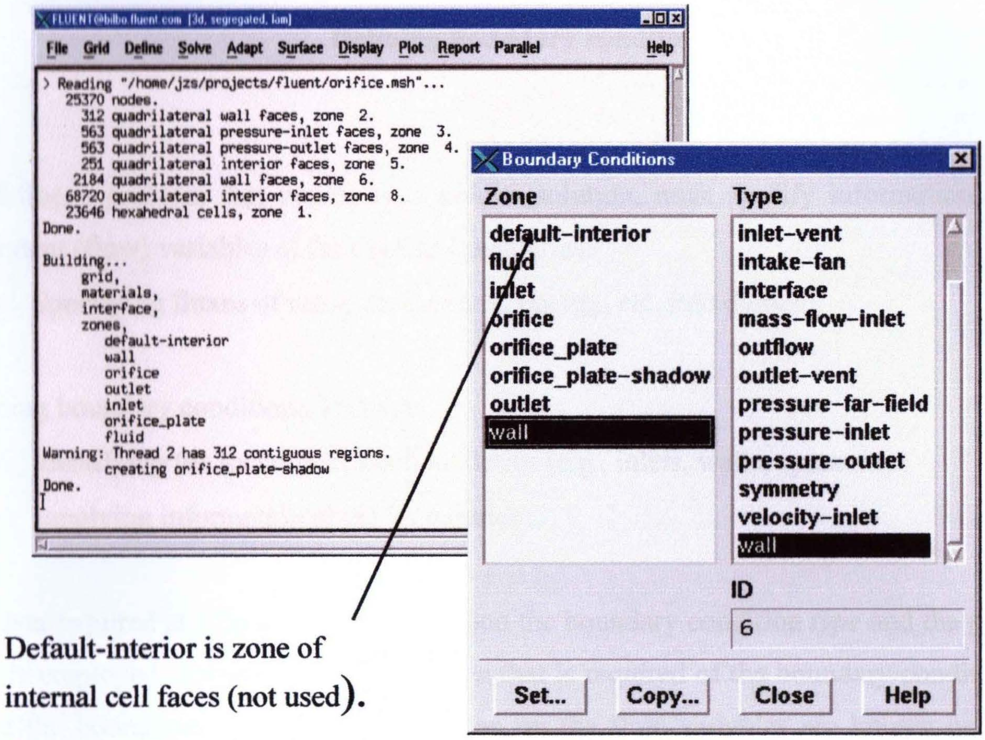
APPENDIX B

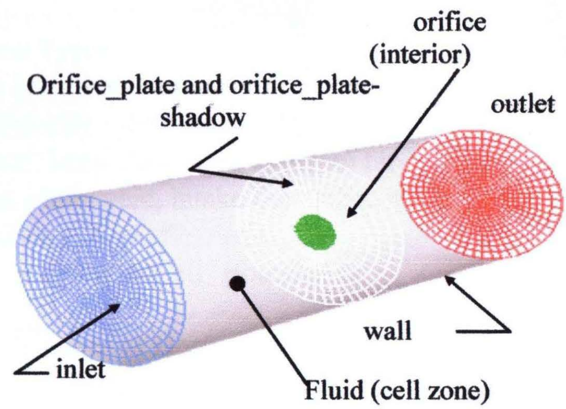
The Basic Guide in Using FLUENT



According to the above figure, menu is laid out such that order of operation that is generally left to right:

- Import and scale mesh file.
- Select physical models.
- Define material properties.
- Prescribe operating conditions.
- Prescribe boundary conditions.
- Provide an initial solution.
- Set solver controls.
- Set up convergence monitors.
- Compute and monitor solution.


FLUENT@bilbo.fluent.com [3d, segregated, lam]
File Grid Define Solve Adapt Surface Display Plot Report Parallel Help
> Reading "/home/jzs/projects/fluent/orifice.msh"...
25370 nodes.
312 quadrilateral wall faces, zone 2.
563 quadrilateral pressure-inlet faces, zone 3.
563 quadrilateral pressure-outlet faces, zone 4.
251 quadrilateral interior faces, zone 5.
2184 quadrilateral wall faces, zone 6.
68720 quadrilateral interior faces, zone 8.
23646 hexahedral cells, zone 1.
Done.
Building...
grid,
materials,
interface,
zones,
default-interior
wall
orifice
outlet
inlet
orifice_plate
fluid
Warning: Thread 2 has 312 contiguous regions.
creating orifice_plate-shadow
Done.
Boundary Conditions
Zone
default-interior
fluid
inlet
orifice
orifice_plate
orifice_plate-shadow
outlet
wall
Type
inlet-vent
intake-fan
interface
mass-flow-inlet
outflow
outlet-vent
pressure-far-field
pressure-inlet
pressure-outlet
symmetry
velocity-inlet
wall
ID
6
Set...
Copy...
Close
Help
Default-interior is zone of internal cell faces (not used).


orifice
(interior)
Orifice_plate and orifice_plate-shadow
outlet
wall
inlet
Fluid (cell zone)

APPENDIX C

Defining boundary condition

To define a problem that results in a *unique* solution, must specify information on the dependent (flow) variables at the domain boundaries.

- Specifying fluxes of mass, momentum, energy, etc. into domain.

Defining boundary conditions involves:

- identifying the location of the boundaries (e.g., inlets, walls, symmetry)
- supplying information at the boundaries

The data required at a boundary depends upon the boundary condition *type* and the physical models employed. Aware of the information that is required of the boundary condition and locate the boundaries where the information on the flow variables are known or can be reasonably approximated. Poorly defined boundary conditions can have a significant impact on the solution.

- Boundary Condition Types of External Faces
- General: Pressure inlet, Pressure outlet
- Incompressible: Velocity inlet, Outflow
- Compressible flows: Mass flow inlet, Pressure far-field
- Special: Inlet vent, outlet vent, intake fan, exhaust fan
- Other: Wall, Symmetry, Periodic, Axis
- Boundary Condition Types of Cell 'Boundaries'
- Fluid and Solid
- Boundary Condition Types of Double-Sided Face 'Boundaries'
- Fan, Interior, Porous Jump, Radiator, walls

APPENDIX D

Defining Node Value

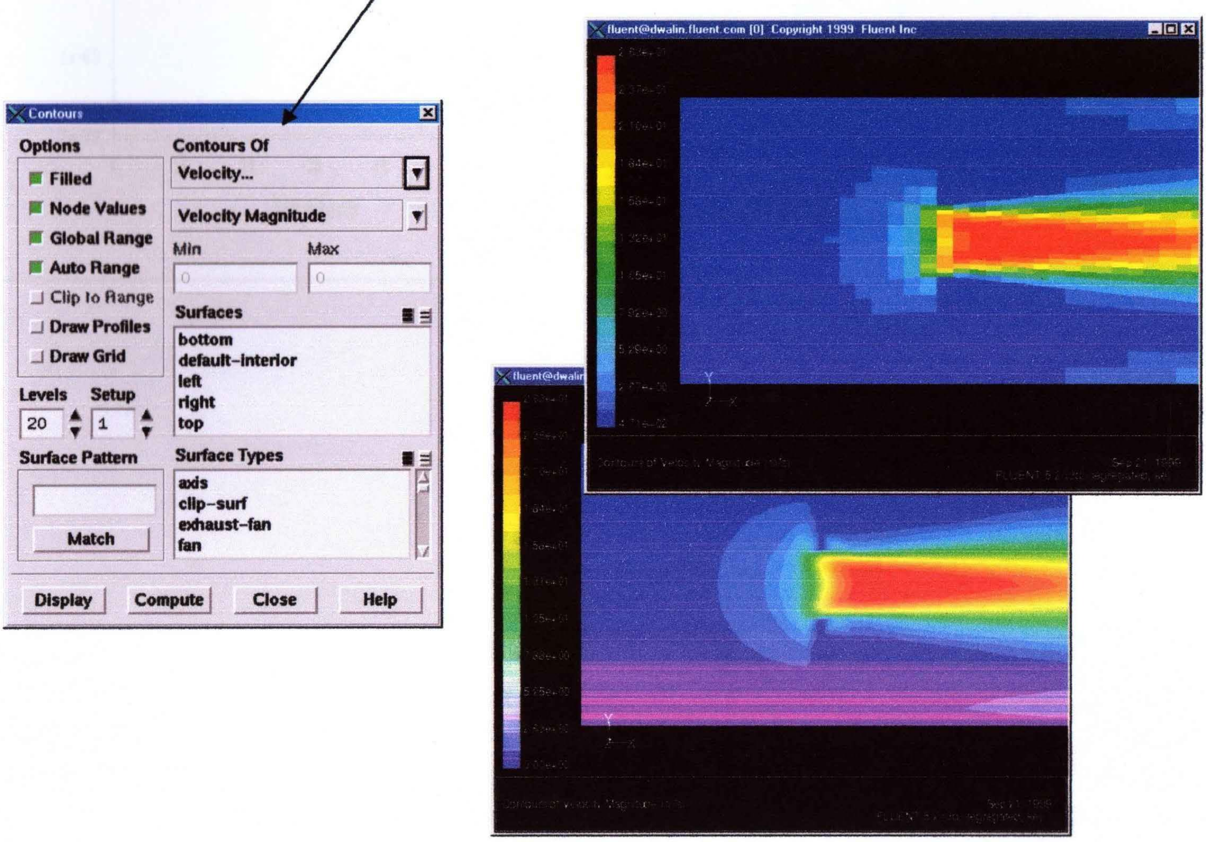
Fluent calculates field variable data at cell centers. Node values of the grid are either:

- calculated as the average of neighboring cell data, or,
- defined explicitly (when available) with boundary condition data.

Node values on surfaces are interpolated from grid node data.Data files store:

- data at cell centers
- node value data for primitive variables at boundary nodes

Enable Node Values to interpolate field data to nodes.



APPENDIX E

Convergence Monitor : Residuals