Application of computational fluid dynamics to the analysis of inlet port design in internal combustion engines

This item was submitted to Loughborough University's Institutional Repository by the/an author.

Additional Information:

- A Doctoral Thesis. Submitted in partial fulfillment of the requirements for the award of Doctor of Philosophy of Loughborough University.

Metadata Record: [https://dspace.lboro.ac.uk/2134/6730](https://dspace.lboro.ac.uk/2134/6730)

Publisher: © Anqi Chen

Please cite the published version.
This item is held in Loughborough University’s Institutional Repository (https://dspace.lboro.ac.uk/) and was harvested from the British Library’s EThOS service (http://www.ethos.bl.uk/). It is made available under the following Creative Commons Licence conditions.

For the full text of this licence, please go to: http://creativecommons.org/licenses/by-nc-nd/2.5/
APPLICATION OF
COMPUTATIONAL FLUID DYNAMICS
TO THE ANALYSIS OF INLET PORT DESIGN IN
INTERNAL COMBUSTION ENGINES

by

Anqi Chen

A Doctoral Thesis
Submitted in Partial Fulfilment of the Requirements
for the Award of
Doctor of Philosophy of Loughborough University of Technology

June, 1994

Supervisor: Professor J. C. Dent, Ph.D., C.Eng.

© by Anqi Chen
In memory of my grandmother
SYNOPSIS

The present research describes an investigation of the flow through the inlet port and the cylinder of an internal combustion engine. The principal aim of the work is to interpret the effects of the port shape and valve lift on the engine's "breathing" characteristics, and to develop a better understanding of flow and turbulence behaviour through the use of Computational Fluid Dynamics (CFD), using a commercial available package STAR-CD. A complex computational mesh model was constructed, which presents the actual inlet port/cylinder assembly, including a curved port, a cylinder, moving valve and piston. Predictions have been carried out for both steady and transient flows.

For steady flow, the influence of valve lift and port shape on discharge coefficient and the in-cylinder flow pattern has been examined. Surface static pressures predicted using the CFD code, providing a useful indicator of flow separation within the port/cylinder assembly, are presented and compared with experimental data. Details of velocity fields obtained by laser Doppler anemometry in a companion study at King's College London, using a steady flow bench test with a liquid working fluid for refractive index matching, compared favourably with the predicted data. For transient flow, the flow pattern changes and the turbulence field evolutions due to valve and piston movement are presented, and indicate the possible source of cyclic variability in an internal combustion engine.
ACKNOWLEDGEMENT

I wish to express my deepest gratitude to Prof. J. C. Dent, for his guidance and supervision, his continuous encouragement and support throughout this project.

I am also indebted to:

Mr. B. O. Mace for his help in building the experimental equipment,

Mr. K. W. Topley and Mr. V. Scothern for producing the photographs appearing in this thesis,

Dr. M. Yianneskis at King's College London for the companion study of LDA measurements and his continuous support,

Dr. R. Sanatian at Computational Dynamics Ltd. for his advice on using STAR-CD,

Mr. P. Holloway at SIA Ltd. for his particular support of computer network service.

Finally, I greatly appreciate the financial support from Ford Motor Company (UK) in the conduct of this work.
## NOMENCLATURE

<table>
<thead>
<tr>
<th>Symbols</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A_m$</td>
<td>Effect of convection and/or diffusion</td>
</tr>
<tr>
<td>$A_1$</td>
<td>Valve curtain area</td>
</tr>
<tr>
<td>$C_d$</td>
<td>Discharge coefficient</td>
</tr>
<tr>
<td>$C_v$</td>
<td>Specific heat at constant volume</td>
</tr>
<tr>
<td>$\overline{C_v}$</td>
<td>Mean specific heat at constant volume</td>
</tr>
<tr>
<td>$C_{vo}$</td>
<td>Reference specific heat at temperature $T_o$</td>
</tr>
<tr>
<td>$C_\mu$</td>
<td>Coefficient in the k-$\varepsilon$ turbulence model</td>
</tr>
<tr>
<td>$C_{\varepsilon_1}, C_{\varepsilon_2}$</td>
<td>Coefficient used in $\varepsilon$ - equation</td>
</tr>
<tr>
<td>$C_{\varepsilon_3}, C_{\varepsilon_4}$</td>
<td>Coefficient used in $\varepsilon$ - equation</td>
</tr>
<tr>
<td>$D$</td>
<td>Diameter of the entrance plane of inlet port</td>
</tr>
<tr>
<td>$D_p$</td>
<td>Geometrical coefficient</td>
</tr>
<tr>
<td>$D_v$</td>
<td>Valve diameter</td>
</tr>
<tr>
<td>$e'$</td>
<td>Fluctuating internal energy</td>
</tr>
<tr>
<td>$e$</td>
<td>Internal energy</td>
</tr>
<tr>
<td>$E$</td>
<td>External energy</td>
</tr>
<tr>
<td>$F_{x,j}$</td>
<td>Diffusional energy flux in direction $x_j$</td>
</tr>
<tr>
<td>$f_i$</td>
<td>Geometrical factors</td>
</tr>
<tr>
<td>$f_\mu, f_1, f_2$</td>
<td>Function in the k-\varepsilon group</td>
</tr>
<tr>
<td>$\sqrt{g}$</td>
<td>Determinant of metric tensor</td>
</tr>
<tr>
<td>$g_m$</td>
<td>Gravitational field components</td>
</tr>
<tr>
<td>$H_k$</td>
<td>Heat of formation of constituent $k$</td>
</tr>
<tr>
<td>$I$</td>
<td>Turbulence intensity</td>
</tr>
<tr>
<td>$k$</td>
<td>Turbulent kinetic energy</td>
</tr>
<tr>
<td>$\ell$</td>
<td>Turbulence length scale</td>
</tr>
<tr>
<td>$L$</td>
<td>Specified characteristic length of flow domain</td>
</tr>
</tbody>
</table>
\( L_v \quad \) Valve lift

\( \dot{m} \quad \) Mass flow rate

\( m_k \quad \) Mass fraction of mixture constituents

\( M_{\phi} \quad \) Normalisation factors

\( N \quad \) Iteration number, number of time steps

\( N_c \quad \) Number of cycle

\( P \quad \) Piezometric pressure

\( P_k \quad \) Production rate of turbulence energy

\( P_s \quad \) Static pressure

\( p' \quad \) Fluctuating pressure

\( \dot{q}_i \quad \) Diffusive source of energy

\( Q \quad \) External source of energy

\( R_{\phi} \quad \) Normalised coefficient

\( S_e \quad \) Energy source

\( S_{ij} \quad \) Rate of strain tensor

\( S_i \quad \) Momentum source component

\( \tilde{S}_m \quad \) Surface vectors normal to the cell faces

\( t \quad \) Time

\( T \quad \) Temperature

\( \bar{U}_j \quad \) Mean relative velocity between fluid and local coordination frame

\( U_i \quad \) Mean fluid velocity component in direction \( x_i \)

\( u'_i \quad \) Fluctuating velocity component in direction \( x_i \)

\( \bar{u}_j \quad \) Relative velocity between fluid and local coordinate frame

\( u_{ej} \quad \) Velocity at which local coordinate frame moves

\( u'_i u'_j \quad \) Reynolds stress tensor

\( u_w \quad \) Wall velocity
\( u^+ \) Relative non-dimensional velocity between
tangential fluid velocity and wall velocity

\( u,v,w \) Velocity component in x,y,z direction respectively

\( V \) Turbulent velocity scale

\( X \) Body force component in x direction

\( x_{ij} \) Cartesian coordinate (i, j =1,2,3)

\( x_m \) Coordinates from a datum

**Greek symbols**

\( \gamma \) Blending factor

\( \gamma \) Ratio of the specific heat at constant pressure and volume

\( \gamma_i \) Face-related weighting factors

\( \delta_t \) Time increment

\( \delta_{ij} \) kronecker delta

\( \rho \) Density

\( \rho_0 \) Reference density

\( \mu \) Fluid viscosity

\( \mu_t \) Turbulent viscosity

\( \varepsilon, \bar{\varepsilon} \) Turbulence dissipation rate

\( \phi \) Dependent variables

\( \phi' \) Fluctuation about the mean value of the variable \( \phi \)

\( \tau_{ij} \) Stress tensor components

\( \tau_w \) Wall shear stress

\( \tau_{xy} \) Shearing stress parallel to y direction

\( \tau_{xz} \) Shearing stress parallel to z direction

\( \sigma_k \) Coefficient in the k-\( \varepsilon \) turbulence model

\( \sigma_c \) Coefficient in the k-\( \varepsilon \) turbulence model
\(\sigma_x\) Normal stress in x direction
\(\sigma_\phi\) Molecular Prandtl/Schmidt number
\(\Gamma_T\) Diffusion coefficient for temperature \(T\)
\(\Gamma_\phi\) Diffusion coefficient for variable \(\phi\)
\(\Gamma_{\phi,i}\) Face diffusivity
\(\theta\) Crank angle
\(\kappa\) Empirical coefficient
\(E\) Empirical coefficient
\(\lambda\) Value of criterion for convergence
\(\omega\) Pseudovorticity

**Subscript**
- \(a\) Actual
- \(E\) East node of control volume
- \(e\) East face of control volume
- \(i\) Component corresponding to i direction, ideal
- \(ij\) Component corresponding to index i and j
- \(N\) North node of control volume
- \(n\) North face of control volume, "new" time level
- \(o\) "Old" time level
- \(p\) Central node of control volume
- \(S\) South node of control volume
- \(s\) South face of control volume
- \(W\) West node of control volume
- \(w\) West face of control volume, wall
# CONTENTS

<table>
<thead>
<tr>
<th>Abstract</th>
<th>i</th>
</tr>
</thead>
<tbody>
<tr>
<td>Acknowledgements</td>
<td>ii</td>
</tr>
<tr>
<td>Nomenclature</td>
<td>iii</td>
</tr>
</tbody>
</table>

## Chapter 1 Introduction

1.1 Nature and significance of the problem studies 1

1.2 General characteristics of intake flows 2

1.3 Objectives and present contribution 4

1.4 Outline of the thesis 5

## Chapter 2 Literature survey

2.1 Introduction 8

2.2 Experimental work 9

  2.2.1 Flow through inlet valves 9

  2.2.2 In-cylinder flow 13

  2.2.3 Improvements in port/valve design 15

    2.2.3.1 Effect of valve shape 15

    2.2.3.2 Effect of port shape 17

    2.2.3.3 Effect of valve lift 19

    2.2.3.4 Effect of pressure drop 20

2.3 Computational work 21

  2.3.1 Introduction 21

  2.3.2 Mathematical modelling 23

  2.3.3 The k-ε model 28

  2.3.4 Calculation of the flow in intake port/cylinder 33

    2.3.4.1 The computational mesh model 33
2.3.4.2 Prediction of intake flow 37
2.4 Engine design and the contribution of CFD 40

Chapter 3 Mathematical description and numerical approximation 54

3.1 Mathematical description 54
  3.1.1 Foundation 54
  3.1.2 The general differential equation 57
  3.1.3 Average equations 58
  3.1.4 Turbulence model 61
    3.1.4.1 The k-ε model 63
    3.1.4.2 Limitation of the k-ε model for predicting the intake flow and in-cylinder flow 66

3.2 Numerical approximation 73
  3.2.1 Introduction 73
  3.2.2 The finite-volume discretization 76
    3.2.2.1 Finite volume equations 76
    3.2.2.2 Finite differencing schemes 77
    3.2.2.3 Solution algorithm 80

Chapter 4 The STAR-CD CFD code 86
  4.1 Introduction 86
  4.2 Mathematical modelling 86
    4.2.1 Conservation equations 86
    4.2.2 Turbulence modelling 89
  4.3 Numerical solution procedure 92
    4.3.1 Approximation process 92
      4.3.1.1 General equations 92
4.3.1.2 Convection discretization 94
4.3.1.3 Final finite volume equation 96
4.3.2 Solution algorithm 97
  4.3.2.1 Finite-volume equation 98
  4.3.2.2 SIMPLE algorithm 99
  4.3.2.3 PISO algorithm 101
4.3.3 Algorithm selection 103
4.4 Boundary conditions 104
4.5 Completion tests 108
4.6 STAR-CD system structure and conclusion 109
  4.6.1 STAR-CD system structure 109
  4.6.2 Restrictions 110
  4.6.3 Conclusion 111

Chapter 5 Steady flow in the inlet port and the cylinder 116
  5.1 Introduction 116
  5.2 Computation of steady flows 116
    5.2.1 Flow configuration 116
    5.2.2 Boundary condition specification 118
    5.2.3 Flow field predictions 122
      5.2.3.1 Flow structure 122
      5.2.3.2 Discharge coefficient 130
  5.3 Comparisons with experiments 132
    5.3.1 Wall pressure measurement 132
      5.3.1.1 Introduction 132
      5.3.1.2 Comparisons between measured and predicted static pressures 133
    5.3.2 Flow visualization and LDA measurements 135
References 262
Appendix A Normalisation factors 278
Appendix B Calculation of mass flow rate through the test section 280
CHAPTER 1
INTRODUCTION

1.1 NATURE AND SIGNIFICANCE OF THE PROBLEM STUDIED

The gas flow through the inlet port and its magnitude and characteristics govern the subsequent charge motion and turbulence within the cylinder of an internal combustion engine, significantly control the combustion process and thereby influence the engine’s thermal efficiency and exhaust emissions.

A well designed intake port and valve assembly is required to induct the maximum charge mass into the cylinder, particularly at high engine speed, and is also required to provide a flow structure appropriate to the engine type, either Diesel or homogeneous charge.

A Diesel engine requires air swirl to achieve faster fuel-air mixing and burning rate for its combustion; the spark ignition engine combustion needs a sufficiently turbulent flow field to ensure rapid flame development and propagation. Air swirl is generated by a suitable design of the inlet port; the turbulent flow field is produced during the intake process and modified during the compression. Therefore, the geometry of the inlet port, valve and cylinder head plays a critical role in the engine’s performance.

Figure 1.1 shows the configuration of a typical inlet valve and port
(Barnes-Moss, 1975), in which, D is the valve inner seat diameter and all remaining proportions are relative to D. The inlet port is generally circular. The cross-sectional area should be designed to be no larger than that required to achieve the desired power output. Figure 1.2 shows the descriptions of some particular positions in the inlet port/valve assembly.

A high speed direct injection Diesel engine requires a high in-cylinder swirling air motion to achieve the desired fuel-air mixing rates, with the injected fuel sprays, leading to efficient combustion with a minimum of soot formation during combustion. In a high speed spark ignition engine, an adequate and cyclically repeatable turbulent flow field is required to minimise cycle to cycle variation in the combustion process. Excessive gas motion and turbulence results in loss of efficiency and an increase in unburned hydrocarbon emissions due to gas quenching.

The present investigation is of the characteristics of the intake flow motion and in-cylinder flow structure, governed by the inlet port shape and valve lift, which are expressed in terms of the discharge coefficient \( C_d \) and the velocity distribution in the annular opening between the valve head and seat under steady or transient condition.

### 1.2 GENERAL CHARACTERISTICS OF INTAKE FLOWS

It is generally acknowledged that the shear flow past the inlet valve is the major source of turbulence in the cylinder during induction. Early in the intake process, the flow field is generated by a conical jet which
emanates from the valve passage with an orientation determined by the valve seat angle at low lift. The air jet separates from the valve head and the valve seat, producing shear layers with large velocity gradients. This separation of the jet sets up recirculation regions beneath the valve head and in the corners between the cylinder wall and the cylinder head. When the jet reaches the cylinder wall, the wall deflects the major portion of the jet downwards toward the piston; a substantial fraction flows upward toward the cylinder head. The interaction of the intake jet with the wall produces large scale rotating flow patterns within the cylinder volume, which are strongly dependent on the inlet port geometry, the valve location and valve lift. During induction and compression processes, the flows become unstable, and break down into three-dimensional turbulent motions.

Swirl is created by bringing the intake flow into the cylinder with an initial angular momentum. When the air flow enters the cylinder tangentially towards the cylinder wall, the flow is deflected sideways and downwards in a swirl motion. This flow has a substantial net angular momentum about the cylinder axis, with non-uniform distribution around the circumference of the inlet valve. The swirl around the cylinder axis can also be generated by a masked or shrouded valve. Special port design such as a helical port generates the flow rotation about the valve axis before it enters the cylinder (Tindal et al, 1982). Swirl increases with increasing valve lift. Within the cylinder of an operating engine, the swirl velocities generated during the first half of the induction stroke are higher than the swirl generated during the latter half. The angular momentum of the air swirl within the cylinder at the end of induction will be less than the
entering angular momentum, and will be further reduced at the end of compression due to wall friction, provided there is no bowl in the piston or cylinder head, which will produce strong enhancement of the angular flow velocity because of momentum conservation as the air mass is displaced from the cylinder volume into reduced volume of the bowl. The change of the angular momentum with time satisfies the conservation of the moment of momentum. Neglecting the effects of friction, angular momentum of the swirling air is conserved. During compression, the swirl velocity increases as the moment of inertia of the air is decreased.

The flow through the inlet port/valve is responsible for many features of the air motion in the cylinder. The inlet port and its valve produce a three-dimensional, time-dependent turbulent flow having a highly complex structure such as turbulent shear layers, recirculation regions, and strong pressure gradients.

1.3 OBJECTIVES AND PRESENT CONTRIBUTION

The present research project is an investigation of the intake flow through the inlet port/valve and the cylinder. The research effort aims at interpreting the effects of the port shape and valve lift on the engine's breathing characteristics.

This work is carried out through the use of an existing three-dimensional computer code - STAR CD. A complex computational mesh model was constructed, including a curved port, its valve and the cylinder with/without the piston.
A major portion of the present contribution focuses on fully representing the complicated fluid mechanical phenomena of the flow pattern caused by variations of the inlet port shape on discharge coefficient, and the flow pattern. The method of surface pressure measurement which is interpreted in terms of isobaric diagrams (Hardenberg et al, 1975) was performed for comparisons with the prediction. In addition, detailed steady flow experimental data undertaken at King's College London, employing laser Doppler anemometry, was available to validate the computational results for different valve lifts.

1.4 OUTLINE OF THE THESIS

The remaining part of this thesis consists of six chapters. Chapter 2 reviews recent fundamental studies on engine port and in-cylinder flows, both experimental and theoretical. In chapter 3, a full description of the equations governing the motion of the fluid flow, the numerical procedure, and the solution algorithm are presented. There follows, in chapter 4, an introduction of the computation code - STAR CD. Chapter 5 then presents and discusses computed results of the steady flow and the influence of valve lift. Comparisons with surface pressure measurements and LDA experimental data are given. The effects of the curved port with the valve and seat in which the diameter has been reduced, are also presented in chapter 5. Chapter 6 gives an analysis of the transient flow in the port and the cylinder with moving valve and piston. Finally, conclusions and recommendations for further work are summarised in chapter 7.
Figure 1.1 Configuration of a typical inlet port/valve assembly (Barnes-Moss, 1975)
Figure 1.2 Descriptions of some positions in the inlet port/valve assembly
CHAPTER 2
LITERATURE SURVEY

2.1 INTRODUCTION

In order to achieve improvement in engine development and design, understanding the fundamentals of the complex three-dimensional unsteady turbulent flow in the engine is necessary, and increasingly has been the focus of numerous studies, both experimental and theoretical.

Early research workers used classical diagnostic techniques to investigate the engine flow field, such as hot-wire anemometer, ion probe, spark discharge probe etc. for measurements (Amann, 1985); the method of characteristics for computations (Annand and Roe, 1974). In recent years, new tools - both experimental and computational - have developed rapidly, and led to substantial progress in understanding of the intake and cylinder flow fields. Optical techniques, such as laser-Doppler anemometry (LDA), avoid the flow distortion resulting from physical probes, which can be detrimental to accuracy. Rapid advances in speed and storage of digital computers make realistic three-dimensional predictions of the flow field in the port and valve assembly feasible. However, most of the computational studies are limited by simplifying assumptions in some of the turbulence sub-models and the substantial computer time requirements to achieve reliable predictions. They are largely focused on developing multidimensional computational approaches for understanding the fundamentals of intake flows, and only to a limited extent in design
issues.

2.2 EXPERIMENTAL WORK

2.2.1 Flow through inlet valves

The pioneering work of Tanaka (1929), using steady flow tests, showed that the air flow through the valve and valve seat, with increasing valve lift and at constant pressure drop, has four different flow regions (see Figure 2.1). The first region I shows the flow attached to the valve head and the valve seat at small lift. If the lift is increased, the flow first detaches from the valve head as state II, then continues to detach from the valve seat at high lift and becomes the third region III. The flow shown in the fourth region is influenced by the proximity of the cylinder wall.

Figure 2.2 shows the range of each of the regimes I to IV in the curves of the velocity at the centre of the port against the valve lift. A progressive reduction in the flow rate occurs with change in flow regime.

Tanaka, followed by Wood et al. (1942) have examined the valve flow behavior by means of discharge coefficient measurements under steady flow condition. The discharge coefficient, $C_d$, is defined as the ratio of the actual and the theoretical flow quantities through the valve. The mass flow rate is related to the pressure drop across the valve, the upstream stagnation temperature and a reference area $A_r$, characteristic of the valve design. According to the relation between mass flow rate
and the flow area, the discharge coefficient $C_d$ may also be thought of as the ratio of the effective flow area of the valve $A_e$, to the characteristic area chosen as a reference, $A_r$. $A_e$ is considered to be the throat area of an ideal nozzle, through which the measured flow passes. Several different areas have been chosen to be the characteristic area, such as the valve head area $\frac{1}{4}\pi D_v^2$, the port area at the valve seat $\frac{1}{4}\pi D_p^2$, the geometric minimum area $A_m$, which is the instantaneous valve flow area in the valve passage, and the peripheral lift curtain area $\pi D_v L_v$ (Heywood, 1987). The most convenient reference area is the "curtain" area. Annand and Roe (1974) showed the results of steady flow on a typical inlet poppet valve in Figure 2.3. The discharge coefficient, defined as the ratio of the effective area to the "curtain" area, is shown to be three segments against the valve lift to diameter ratio. These three segments corresponding to different flow patterns at the valve seat were identified, and showed the same as Tanaka described. At the lowest valve lift, the flow separates at the inner seat corners because of its inertia, then re-attaches within the valve passage and emerges as a jet which fills the valve passage because of the viscosity of the fluid. At the intermediate range of lift, the jet is free on the surface of the valve head, but re-attached on the side of the valve seat. In the high lift, the flow separates from both sides of the valve passage, and a free jet is formed. Abrupt decreases in discharge coefficient were observed whenever flow separation occurred. At low valve lifts, when viscosity does play a part in establishing the flow pattern, increasing the Reynolds number decreases the discharge coefficient. At high lifts, the discharge coefficient is almost independent of the Reynolds number.
El Tahry et al (1987) measured mean velocity and turbulence intensity at the valve exit of a motored engine over a range of engine speeds. Comparisons were made with steady flow tests through the intake port, over a range of fixed valve lifts. They identified four phases of the engine induction/compression processes, which fall into the following approximate crank angle ranges:

1) 20° - 60° ATDC during which the flow through the inlet valve is developing strongly with increasing lift, and there is a strong interaction with the piston.

2) 60° - 120° ATDC, in which the mass flow rate through the valve is around its maximum value and the piston acceleration is at a minimum.

3) 120° - 270° ATDC, in which the intake valve is closing and the piston is moving through BDC and upwards into the compression stroke.

4) 270° - 20° ATDC, is the late compression and early expansion stroke of the engine.

These investigators found that strong effects of piston motion and valve lift were observed in phases 1 and 3, and were manifest through significant changes in mean velocity magnitude and direction around the valve exit, with change in crank angle, while in phase 2 the flow is insensitive to piston movement, and the normalised velocity profiles around the valve exit, do not change significantly with crank angle. It was also observed that phase 1 was significantly affected by a change
in upstream port geometry, while phase 2 was not. Major effects of engine speed were noted for phases 1 and 3, whilst phase 2 was insensitive to speed change in regard to change in velocity profile around the valve exit. Comparisons by the authors with steady flow tests led them to conclude that it is the phase 2 region of the transient engine flow that is best represented by steady flow tests. In regard to turbulence intensity at the valve exit the authors found their results falling into the four phases observed for the mean flow.

Suen (1992) undertook an experimental study of the mean flow and turbulence intensity in the exit region of the inlet valve of a motored engine, in which the inlet valve axis is offset to the cylinder axis (see Figure 2.4). His findings confirmed the four phases observed by El-Tahry et al, but detailed measurements in the jet flow region of the valve exit, showed the jet to consist of a central core and highly turbulent shear layer at its edges, which increases with valve lift. The jet flow issuing from the valve, was found to be more stable in the XZ than the YZ plane through the valve axis. This instability gives rise to a "flapping" motion of the jet flow in the YZ plane, resulting in formation of secondary flows. It was occasionally observed that the flapping action caused flow reversal at crank angle positions near maximum valve lift, and this impeded the induction flow despite the fact that the piston speed was at its maximum. Significant cycle to cycle variation in mean velocities in the valve exit region were observed in phase 3 - the closing period of the intake valve.

Bicen et al (1984) used LDA to measure the velocity components near the valve exit to show the velocities in the jet to be about 15 times the
mean piston speed. The separation of the jet generates the recirculation flow in the cylinder.

2.2.2 In-cylinder flow

The measurement of the in-cylinder flow is best achieved by laser-Doppler anemometry, which can measure the instantaneous velocity accurately without intrusive probes or corrections for pressure and temperature. Detailed LDA measurements coupled with flow visualisation experiments map out the flow field prior to combustion in engines, giving a good understanding of its overall character.

Ekchian and Hoult (1979) in flow visualisation studies of the induction process in an engine like model with an inlet valve axisymmetric to the cylinder bore, observed a free vortex structure in the flow, in which a double vortex structure existed in the cylinder - a small toroidal vortex trapped adjacent to the cylinder head and rotating in a direction opposite to that of a much larger toroidal vortex located at approximately half instantaneous stroke. In studies with an offset valve the authors found that the planes containing the axes of rotation of the vortex system were now at an angle to the cylinder axis.

It was found that there was little dissipation of energy during the vortex formation in the period to BDC and that a significant amount of the induction flow kinetic energy is contained in the large vortex structure, which was found to be independent of engine speed.

Rapid break up of the large vortex structure was found to be dependent
on engine stroke and the strength (circulation $\Gamma$) of the vortex. For a given engine the critical break up time $t_c$ is proportional to $\Gamma$. Hence the large scale vortex structure of the induction period is less than $t_c$. Break up of the vortex structure late in the compression stroke results in the conversion of stored kinetic energy of the vortex to turbulence. The observations of Echian and Hoult were found to be independent of speed in the range of their experiments (750-2500 rpm).

In-cylinder flow is strongly dependent on the inlet port, valve and cylinder head geometry. Inlet valve location affects the in-cylinder flow pattern. Uzkan et al (1983) showed in their photographs the motion of small air bubbles in water flow generated by an offset inlet valve at fixed valve lift and at successive planes below the valve head. Figure 2.5 shows the flow on the plane which was 1.75 diameters below the head surface. When the ratio of the valve lift to valve diameter was 0.056 ($L/D = 0.056$), two counter rotating vortices were displayed. The two vortices were also seen at $L/D = 0.11$. When $L/D$ was 0.17, there was only a trace of the second weaker vortex. At $L/D = 0.22$, there was only one vortex visible. Uzkan et al. pointed out that the relative strength of two vortices depends on the flow asymmetry. The two vortices gradually merged at increasing axial distances from the valve head, with the stronger vortex tending to overwhelm the weaker one. They found that offsetting the valve brought an increase in the swirl level. The offset effects were maximum at high valve lifts, but were quite small at low lifts as shown in Figure 2.6.

A four-valve-pent-roof cylinder head can generate tumbling which is
also called vertical swirl or barrel swirl. Khalighi (1990), and Kent et al (1989) both observed the effects of two open intake ports or one open port using full field flow visualisation techniques. Closing one port reduces the effective flow area of the port/valve, but increases the intake jet flow velocity for generating more strong swirl and tumble. Therefore, it is inevitable that increased flow restriction is required for intake generated swirl or tumble. That is the reason for stronger swirl or tumble generated by the marked intake valve.

2.2.3 Improvements in port/valve design

In view of the importance of the inlet port and the inlet valve in determining the performance of an internal combustion engine, it has been recognised by engine designers that volumetric efficiency, and therefore power output, may be increased by improvement in the flow characteristics of the induction system. This work is usually done by empirical studies which are based on the method of "Trial and error".

2.2.3.1 Effects of valve shape

Tanaka (1929) tested the influences on the air flow characteristics by changing five elements of the valve: fillet radius of the valve stem, angle of the valve head, overlap width between the faces of the valve and seat, angle of the valve seat, and sharp corners on the valve and seat. The most effective influence was the rounding of the sharp corners on the valve and the seat. He remarked that rounding of the corners in effect made a convergent-divergent nozzle of the crevice between the valve and the seat, which obtained an increase of flow
quantities up to 23%. But, he did not attempt to extend the application of this principle in practice to the valve design.

Dennison et al (1931) tried to realize this application. They compared the steady air flow test results between the standard valve and two special types of the valve which were called "tulip" type and "drop" type, representing an effort to reduce resistance by giving the valve a favourable shape. The "tulip" valve and "drop" valve are both an improvement in regard to increasing the intake mass flow amounting to between 10 to 50 percent at operating valve lifts. Dennison et al did extend these models in an effort to improve the flow characteristics by modification of the valve. But, these modifications were on idealised valve shapes, unsuitable for use in an engine.

Starting with an elementary valve and seat, Hunter (1983) built up various shapes with plasticine for developing valve and port designs. This is a typical "Trial and error" method. Figure 2.7 shows one combination of these modifications and obtained coefficients. The enlarged valve head is similar to the "tulip" valve. Though it is impractical, it gives the possibility that an appreciable pressure recovery may occur in the expanding portion of the passage between the valve and its seat. Once again, it proved that a convergent-divergent passage would obtain higher coefficients.

Kastner et al (1963, 1964) used "static" and "kinetic" rigs to test the influence of different valve shapes on the flow. For "static" test, the valve was held in a series of fixed lifts by a multifaced cam. The tests under "dynamic" conditions were performed as "static" test, except that
the multifaced cam for static lift settings was replaced by cams of polynomial type. Thus, the valve was reciprocated whilst the overall pressure drop across the valve and port was kept constant, similar to those which would occur in high speed engines. Pressure drops within the range 0-20 inHg were employed and camshaft speeds from 400 rev/min to 3500 rev/min (equivalent to 7000 rev/min crankshaft speed in a four stroke engine). For "static" tests, comparison of the discharge coefficient, which was defined as the ratio of the measured flow to theoretical flow, was made for different valve shapes. It showed that rounding the corners of the valve seat and the valve head increased the mass flow by up to 20%, because the vena contracts had less restriction, and the flow separation was delayed from the valve seat and the valve head until a higher valve lift was reached. Further improvements could be gained by a valve passage acting as an expansion nozzle. Both "static" and "dynamic" tests showed the influence of the port shape on the intake flow, which will be presented in the following section.

2.2.3.2 Effect of port shape

A series of "dynamic" tests mentioned above was used for different curved inlet port configurations by Kastner et al. The port configuration - in particular the shape of the port near the port exit and the passage between the valve seat and the valve head - affects the flow pattern. Plotting experimental values of the overall coefficient, defined as the ratio of the measured flow rate under dynamic conditions to that the theoretical flow rate exhibited a higher coefficient when the square ratio of the port diameter to the valve seat
diameter was from 2.25 to 4 due to reducing resistance in the port, further increase had little improvement of the overall coefficient. The direction of flow as it entered the valve passage was important in influencing the point at which flow separation occurs. Figure 2.8 shows four different tapered port configurations based on the basic shape III. The minimum throat diameters of III, and IIIb were the same as the valve seat diameter, whilst those of IIIc and IIId were smaller than the valve seat diameter by 13% and 30% respectively. The overall discharge coefficient increased with the tapered port IIIc to IIIc, because these tapered port shapes directed the flow into a more favourable direction. The taper could not be too big, due to the restriction area at the throat.

Hardenberg and Daudel (1975) investigated the effects of the helical inlet port shape on the volumetric efficiency of an engine by the wall pressure test method. They found that the volumetric efficiency increased continuously as the valve seat diameter of the port was reduced from 59 mm to 53 mm with overall pressure dropping. When the diameter was further reduced to 51 mm, the pressure was increased again, and the volumetric efficiency decreased slightly. Therefore, 53 mm was the optimum valve seat diameter with which the port induced air to the cylinder with an optimum intake velocity. This phenomenon is shown clearly with the surface pressure distribution in Figure 2.9. The high pressure in the zone above the valve showed that the air tended to follow the shortest path into the cylinder, indicating the ratio of the diameter of the upper part to the valve seat diameter could not too big. Hardenberg and Daudel concluded that the high volumetric efficiency was not obtained in the largest port, but in a smaller one
where the lowest pressure and highest velocity occurred along the passage between the valve seat and the valve head.

2.2.3.3 Effect of valve lift

In many cases, the pattern of the flow through the inlet valve, and the curve of discharge coefficient, $C_d$, versus the ratio of valve lift to valve diameter, $L/D$, show an increase of discharge coefficient at low valve lift. Near $L/D = 0.08 - 0.1$, $C_d$ decreases slightly, then increases until $L/D$ is up to 0.16. In the region of $L/D = 0.16 - 0.27$, there is a steady fall of $C_d$ which reaches its minimum value at $L/D = 0.27$. The valve lift is limited in an engine by the necessity of getting the valve open and shut quickly. In automobile and aircraft engines, the maximum valve lift is about one-fifth of the valve head diameter (Wood et al, 1942). The typical maximum value of $L/D$ is 0.25.

The flow pattern in the vicinity of the inlet valve and inside the cylinder vary with the valve lift. Cheung et al (1990) presented their LDA detailed mean flow measurements and swirl velocities in a helical port/cylinder assembly. Their experimental results show the steady flow in the port, further upstream than 15 mm from the cylinder head surface, is hardly affected by variation of valve lift. In-cylinder flow pattern is affected by the valve lift: at low lifts, the swirl pattern is characterised by a double rotation under the valve; at higher lifts, there is only a single sense of rotation. This phenomenon is also observed by Khalighi (1990).
2.2.3.4 Effect of pressure drop

The flow of air between an intake valve and its seat is of similar nature to the flow through a venturi in which the passage converges and diverges. The flow characteristics are determined by effects of viscous friction, flow separation, and pressure recovery.

Separation of the flow from the wall is the main factor to effect the discharge coefficient. It causes poor pressure recovery, and reduce of the discharge coefficient. If the overall static pressure drop across the valve is given, pressure recovery tends to increase the discharge coefficient because the velocity at the throat is increased with lower pressure.

Woods et al (1942) compared the discharge coefficients against pressure drop ratio between three different valve port designs and the venturi, which was assumed to operate with no friction, no flow separation and 100% pressure recovery (Figure 2.10). The comparison was based on the same throat area indicating at a fixed valve lift. Case C works like a venturi, when the pressure drop ratio is lower, the discharge coefficient is considerably higher than unity because of good pressure recovery. When the pressure drop ratio is increased, the discharge coefficient decreases rapidly. For actual valves such as case B and A, the discharge coefficient is not greatly affected by a change of the pressure drop ratio. Similar results were obtained in Tanaka's study (1929), shown in Figure 2.11. At small valve lift, the discharge coefficient decreases a little with increase of the pressure drop, because it approaches venturi better than at high valve lift. When the valve lift
increases up to about $L/D = 0.20$, (in Tanaka's test, the valve lift is 16 mm), the discharge coefficient increases a little with the pressure drop because of increase of the effective area. Further more with an increase of the valve lift ($L/D > 0.25$), the discharge coefficient increases a little at the range of low pressure drop, and thereafter remains substantially constant. At high valve lift, the passage has more divergence, the flow breaks away from the wall, the discharge coefficient is independent to the pressure drop then.

Therefore, in general, the discharge coefficient is approximately constant and independent of the pressure drop except at small valve lift, when it decreases a little with the pressure drop.

2.3 COMPUTATIONAL WORK

2.3.1 Introduction

The induction system generates an unsteady, highly three-dimensional flow within the intake port and the cylinder. The periodic motions of piston and valve cause the flow to be unsteady, and a combination of geometric constraints such as curved port with valve stem, off-centre valve locations, cylinder head and piston geometry, result in the flow being three-dimensional in space.

A well-designed intake port induces the flow to have a desired velocity profile at the inlet valve exit and provide maximum airflow. In production engines, there are two main types of port: directed ports and helical ports. In the directed port, the flow has a non-uniform
distribution around the inlet valve periphery. With a substantial net angular momentum about the cylinder axis, the flow enters the cylinder tangential towards the cylinder wall where it is reflected sideways and downward in a swirling motion. In the helical port, the flow is forced to rotate about the valve axis prior to entering the cylinder.

Flow characteristics at the valve exit, depend on the port shape and the valve lift. The flow through the valve produces sharp shear layers off the valve head and the valve seat. These shear layers are dynamically unstable and break down initially into ring-like vortices which merge to form large-scale vortices. These large-scale vortices in turn break down into three-dimensional turbulence motion.

The annular transient jet interacts with the cylinder head, cylinder walls, and piston surface to establish unsteady, high three-dimensional turbulent flow with strong recirculations whose size and strength may vary substantially cycle to cycle.

As the inlet valve closes, the sharp shear layer disappears, the intake-generated vortex pattern breaks down except the main vortex sometimes survives. The large-scale circulations convect the small scale turbulence, which will be diffused through the cylinder by the large scale motion caused by the piston-induced compression.

During the beginning of the compression stroke, the flow pattern reflects the effect of inlet port/valve geometry and the small effect of the piston geometry. As the piston approaches top dead centre (TDC),
the piston and cylinder head geometries play the dominant role in the flow structure. The angular momentum of the air which enters the cylinder during the intake stroke decays during the compression process due to friction at the walls and turbulent dissipation within the fluid, in a cylindrical chamber with flat piston in the absence of squish. Turbulence levels are of 0.3-0.5 times the mean piston speed, at TDC, with tendencies towards homogeneity and isotropy. Neglecting the effect of friction, the angular velocity of the swirling in-cylinder air flow increases to conserve its angular momentum, resulting an increase in turbulence intensity.

2.3.2 Mathematical Modelling

As the flow in the inlet port and cylinder exhibits spatial and temporal variations, it is impractical to obtain a direct numerical solution of these differential equations from the basis of the whole science of fluid mechanics, which are referred to as the Navier-Stokes equations. For engineering applications, two statistical methods have been developed and tested: Statistical Flux Modelling (SFM), and Large-Eddy Simulation (LES) or Subgrid-Scale Modelling (SGS). They both decompose the instantaneous velocity \( u_i \) into a "mean" component \( U_i \) and a fluctuating velocity about the mean denoted by \( u'_i \).

thus: \[ u_i = U_i + u'_i \]

SFM employs an ensemble-averaging process to convert the full Navier-Stokes equations for unsteady laminar flow into the ensemble-average equations for turbulent flow in which every variable is
expressed through its mean value and the random fluctuating turbulent components. For periodic flow, the ensemble-average velocity \( U_1 \) are defined as the cycle averaged values, and the \( u'_i \) are departures from \( U_1 \). The additional terms arising from the averaging process, \( \overline{u_i u_j} \), are called turbulent stresses or Reynolds stresses. To relate these stresses directly to the mean properties of the flow is the task of a turbulent model.

In ensemble-averaging, the turbulence contains some contribution from cycle to cycle flow variation. The ensemble-averaged properties vary much less rapidly in space than the instantaneous values, and may exhibit significant variations in only one or two dimensions even though the instantaneous motion is always three dimensional. Hence, further major simplifications are possible for solving the equations which describe the fluid's mean motion. The \( k-\varepsilon \) two equation model (Launder and Spalding, 1972) and Reynolds stress model (Launder et al, 1975) are the turbulence models under the ensemble-averaging heading for linking the turbulence stresses to calculable properties of the mean motion.

The \( k-\varepsilon \) model is the most widely used turbulence model. It assumes a Newtonian relationship between the turbulence stresses and mean strain field, and calculates the turbulent viscosity appearing in this relationship from the local ensemble-averaged values of two parameters of turbulence structure: turbulent kinetic energy \( k \) and its dissipation rate. The Reynolds stress model directly employs modelled transport equations for the Reynolds stress components, \( \overline{u_i u_j} \). The process comprises seven simultaneous partial differential equations for
the six stress components and the dissipation rate. This model provides more fundamentally-based estimates of the Reynolds stress components. But, computationally, it is expensive to solve the seven equations.

In LES, the turbulence is related to events in the current cycle. It calculates the large-scale three-dimensional time-dependent turbulence structure in a single realisation of the flow. Only one analogous subdivision needs be modelled according to how the small-scale turbulence stresses are obtained. The Large-Eddy Simulation separates the large and small scale motions, and can solve cycle to cycle variations. However, the computing time and storage requirements of the calculations are enormous. It always requires a three-dimensional time-dependent solution even where mean flow is two-dimensional and steady. Therefore, the Large-Eddy Simulation cannot be regarded as a practical means of computing turbulent flows. It remains a research tool for investigating the phenomenon of turbulence.

The Reynolds stress model has emerged in engine applications, but compared to the two equation k-ε model, it imposes an additional computational load on the mathematical model. The first application of the Reynolds stress model was performed by El Tahry (1984). Later, he presented a comparative study mainly between the k-ε model and the Reynolds stress model in engine type flow for non-swirl and swirl flow cases at 200 rpm (1985).

For mean flow velocities, the level of agreement between experimental data and the values obtained with the k-ε model and the Reynolds stress
model is similar for both non-swirl flow and swirl flow. At 36° ATDC, mean velocities predicted by the two models were in excellent agreement with experiments except those at the large recirculation area, where the Reynolds stress model predicted the velocities in fair agreement with the experiments, while the k-ε model underpredicted the peak velocity by 50%. At 90° ATDC, the peak mean velocity at recirculation areas, calculated by the Reynolds stress model, were in excellent agreement with the experimental data except in the vicinity of the large recirculation where the peak velocity was overestimated by 35%. The k-ε model underpredicted the peak velocity within 25% at recirculation areas.

For turbulent velocities, the Reynolds stress model underpredicted the peak turbulent velocity by 15% for non-swirl flow. Unlike the experiments, the predicted turbulent velocity reduced sharply with increasing radii beyond the location of the peak velocity. The k-ε model underpredicted the peak turbulent velocity within 30%, but the overall trend of the turbulent velocity profile agreed reasonably well with the experiments. It did not fall as sharply as the Reynolds stress model with increasing radii beyond the location of peak velocity. For swirl flow, both the k-ε model and the Reynolds stress model predicted the turbulent velocities in good agreement with the experiment at 36° ATDC, except in the large recirculation area where the k-ε model underpredicted the turbulent velocities. At 90° ATDC, the Reynolds stress model was more accurate than the k-ε model in predicting the turbulent velocities, but at the out radii close to the vicinity of the jet, the results obtained by the k-ε model were in better agreement than the Reynolds stress model with the experiments.
The results showed that for the turbulent field overall, the Reynolds stress model gives a more accurate representation of turbulence. It predicted the energy levels in better accord with measurements than the k-ε model. But, it needed a much more complex set of equations, resulting in additional storage and computing time. Besides, El-Tahry found that the velocity vector oscillations which were inherent in the solution of the momentum equation obtained by the computer code CONCHAS, are exacerbated with the Reynolds stress model. In CONCHAS, the solution algorithm employed is semi-implicit in character. The stability limit of the numerical treatment is affected by the Reynolds stress model.

Dupont (1988) carried out a computational study in four variations of a production spark ignition engine using the KIVA code, in which the initial turbulence model was Large-Eddy Simulation. He found that the Large-Eddy Simulation model did not correctly account for the evolution and level of turbulent kinetic energy with most inaccuracies occurring during the compression stroke. Argueyrolles et al (1988) calculated wall heat flux in a pent-roof-shaped combustion chamber using KIVA's original Large-Eddy Simulation model. They had to apply a corrective factor of 8 to the local heat fluxes to make the globally computed heat loss consistent with that obtained using thermodynamic analysis of recorded pressure trace. Because of these defects, Taghavi et al replaced KIVA's initial turbulence model by a k-ε model based on that proposed by El Tahry (1983). Then, they successfully simulated the flow in an inlet port and a combustion chamber using the KIVA code (1989). They observed the effect of the cylinder wall on the development of the large scale structure of the
flow inside the combustion chamber, and effect of the relative position of the intake valve with respect to the rest of the cylinder head on the prevailing flow field in the combustion chamber. They noticed that the limitation of the k-ε model in predicting flow separation correctly might lead to inaccurate valve annular jet angle and discharge coefficients. The discharge coefficient $C_d$ will be lower than expected, because of overpredicted pressure drop.

2.3.3 The k-ε turbulence model

The k-ε model (Launer and Spalding, 1972) is only valid in the fully turbulent flow region in which the Reynolds number is sufficiently high for the viscous effects to be unimportant. In the near wall region, turbulence Reynolds numbers are low, and the turbulent flow is strongly affected by viscosity, hence the high Reynolds number version of the k-ε model ceases to apply. A special near wall treatment is required. The standard k-ε model uses the wall function to bridge the viscous wall layer (Launer and Spalding, 1974).

The k-ε model is used by nearly all researchers developing or applying multi-dimensional modelling methods for predicting the flow in engines. However, for the very complex intake flow, which contains sharp streamline curvature, steep pressure gradients and separation regions, the standard k-ε model is inadequate to reproduce the complicated small-scale fluid motion. Rodi and Scheuerer (1986) showed that the k-ε model does not respond properly to adverse pressure gradients, it allows the flow to go beyond the point where it should in practice separate. The reason of the failure is that the k-ε
model underpredicts the value of the turbulence dissipation rate in the boundary layer, and results in higher turbulent kinetic energy $k$ and consequently higher shear stress $-\rho u'v'$, which causes too much momentum to be diffused into the boundary layer from the core flow. The modification in the case of adverse pressure gradient is an effect of enhancement of the generation rate of the turbulence dissipation $\varepsilon$, such as the model of Hanjalic and Launder (1980), in which production is suggested to be more sensitive to the "normal-stress" production than the "shear-stress" production. In the case of adverse pressure gradient, $\frac{\partial U}{\partial x}$ is negative, the generation is due to normal strains, creating higher values of $\varepsilon$, lower energy and shear-stress levels. But, there is no overall improvement on the standard $k-\varepsilon$ model.

Streamline curvature has a remarkably large effect on the turbulence stresses (Bradshaw, 1973). The standard $k-\varepsilon$ model is inadequate to predict the effect of the streamline curvature on the turbulent flow due to error in assuming a laminar-like stress relation and isotropy of turbulent exchange.

The observed effect of streamline curvature on a shear layer is to diminish the turbulent intensity and shear stress when the angular momentum of the mean flow increases with increasing radius of curvature and to increase these quantities in the opposite situation. The generation of different Reynolds stress is highly non-uniform. Therefore, modification is carried out by introducing a curvature "Richardson number", either a gradient Richardson number or a flux Richardson number, as a correlating parameter to account for the
effect of streamline curvature on turbulence transport. The gradient
Richardson number is defined as the ratio of body force to inertia
force, the flux Richardson number is defined as the ratio of the
generation of turbulence energy by extra body force to the total
production of turbulence energy. Different interpretations of
Richardson numbers are needed for different flows.

In the case of strong flow acceleration, the k-ε model is unable to
reproduce the effect either, due to the reduction and anisotropy of
turbulence intensity.

Because of the deficiencies of the k-ε model, many suggestions have
been made for extending the k-ε turbulence model to enable its use in
low Reynolds number regions.

One low-Reynolds-number treatment is called a "two-layer" model. A
one equation eddy viscosity model is employed purely to cover the
near-wall region, beyond where the k-ε model is used. In the one
equation model, a transport equation for turbulent kinetic energy is
provided. The one equation model of Norris and Reynolds (1975) can
capture the behaviour of the flow at the boundary layer with adverse
pressure gradient due to high levels of turbulence energy near the
attached point of the flow with the boundary layer (Rodi and
Scheuerer, 1986). Yap (1987) used the one equation model of
Wolfshtein (1969) for the viscous sublayer of a pipe expansion. It gave
predicted levels of heat transfer coefficient downstream of the pipe
expansion lower than the experiment levels.
Low-Reynolds-number treatment using two equation models have focused on introducing various viscosity dependent terms and/or damping functions in order to achieve the observed reduction of turbulent transport very near to the wall. The original formulation of Jones and Launder (1972) adopted an equation for "dissipation rate" $\tilde{\varepsilon}$ rather than $\varepsilon$ as dependent variable, in order to simplify the application of the wall boundary condition to be $\tilde{\varepsilon}=0$. $\tilde{\varepsilon} = \varepsilon - v v_i \left( \frac{\partial^2 U_i}{\partial y^2} \right)^2$, where $v = f C_{\mu} \frac{k^2}{\tilde{\varepsilon}}$, and $f$ is a function of $R_t$. $R_t = k^2 / \tilde{\varepsilon}$.

Patel et al (1985) have provided an assessment of seven such model's performances in boundary layers with pressure gradients. These models of the $k$-$\varepsilon$ group have collectively been applied to a wide range of turbulent flows in which the viscous sublayer becomes thicker than the standard $k$-$\varepsilon$ model. Three of the variants of the $k$-$\varepsilon$ model, called LS - Launder and Sharma (1972), CH - Chen (1982) and LB - Lam and Bremboret (1981), and one another model called WR - Wilcox and Rabesin (1980) will be stated in Chapter 3, section 3.1.4.2., where a typical example of the prediction for a moderately accelerated boundary layer by these models will be shown.

Yap (1987) applied the LS model to predict heat transfer downstream of an abrupt pipe enlargement and found the peak heat transfer rates were overpredicted by five times. This was because the near-wall length scale became far too large in the separation region, producing excessive near-wall diffusion coefficients. For complex flow involving separation, Yap modified the LS model by introducing a source term $S_{\varepsilon}$ to the $\varepsilon$ equation which acted throughout the near-wall region not
just in the viscosity affected sublayer. The term vanishes in local equilibrium region where $\ell = C_t Y$ ($C_t = 2.44$, $Y$ is the distance from the wall) and in regions far from the wall where $\ell / C_t Y << 1$. In the near-wall region of separated flows where $\ell / C_t Y > 1$, $S_\varepsilon$ is positive and the level of $\bar{\varepsilon}$ is thereby raised, thus reducing the length scale. This modified model produced a very beneficial effect on predicted heat transfer rates in a range of separated and stagnating flows.

It should be noticed that any low-Reynolds-number model has only a limited range of applicability. Extensions of the k-\( \varepsilon \) model tend to be arrived at in a phenomenological manner, and are often just for specific cases.

However, the current version of the CFD code STAR-CD for use in the present study has no access for addressing any of the modifications of the k-\( \varepsilon \) turbulence model that are required. The turbulence model employed in the code is the standard k-\( \varepsilon \) model.

Nevertheless, the k-\( \varepsilon \) model is very widely used in industrial fluid flow computations because of its relative simplicity, robustness and fairly general applicability. Some studies showed that if care is taken to describe the inlet boundary conditions properly, the k-\( \varepsilon \) model can predict reasonably good agreement with experiments.

An example is the study by Leschziner and Rodi (1981). They studied two strongly swirling jets in still air for examining the performances of the standard k-\( \varepsilon \) model and its two variants, one was the streamline curvature modification based on the algebraic stress model of Gibson
(1978), the other was the preferential dissipation modification originally proposed by Hanjalic and Launder (1979) in which the basic idea of preferential-normal-stress treatment was retained, with the stresses defined in a streamline coordinate system to ensure the main effect in the important shear-layer regions of the flow. In this study, it was found that the standard k-ε model could be made to yield good prediction if the initial condition is properly described. Specifically, the inlet condition for velocity profile and the turbulent dissipation rate, the importance of which was usually ignored. These were shown to play a crucial role in achieving the accuracy of the prediction. Also, in the study of Hendricks and Brighton (1975), the k-ε model was used for turbulent confined jet mixing without swirl. Comparing the predicted results with experimental data, Hendricks and Brighton found that the inlet turbulence level of the stream had a significant effect on the mean velocity and pressure development. Comparison of the results between the entrance turbulent kinetic energy levels of 0.01, 0.03, 0.05 and 0.07, showed that the highest turbulence level jet had the highest pressure that was three times higher than the lowest turbulence level jet, at the plane located four radii's distances from the entrance of the pipe.

2.3.4 Calculation of the flow in intake port/cylinder

2.3.4.1 The computational mesh model

Because of the complex geometry, most of the computations of the intake flow in engines are restricted to in-cylinder flow. El Tahry (1982), Jones (1984), Arcoumanis et al (1986) and Yamada et al
(1986) made the prediction of fluid motion in engine cylinders without port/valve geometry. They defined the interface of the cylinder and the inlet valve to be an inlet boundary, and specified all variables as boundary conditions at the inlet, similarly initial conditions were obtained from prior experiments or intelligent estimates.

Figure 2.12 shows the computational domain represented by Jones (1984). He pointed out that a routine method for applying boundaries as an inlet condition must be developed and tested. Specifically, the information required was the instantaneous velocity profiles (all three components), turbulent kinetic energy and length scale. In order to represent the port characteristic correctly, Brandstatter et al (1985) interpolated measured velocity profiles in the axial, radial and circumferential directions to yield the three velocity components in each mesh cell, within the valve gap at each valve lift. El Tahry (1982) initiated the swirl velocity, axial velocity, turbulent kinetic energy and the dissipation rate at inlet valve closure to perform the computation of the flow in the cylinder. Yamada et al (1986) adopted the flow velocity vectors, at certain points on the inlet valve outlets, as boundary conditions to investigate the effects of the port configuration on the in-cylinder flow. The flow velocity vectors were measured by hot wire anemometer at maximum valve lift using a steady air flow test rig. Figure 2.13 shows the measured velocity profiles obtained for a helical port and a straight port respectively. As the flow velocity from the valve outlet changed with crank angle, they used a program to calculate the volumetric efficiency based on the pressure pulsation in the intake manifold, then derived the flow velocity for every mesh point at each crank angle from the ratio of the mass flow rate at the
particular cell to the total flow rate.

However, it is clear that for a real engine, the conditions for transposition of steady state results to transient flow are not satisfied in practice. With recent development of digital super computers realistic, three-dimensional predictions of the flow field, including the inlet port and the cylinder with fluid dynamic computer code have only recently become feasible. But, creating the computational domain of the practical inlet port/valve assembly will always be a thorny task because of its much more complex geometry.

Isshiki et al (1985) generated the inlet port computational domain with staircase-like boundaries to realise the prediction of the flow in the inlet port and its valve. The effects of the intake port configuration and valve head geometry on the induction swirl intensity and the volumetric efficiency were investigated for the design issues. Figure 2.14 shows the addressed port configurations. Figure 2.15 shows the intake valve models.

Some studies had to make several simplifications for analysing the effects of complex inlet port configuration in in-cylinder flow, such as the axis of the valve stem coincident with port axis (Errera, 1987 and Errera et al, 1988), and no valve stem (Haworth et al, 1990) etc.

In the study of Errera (1987), the inlet boundary condition was specified by the "guessing" velocities, not according to the experimental data. The inlet port was just a curved pipe. Though the valve axis was offset to the cylinder axis and the port orientation was
adjusted, there was not enough effect of swirl on the in-cylinder flow. A swirl generator had to be used. The predicted flow showed was the features of the flow in the curved pipe, not in a curved port. The in-cylinder flow did not be validated by experiments. Its continuous research (Errera et al, 1988) predicted the flow during compression. One of the conclusions of the research is that the computational results of the flow in such an inlet port/cylinder assembly without imposing experimental intake velocities are questionable. Haworth et al (1990) simulated the flow in a four valve engine with three different inlet port/valve configurations. They made some simplifications of mesh generation such as no valve stem, disk shaped valve etc. They found that the tumbling configuration (the inlet valve with shrouds orientated to produce tumble) created about twice turbulence level at 30° BTDC than the swirl configuration (the inlet valve with shrouds orientated to produce swirl). Much of the turbulence production during the late compression were attributable to the tumbling components of angular momentum. However, their computational domain did not represent the practical port/valve geometry. They mentioned that their further investigation of port designs would be more representative of actual engine geometry.

The effects of practical curved or helical inlet/valve shape on the port and in-cylinder flows have been represented only in the last few years (Taghavi et al, 1989, Sugiura, 1990 and Naitoh et al, 1990). Sugiura et al (1990) compared the computational results between the cases with and without valve stem. They found there were differences in the flow structures such as the flow separations near the valve stem and the valve seat, the vortices in the cylinder corner. The predicted mass flow
rate in the case with valve stem was in a good agreement with the measured one, which was 1.8% less; while the calculated mass flow rate in the cases without valve stem was 11% overestimated due to overpredicted pressure drop across the valve.

2.3.4.2 Prediction of intake flow

The techniques for realistic prediction of the intake flow have been developing during the last few years. Most of the theoretical studies aim to assess the accuracy of the multi-dimensional model predictions of the flow, with emphases on their sub-models and numerical procedures.

Computations were first performed in an axisymmetric port/valve assembly by Gosman and Ahmed (1987). They employed a body-fitted computational mesh and the k-ε turbulence model to realise the prediction of the steady flow within the port/valve assembly. Figure 2.16 shows the comparison of the predicted discharge coefficient $C_d$ with the measured $C_d$. The predicted mean velocity profile at the valve exit agreed reasonably well with LDA measured mean flow data except where influenced by the flow separations, which were not predicted. Similar results were obtained by Demirdzic et al (1987) in an axisymmetric inlet port/cylinder assembly at a high valve lift. One of the reasons causing disagreement at high valve lift between predictions and measurements is the inability of the standard k-ε model in accurately predicting flow separations caused by the effects of streamline curvature and steep pressure gradients.
Because of the profound effects of the steep variations in pressure and sharp streamline curvature, Naser (1990) avoided the use of the wall function at the wall boundary layer. He developed a "2-L-M" model to simulate the flow in an axisymmetric port/cylinder assembly. The "2-L-M" model employs a variant of the standard k-ε model of Hanjalic and Launder (1980) in the region away from the wall, and one-equation model of Norris and Reynolds (1975) in the near-wall region. The boundary layer thickness \( d \) is defined as the distance from the wall to the point where the mean velocity is 99% of the first inflexion point in the cross-stream mean velocity profile. The switch over to the model of Hanjalic and Launder is made at the location where \( \mu_s/\mu \geq 10 \), corresponds to a turbulence Reynolds number \( R_y \geq 60 \).

Using this model, the small recirculation bubbles on the valve head surface and at the valve seat corner were reproduced at high lift. The predicted results showed substantially overall improved agreement with the experimental data. Compared with the measurement, the predicted discharge coefficient \( C_d \) using the 2-L-M model overpredicted 3\%, while the \( C_d \) using the k-ε model overpredicted 26\%. For the mean velocities near the valve seat surface, the 2-L-M model underestimated 5-10\%, the k-ε model underestimated by approximately 30\%. It is clearly seen that the discrepancies between the prediction and experiment still exists. Also, the length scale obtained by the 2-L-M model showed a sudden increase in the vicinity of the valve crown, which was within the potential core-like region, and created very little variation in the distribution of shear stress. This is because the proper account of the effect of the streamline curvature can not be considered in the 2-L-M model which is a combination of the two-layer model and a modified k-ε model made following the
recomedation of Hanjalic and Launder (1980). Both of these two models perform well in the near-wall region with adverse pressure gradient, but there is no correlating parameter with the effect of the curvature.

Naitoh et al (1990) focused on the numerical technique to realise predictions of the high Reynolds number incompressible flow in the engine with one or two inlet ports. No explicit turbulence models were used. The flow was simulated with large heat transfer and homogeneous compression. Taghavi et al (1989) and Aita et al (1991) both demonstrated successfully the use of CAD/CAE in creating the complete intake port/valve and cylinder domains for applying the CFD code to analyse the flow field. Taghavi et al used the KIVA code with the standard k-ε model to simulate the flow during induction under motoring condition and obtained preliminary analysis of the velocity and residual burned gas mass fraction field. Effects of the offset of the valve centre from the cylinder centre on the flow structure in the combustion chamber were observed. The effect of wall proximity is to create an axis of symmetry for the flow on the horizontal plane. The axis joins the closest and the farthest points from the cylinder wall to the valve periphery, and persists at least up to BDC. This effect seems to erase the influence of other geometric features of the cylinder head, such as the orientation of the inlet port, the shape and the position of the spark plug well. The port flow accelerates in the vicinity of the concave wall, and decelerates near the convex wall. Consequently, the fresh charge penetrates the cylinder first from the passage between the valve and the closest cylinder wall. Aita et al used the PAM-FLUID code with both the k-ε model and Subgrid-scale Modelling (or Large-
Eddy simulation) for predicting steady flow and transient flow in a helical port/cylinder assembly. Swirl evolution then was reported. The streamlines visualization of the steady flow was made comparison between the simulation and experiment. The particle traces showed that separation happened in several regions due to the impingement of the flow on the valve stem, and confirmed the important influence of the valve stem on the inlet jet characteristics.

All these work are in the first stage of realizing the simulation of the flow in an inlet port-valve-cylinder assembly. The flow analyses are not detailed, and the accuracy of the results are not validated with detailed experimental results. However, these work proved that the application of the computational fluid dynamics simulation can be extended to a wide range of practical engine geometries and flow situations.

2.4 ENGINE DESIGN AND THE CONTRIBUTION OF CFD

Empiricism has been the traditional method for engine design. In order to design an inlet port/valve assembly, giving the desired velocity profiles at the inlet valve exit with minimum pressure drop across the port and the valve, a designer would evaluate several designs by trial and error, to see which one worked best. Such development testing is time consuming and expensive.

The new experimental techniques - detailed LDA measurements coupled with flow visualisation experiments, help the designer to understand what is going on within the intake system. But, the
limitation on optical access for laser-Doppler anemometry and the pointwise data obtained with the method, and the qualitative nature of visualization, gives overall, a limited insight to the flow.

Realistic prediction with a computational fluid dynamics model maps the full flow pattern in the inlet port/valve and the cylinder, giving a significant understanding of the fundamentals of intake flow in engines. The study of Isshiki et al (1985) is an early work addressing the design issue, but the mesh model is not realistic. Gosman et al (1985), Henriot et al (1989), and Haworth (1990) predicted the flow structure near TDC resulting from different intake port configurations with or without a shrouded valve to discern the differences between swirl and tumble air motions. However, these studies described the port/valve geometry in a simple manner either to avoid expensive running or according to their conditions of computational capability at the time.

Our ultimate aim in using the method of computational fluid dynamics is to realise an improvement in engine design. To be a design tool, the computational model should be able to predict the observed trends with design parameters. The computational domain should represent actual engine geometry including port, valve and valve stem.

The present work is concerned with the turbulent flow characteristics in a single cylinder model engine, in which both steady and transient flow are considered. The induction process is through a single right angled port/valve assembly located asymmetrically to the cylinder axis. The cylinder head and the piston surface are flat. The computational
domain represents the actual geometry without any simplification, so as to obtain detailed information of the flow representative of that in the real engine. The prediction is realised through the use of the CFD code STAR-CD, in which the k-ε model is employed. The simulation results are validated experimentally.
Figure 2.1 Flow patterns through the valve passage (Tanaka, 1929)

Figure 2.2 Variation of velocity with valve lift (Tanaka, 1929)
Typical variation of discharge coefficient with lift/diameter ratio for an isolated, sharp-cornered inlet valve

(a) High lift
Free jet formed

(b) Intermediate lift

(c) Low lift
Jet fills gap

Patterns of flow through a sharp-cornered inlet valve

Figure 2.3 Flow patterns corresponding to different segments in the discharge coefficient curve

(Annand and Roe, 1974)
The radial locations for the measurements of the intake valve flow.

Axial profiles of radial mean and r.m.s. velocities, (relative to the valve centre), at location A, 
N = 1500 rev/min.

Figure 2.4(a) Measured velocity profiles at valve exit (Suen, 1992)
Figure 2.4 (b) Measured velocity profiles at valve exit (Suen, 1992).

Axial profiles of radial mean and r.m.s. velocities (relative to the valve centre), at location B.

N = 1500 rev/min.

Axial profiles of radial mean and r.m.s. velocities, relative to the valve centre, at location C.
Figure 2.5 Visualisation of the flow structures at different ratios of the valve lift to the valve diameter (Uzkan, 1983)
Figure 2.6 Effects of valve offset on the swirl level (Uzkan, 1983)

Figure 2.7 Effects of valve shape (Hunter, 1983)
Figure 2.8 Effects of port shape (Kastner et al, 1963, 1964)

Figure 2.9 Pressure distribution (Hardenberg and Daudel, 1975)
Figure 2.10 Effect of pressure drop (Woods et al, 1942)

Figure 2.11 Effect of pressure drop (Tanaka, 1929)
Figure 2.12 Computational domain (Jones, 1984)

Figure 2.13 Measured velocity profiles for inlet boundary conditions (Yamada et al, 1986)
Figure 2.14 Different port configurations (Isshiki et al, 1985)

Figure 2.15 Intake valve model (Isshiki et al, 1985)
Discharge coefficient

$L^*$ - ratio of valve lift to valve diameter

Figure 2.16 Discharge coefficient and flow pattern

(Gosman and Ahmed, 1987)
CHAPTER 3
MATHEMATICAL DESCRIPTION
AND NUMERICAL APPROXIMATION

3.1 MATHEMATICAL DESCRIPTION

3.1.1 Foundation

The physical laws governing fluid motion have been expressed in mathematical form, generally in terms of differential equations which describe the conservation of mass, momentum, and energy for non-isothermal flows. Each equation employs a certain physical quantity as its dependent variable. In the case of three-dimensional non-steady flow of a compressible viscous fluid, the flow field is specified by the velocity vector with three orthogonal components $u$, $v$, $w$, the pressure $p$, the density $\rho$, and temperature $T$, which are functions of the coordinates $x$, $y$, $z$ and time $t$.

There are six equations to determine these six variables: the continuity equation for conversation of mass, the three equations of motion for conservation of momentum in the x, y, z directions, the thermodynamic equation of state, and an equation for the conservation of energy on the form of the First Law of Thermodynamics.

On the assumption that the fluid is Newtonian, the relation between the components of stress and those of the rate of strain is linear. The dynamic state of the fluid is described by the fundamental equations:
• Momentum conservation

All three equations for the x, y, and z directions are similar in form, hence only the equations in the x direction are shown below.

Momentum equation in the x direction

\[
\frac{d(\rho u)}{dt} = X - \frac{\partial p}{\partial x} - \frac{\partial \sigma_x}{\partial x} - \frac{\partial \tau_{xy}}{\partial y} - \frac{\partial \tau_{xz}}{\partial z}
\]  

(3-1)

where,  
- \(X\) - body force component in x direction  
- \(\sigma_x\) - normal stress in x direction  
- \(\tau_{xy}\) - shearing stress parallel to y direction  
- \(\tau_{xz}\) - shearing stress parallel to z direction

\[
\sigma_x = -2\mu \frac{\partial u}{\partial x} + \frac{2}{3}\mu \left( \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} \right)
\]

\[
\tau_{xy} = -\mu \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right)
\]

\[
\tau_{xz} = -\mu \left( \frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \right)
\]

where, \(\mu\) stands for the laminar viscosity. Then, the momentum equation in the x direction will be

\[
\frac{\partial (\rho u)}{\partial t} + \frac{\partial (\rho u^2)}{\partial x} + \frac{\partial (\rho uv)}{\partial y} + \frac{\partial (\rho uw)}{\partial z} =
\]

\[
X - \frac{\partial p}{\partial x} + \frac{\partial}{\partial x} \left\{ \mu \left[ 2 \frac{\partial u}{\partial x} - \frac{2}{3} \left( \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} \right) \right] \right\} + \frac{\partial}{\partial y} \left[ \mu \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) \right] + \frac{\partial}{\partial z} \left[ \mu \left( \frac{\partial w}{\partial z} + \frac{\partial u}{\partial x} \right) \right]
\]  

(3-2)
equation (3-2) is referred to as the Navier-Stokes equation.

In Cartesian tensor notation, the Navier-Stokes equations for the x, y and z directions are:

\[
\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_j} = -\frac{\partial \rho}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial u_k}{\partial x_k} \right) \right] + s_i \\
(i, j, k=1, 2, 3)
\]

(3-3)

The term \(s_i\) contains the body forces. The Kronecker delta \(\delta_{ij}=0\) for \(i \neq j\); \(\delta_{ij}=1\) for \(i=j\).

• Conservation of mass

\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} + \frac{\partial (\rho v)}{\partial y} + \frac{\partial (\rho w)}{\partial z} = 0
\]

(3-4)

in tensor notation:

\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = 0
\]

(3-5)

• Equation of state

\[
\rho = \rho \varphi(p, T)
\]

(3-6)

where, \(T\) is temperature. For a perfect gas, \(\varphi\) becomes \(1/RT\). \(R\) denotes the universal gas constant.
The temperature is related to the total energy $e$. It is the energy equation to draw up a balance between heat and mechanical energy, and to furnish a differential equation for the temperature distribution.

**Energy conservation**

$$
\frac{\partial (\rho e)}{\partial t} + \frac{\partial (\rho u_i e)}{\partial x_j} = \frac{\partial q_i}{\partial x_j} - \frac{\partial (\rho u_i)}{\partial x_j} - \frac{\partial (\tau_{ij} u_j)}{\partial x_j} + Q
$$

where, $e$ - energy $de = c_v dT + \left(\frac{\partial e}{\partial v}\right)_T dv$

- $c_v$ - specific heat at constant volume
- $q_i$ - diffusive flux of energy
- $Q$ - external source of energy

**Temperature equation**

$$
\frac{\partial (\rho T)}{\partial t} + \frac{\partial (\rho u_i T)}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \Gamma_T \frac{\partial T}{\partial x_j} \right) + \rho s_T
$$

The above derivation has yielded six equations for the six variables: $u$, $v$, $w$, $p$, $\rho$, $T/e$. The equation system is closed. However, they are non-linear, coupled, and second order partial differential equations. An analytic solution is only possible for simplified problems. In general a numerical finite difference solution is required.

### 3.1.2 The general differential equation

All the dependent variables obey the conservation law, which can be written in general form:
\[
\frac{\partial (\rho \phi)}{\partial t} + \frac{\partial (\rho u \phi)}{\partial x} + \frac{\partial (\rho v \phi)}{\partial y} + \frac{\partial (\rho w \phi)}{\partial z} = \\
\frac{\partial}{\partial x} (\Gamma_\phi \frac{\partial \phi}{\partial x}) + \frac{\partial}{\partial y} (\Gamma_\phi \frac{\partial \phi}{\partial y}) + \frac{\partial}{\partial z} (\Gamma_\phi \frac{\partial \phi}{\partial z}) + s_\phi
\] (3-9)

where, \( \phi \) stands for the dependent variable, \( \Gamma_\phi \) is the diffusion coefficient, and \( S_\phi \) is the source term. The flow field should satisfy mass conservation.

The governing transport equations in Cartesian tensor notation, for momentum and continuity, are:

\[
\frac{\partial (\rho \phi)}{\partial t} + \frac{\partial (\rho u \phi)}{\partial x} = \frac{\partial}{\partial x_j} \left( \Gamma_\phi \frac{\partial \phi}{\partial x_j} \right) + s_\phi
\] (3-10)

\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_j)}{\partial x_j} = 0
\] (3-11)

In turbulent flow conditions, transport of momentum and other properties are dominated by turbulent mixing rather than molecular transport due to viscosity, unless the mean flow Reynolds number is sufficiently low or regions close to solid boundaries are considered. The viscosity plays a role in dissipating the turbulence energy generated by non-linear instability which leads to the formation of large-scale eddies. Such a chaotic random state of a turbulent flow can be described only in statistical terms.

3.1.3 Averaged equations

Turbulence is a random phenomenon which shows a quasi-permanency
and quasi-periodicity both in time and in space: at a given point in the
turbulent domain a distinct pattern is repeated which is more or less
regular in time; at a given instant a distinct pattern is repeated which is
more or less regular in space; so turbulence, broadly speaking, has the
same over-all structure throughout the domain considered. Therefore,
statistically distinct average values can be discerned. The instantaneous
properties could be decomposed into the sum of the mean and
fluctuating components.

For example, the instantaneous velocity $u$ is represented as

$$u_i = U + u'_i \quad (i=1, 2, 3)$$

(3-12a)

with $\overline{u'_i} = 0$

where, the capital letter indicates mean velocity, and the lower-case
letter with single quotation marks signifies fluctuating velocity. The
overbar stands for averaging processes.

The instantaneous pressure $p$, then the general dependent variable $\phi$ can
be represented in a similar manner:

$$p = P + p'$$

(3-12b)

$$\phi = \phi + \phi'$$

(3-12c)

The time- or ensemble-average, the most common statistical approach,
is adopted to describe and capture the mean characteristics of the
turbulent flow.
The time-average is applied if the flow is statistically steady or stationary. The mean velocity $\bar{U}$ and the fluctuating velocity $u'$ are defined as:

$$U = \lim_{\Delta t \to \infty} \frac{1}{\Delta t} \int_{t_0}^{t_0 + \Delta t} u \, dt$$

$$u' = \lim_{\Delta t \to \infty} \left( \frac{1}{\Delta t} \int_{t_0}^{t_0 + \Delta t} u'^2 \, dt \right)^{1/2}$$

(3-13a) (3-13b)

In periodic flows, the average value of the instantaneous velocity is replaced by ensemble-averaging, which is defined as the average of values at a specific phase in the basic cycle.

$$U(\theta) = \frac{1}{N_c} \sum_{i=1}^{N_c} u(\theta, i)$$

$$u'(\theta) = \left\{ \frac{1}{N_c} \sum_{i=1}^{N_c} u'(\theta, i) \right\}^{1/2}$$

(3-14a) (3-14b)

where,
- $\theta$ - a specific crank angle position
- $i$ - a particular cycle
- $N_c$ - the number of cycle

Substituting equation (3-12) into the Navier-Stokes equations (3-3), the mean momentum equations become:

$$\frac{\partial (\rho U_i)}{\partial t} + \frac{\partial (\rho U_j U_i)}{\partial x_j} =$$

$$- \frac{\partial \rho}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial u_k}{\partial x_k} \right) - \rho u_i' u_j' \right] + S_i$$

(i, j, k=1, 2, 3),

(3-15)
governing the mean velocity field.

where, $u'_i u'_j$, the additional terms arising from this operation are called Reynolds stresses, representing a diffusive transfer of momentum due to turbulent motion.

An analogous approach to equation (3-10) gives rise to the equation

$$
\frac{\partial (\rho \phi)}{\partial t} + \frac{\partial (\rho U_j \phi)}{\partial x_j} = \frac{\partial}{\partial x_j} (\Gamma \frac{\partial \phi}{\partial x_j} - \rho u'_j \phi') + S_{\phi} \tag{3-16}
$$

where, $u'_j \phi'$ can be interpreted as scalar fluxes, standing for scalar property due to the turbulent motion.

The task of expressing the terms of the mean properties of the flow are undertaken using a model for the turbulence phenomena.

3.1.4 Turbulence model

A turbulence model is a scheme for evaluating the six independent Reynolds stresses $-\rho u'_i u'_j$, the turbulent scalar fluxes $-\rho u'_i \phi'$, the missing information, and establishing the relationship between them to the mean flow from the averaged equations.

Because turbulent flows have an extremely complicated nature, all the turbulence models are developed with the aid of semi-empirical hypotheses to deduce the still missing fundamental physical ideas from results of experimental measurements, aiming at complete generality - applicability to any turbulent flow.
The eddy-viscosity (Boussinesq, 1877) and mixing-length (Prandtl, 1925) hypotheses are two ideas of a connection between the Reynolds stress and the local mean velocity gradient. Boussinesq introduced eddy or turbulent viscosity $\mu_t$, by which the Reynolds stresses are modelled as proportional to mean deformation rates.

$$-\rho \overline{u'_i u'_j} = \mu_t \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} \rho k \delta_{ij} \quad (3-17)$$

where, $k$ is turbulent kinetic energy, $k = \frac{1}{2} \overline{u_i'^2}$. \hspace{1cm} (3-18)

Prandtl proposed to describe the eddy viscosity $\mu_t$ through a turbulent velocity scale $v$ and a turbulent length scale $\ell$.

$$\mu_t = C_\mu \rho v \ell \quad (3-19)$$

where, $C_\mu$ is a constant.

The manner in which the characteristic parameters $v$ and $\ell$ are determined distinguishes one model from the others. Zero-equation models, one-equation models, and two-equation models are different classes of the eddy viscosity models.

Zero-equation models use only the partial differential equations for the mean velocity field, and no turbulence partial differential equations; One-equation models involve an additional partial differential equation relating to the turbulence velocity scale; Two-equation models incorporate an additional partial differential equation related to a turbulence length scale. The two-equation models have been found to
give satisfactory predictions in a wide variety of situations including both free and wall-bounded flows, provided that effects arising from curvature and buoyancy-related body forces are insignificant. The k-ε model is one of the two-equation models.

### 3.1.4.1 The k-ε model

Two-equation models, in which both velocity and length scales follow from transport equations for k and a length scale related parameter \( k^m \ell^n \), and form the eddy-viscosity models. There are various proposals, namely k-k' \( = k/\ell^2 \), k-w \( = k^{3/2}/\ell \), and k-ε \( = k^{3/2}/\ell \), for the form \( k^m \ell^n \).

The k-ε model employs the turbulence energy dissipation rate \( \varepsilon \), defined by \( \varepsilon = \nu \left( \frac{\partial u_i'}{\partial x_j} \right) \left( \frac{\partial u_j'}{\partial x_i} \right) \), as the second dependent variable, which does not require additional near-wall correction terms and can be directly derived, as an unknown in an exact transport equation, from the Navier-Stokes equations. Therefore, within the two-equation models group, the k-ε model appears the most popular one. It forms a good compromise between generality and economy of use for many engineering problems.

The standard form of the k-ε model is:

\[
\mu_t = C_{\mu} \rho k^2 / \varepsilon \quad (3-20)
\]

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho U_j k)}{\partial x_j} = \\
\frac{\partial}{\partial x_j} \left( \mu_t \frac{\partial k}{\partial x_j} \right) + \mu_t \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \frac{\partial U_i}{\partial x_j} - \rho \varepsilon \quad (3-21)
\]
\[
\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho U_j \varepsilon)}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \frac{\mu_t}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_j} \right) + C_{\varepsilon_1} \frac{\varepsilon}{k} P_k - \rho C_{\varepsilon_2} \frac{\varepsilon^2}{k} \quad (3-22)
\]

where, \( P_k \) is the production rate of turbulence energy giving by

\[
P_k = -u_i u_j \frac{\partial U_i}{\partial x_j} = \mu_t \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \frac{\partial U_i}{\partial x_j} \quad (3-23)
\]

The constants in the above equations are recommended by Launder and Spalding (1972) as follows:

\[\{C_\mu, C_{\varepsilon_1}, C_{\varepsilon_2}, \sigma_k, \sigma_\varepsilon\} = \{0.09, 1.44, 1.92, 1.0, 1.3\}\]

The \( k-\varepsilon \) model is the high Reynolds number version, valid only in regions where the flow is fully turbulent.

Across the wall layer, the flow undergoes a transition from fully turbulent to complete laminar within the thin viscosity dominated sub-layer adjacent to the solid surface. The high Reynolds number version of the \( k-\varepsilon \) model can not account for this process. Special steps need to be taken to provide wall boundary conditions in \( k-\varepsilon \) computations.

The standard \( k-\varepsilon \) model employs "wall functions" to relate the near wall layer to the flow properties in the fully turbulent region. Some assumptions on which the wall functions are based include following:

1) Variations in velocity and other variables such as temperature, are predominantly normal to the wall, leading to one-dimensional behaviour.
2) The distribution of the shear stress across the layer is uniform.

3) Linear variation of the length-scale.

4) The rate of turbulence energy production $P_k$ balances the rate of dissipation $\varepsilon$.

The velocity profiles across the layer are expressed in the well-known logarithmic law of the wall:

$$U^+ = \frac{1}{\kappa} \ln(EY^+)$$

(3-24)

$$U^+ = \rho C_{\mu}^{1/4} k^{1/2} U / \tau_w$$

(3-25)

$$Y^+ = C_{\mu}^{1/4} Y k^{1/2} / \nu$$

(3-26)

where, $\tau_w$ is wall shear stress, $\kappa$ and $E$ are empirical coefficients, $Y$ is the distance of the node, which is at the outer-edge of the boundary layer, from the solid surface.

During the solution of the $k$-transport equation at the near-wall cell, $P_k$ and $\varepsilon$ are obtained by integration across the cell. The near-wall dissipation is prescribed as

$$\varepsilon = C_{\mu}^{3/4} k^{3/2} / \kappa Y$$

(3-27)
3.1.4.2 Limitations of the k-ε model for predicting the intake flow and in-cylinder flow

In the intake flow and in-cylinder flow, separation occurs due to shear layers, and predictions with the k-ε model in these circumstances cause a number of inaccuracies arising from adverse pressure gradients, favourable pressure gradients, significant streamline curvature, and sharp-edged geometries. In these situations, the generation of different Reynolds stresses are highly non-uniform.

The defect of the k-ε model is its inability to properly capture anisotropy by accounting for the production, redistribution, convection and diffusion of each stress separately.

In the bend of the curved inlet port, and near the port exit, both convex walls and concave walls exist. On the convex walls, the stabilising effect lowers the Reynolds shear stresses and turbulence energy levels. The decrease in turbulence is associated with a corresponding decrease in static pressure in the flow direction and hence acceleration of the flow. A destabilizing effect appears at concave walls, resulting in unusually high levels of Reynolds shear stresses and turbulence kinetic energy. As a consequence of the secondary motion set up in the bend, there is an interchange of turbulence energy between the convex and concave walls, resulting in a highly anisotropic and complex pattern of stresses.

Accurate representation of the stress distribution will require modelling based on solutions of the Reynolds stress equations. The evidence is
shown in Gibson and Rodi's work (1981). They employed a Reynolds stress model to calculate the flow through a highly curved mixing layer for which measurements were reported by Castro and Bradshaw (1976). The most spectacular feature of the measurements is that the Reynolds stress and other turbulent quantities decrease in the region of high stabilising curvature, rise rapidly further downstream and overshoot the plane-layer values before finally decreasing. Gison and Rodi compared the solutions of turbulent energy $q^2$ and shear stress $u'v'$ between the measurements and the predictions obtained by both the Reynolds stress model and the k-ε model. The Reynolds stress model produced a 13% overshoot of the plane-layer values in the turbulent energy and 18% in the shear stress. The measured overshoots were 35% in the turbulent energy and 25% in the shear stress. The k-ε model predicted the streamwise variation of the turbulent energy well, but failed badly to reproduce the shear stress. Because of a constant coefficient 0.09, using in the eddy-viscosity formula, the k-ε model failed to reproduce the observed fall and recovery in the structure parameter $u'v' / q^2$, and predicted an approximately constant value. The Reynolds stress model did predict the fall of the structure parameter $u'v' / q^2$ to the right level, but overshot the measured values downstream, mainly due to the underprediction of the turbulent energy $q^2$.

The comparisons showed that a full Reynolds stress transport closure can reproduce the effects of curvature more satisfactorily than the standard k-ε model. It implies that the fundamental defects of the k-ε model are rooted in a basic assumption embodied in the model, namely isotropy of turbulent exchange.
In order to account, phenomenologically, for the effect of streamline curvature on turbulence transport without recourse to the Reynolds stress model, attempts have been made to modify the k-ε model by terms dependent on a dimensionless parameter which is named by either a gradient- or flux- Richardson number to characterise the effects of body force and streamline curvature on the turbulence. The gradient-Richardson number (Bradshaw, 1969) is the ratio of body force to inertia force. The flux-Richardson number (Bradshaw, 1973) is the ratio of the generation of turbulence energy by an extra body force (centrifugal or buoyancy) to the total production of turbulence energy. These terms modify either the eddy viscosity directly or the turbulence length scale. All these modifications are restricted to the specific situations for which they were developed.

The flow in a region of streamline curvature is governed primarily by pressure gradient effects. When the flow enters the cylinder, it is over a sharp-edged bluff body. Separations and subsequent re-attachment occur with accompanying large variations in velocity and pressure around the detachment location. Adverse pressure gradients are responsible for the separations.

The standard k-ε model does not predict well the behaviour of shear layers subject to adverse pressure gradients. Rodi and Scheurer (1986) showed that the k-ε model overpredicted the velocity in the log-law region and outer part of the boundary layer. It is caused by the behaviour of the ε equation. The k-ε model gives a consistently higher skin friction coefficient for the boundary layers with adverse pressure gradient, because of too steep an increase of the turbulent kinetic energy
k and consequently the shear stress $\overline{u'v'}$. This would have an effect in the prediction of the jet flow into the cylinder and consequently the recirculation regions caused by the separations of the jet from the valve seat and the valve head at higher valve lifts, and also the separations in the wake of the valve stem, the pressure drop would be overpredicted.

Near the port exit, a favourable pressure gradient causes strong flow accelerations, particularly at lower valve lifts. These effects can not be reproduced by the standard $k-\varepsilon$ model either. Patel et al (1984) demonstrated the calculated velocity profiles for a favourable pressure gradient boundary layer by four models, in which three are variants of the $k-\varepsilon$ group: LS - Launder and Sharma (1972), CH - Chen (1982), LB1 - Lam and Bremboret (1981); and the other is WR - Wilcox, Rubesin (1980).

The variants of the $k-\varepsilon$ model modify the empirical constants C in the basic $k-\varepsilon$ model version by the viscous diffusion terms and functions, also extra terms D and E in some cases to represent the near-wall behaviour better.

The relevant equations of the models are:

$$\frac{Dk}{Dt} = \frac{\partial}{\partial x_j} \left[ \left( \frac{v_i}{\sigma_k} + v_{\varepsilon} \right) \frac{\partial k}{\partial x_j} \right] + v_i \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \varepsilon$$  \hspace{1cm} (3-28)

$$\frac{D\varepsilon}{Dt} = \frac{\partial}{\partial x_j} \left[ \left( \frac{v_i}{\sigma_{\varepsilon}} + v_{\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{\varepsilon_1} f_1 v_i \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - C_{\varepsilon_2} f_2 \frac{\tilde{\varepsilon}}{k} + E$$  \hspace{1cm} (3-29)
\[ v_t = C_\mu f_\mu \frac{k^2}{\bar{\varepsilon}} \]  

(3-30)

where, \( \varepsilon = \bar{\varepsilon} + D \), \( R_T = k^2 / \nu \bar{\varepsilon} \), and \( R_y = \sqrt{k_y / \nu} \).

LS and CH use \( \bar{\varepsilon} \) as the "dissipation variable", which is originally adopted by Jones and Launder (1973). The term \( D \) is employed to allow \( \bar{\varepsilon} = 0 \) at the wall, and \( \bar{\varepsilon} = \varepsilon \) in the fully turbulent regime, in order to keep the kinetic energy equation in balance. LB adopt \( \varepsilon \) itself, the wall value \( \varepsilon_w \) is calculated using the boundary condition \( \frac{\partial \varepsilon}{\partial y} = 0 \). The five empirical constants \( C_\mu, C_{e_1}, C_{e_2}, \sigma_k \), and \( \sigma_\varepsilon \), functions \( f_\mu, f_1 \) and \( f_2 \), and the extra terms \( D \) and \( E \), are listed in Table 3.1 - 3.3.

Table 3.1 Constants in various k-\( \varepsilon \) models

<table>
<thead>
<tr>
<th>Code</th>
<th>( C_\mu )</th>
<th>( C_{e_1} )</th>
<th>( C_{e_2} )</th>
<th>( \sigma_k )</th>
<th>( \sigma_\varepsilon )</th>
</tr>
</thead>
<tbody>
<tr>
<td>LS</td>
<td>0.09</td>
<td>1.44</td>
<td>1.92</td>
<td>1.0</td>
<td>1.3</td>
</tr>
<tr>
<td>CH</td>
<td>0.09</td>
<td>1.35</td>
<td>1.80</td>
<td>1.0</td>
<td>1.3</td>
</tr>
<tr>
<td>LB</td>
<td>0.09</td>
<td>1.44</td>
<td>1.92</td>
<td>1.0</td>
<td>1.3</td>
</tr>
</tbody>
</table>

Table 3.2 Functions in various k-\( \varepsilon \) models

<table>
<thead>
<tr>
<th>Code</th>
<th>( f_\mu )</th>
<th>( f_1 )</th>
<th>( f_2 )</th>
</tr>
</thead>
<tbody>
<tr>
<td>LS</td>
<td>( \exp\left(\frac{-3.4}{(1 + R_T / 50)^2}\right) )</td>
<td>1.0</td>
<td>( 1 - 0.3 \exp\left(-R_T^2\right) )</td>
</tr>
<tr>
<td>CH</td>
<td>( 1 - \exp\left(-0.0115Y^+\right) )</td>
<td>1.0</td>
<td>( -2\nu \bar{\varepsilon} / Y^2 \exp\left(-0.5Y^+\right) )</td>
</tr>
<tr>
<td>LB</td>
<td>( \frac{\left[1 - \exp\left(-0.0165R_T\right)\right]^2}{x(1 + \frac{20.5}{R_T})} )</td>
<td>( 1 + (0.05 / f_\mu)^3 )</td>
<td>( 1 - \exp\left(-R_T^2\right) )</td>
</tr>
</tbody>
</table>
Table 3.3 Terms in various k-ε models

<table>
<thead>
<tr>
<th>Code</th>
<th>Code</th>
<th>Code</th>
</tr>
</thead>
<tbody>
<tr>
<td>LS</td>
<td>2ν(∂k/∂y)²</td>
<td>2νν₁(∂²U₁/∂y²)²</td>
</tr>
<tr>
<td>CH</td>
<td>2νk/y²</td>
<td>-2ν(ε/y²)exp(-0.5y⁺)</td>
</tr>
<tr>
<td>LB</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

WR employ an equation for the kinetic energy of the normal velocity fluctuations, together with a transport equation for a pseudovorticity \( \omega \).

\[
\frac{Dk}{Dt} = \frac{\partial}{\partial x_j} \left[ (\nu + \frac{\nu_1}{\sigma_k}) \frac{\partial k}{\partial x_j} \right] + \nu_1 \left( \frac{\partial U_i}{\partial x_i} + \frac{\partial U_j}{\partial x_j} \right) - C_k k \omega \tag{3-31}
\]

\[
\frac{D\omega^2}{Dt} = \frac{\partial}{\partial x_j} \left[ (\nu + \frac{\nu_1}{\sigma_\omega}) \frac{\partial \omega^2}{\partial x_j} \right] + C_\omega f_1 \omega \left( \frac{\partial U_i}{\partial x_i} + \frac{\partial U_j}{\partial x_j} \right) \frac{\partial U_i}{\partial x_j} \tag{3-32}
\]

\[
- C_{\omega_2} \omega^3 + E
\]

\[
\nu_1 = f_\mu \left( k / \omega \right) \tag{3-33}
\]

where, \( \ell = \sqrt{k} / \omega \), and \( R_T = \sqrt{k} \ell / \nu \).

Similar functions are used to modify the constants in WR. Table 3.4 summarized the constants and functions.

Table 3.4 Constants, functions and term E for WR model

<table>
<thead>
<tr>
<th>C_µ</th>
<th>C_α₁</th>
<th>C_α₂</th>
<th>σ_k</th>
<th>σ_α</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.09</td>
<td>1.11</td>
<td>0.15</td>
<td>2.0</td>
<td>2.0</td>
</tr>
</tbody>
</table>

f_µ | 1 - 0.992exp(-R_T) |

f_1 | 1 - 0.992exp(-R_T/2) |

E | \(-2\frac{2}{\sigma_\omega} (\frac{\partial \ell}{\partial y})^2 \omega^3\) |
Figure 3.1 shows the comparisons between the predictions and the measurements obtained by Simpson and Wallace (1975) for the favourable pressure gradient boundary layer in which the acceleration parameter was nearly constant and the acceleration was moderate. The predicted mean velocity rises above the usual logarithmic profile reflecting the thicker viscous region, a behaviour that is imitated by all the models. The profile of the usual law of the wall showed a departure from the measurement. This indicates that the pressure drop in the acceleration region would be underpredicted by wall functions.

The defects of the standard k-ԑ model discussed above will cause discrepancies between predictions and measurements in the present study for the intake and in-cylinder flows. At lower valve lifts, the flow rapidly accelerates in the port and particularly when it is approaching and entering the port exit. Separation in the wake of the valve stem is weak. Recirculations in the cylinder are not strong. Predictions with the k-ԑ model will show lower magnitudes of mean velocities and lower pressure drops than the measured ones. At higher valve lifts, the jet through the valve produces sharp shear layers off the valve seat and the valve head to set up large-scale recirculations. The separation in the wake of the valve stem is also distinct. Adverse pressure gradients will be associated with these separations. The k-ԑ model will overpredict the pressure drops. All these effects will be seen in the results shown in Chapter 5.
3.2 NUMERICAL APPROXIMATION

3.2.1 Introduction

The partial differential equations (3-15 and 3-16) which describe turbulent flow, are generally non-linear, and coupled - the continuity and momentum equations are linked together when the pressure is chosen as a main dependent variable rather than the density. The pressure should be retained as a main dependent variable in the equations because the methods regarding the density as the working variable is inapplicable to incompressible flows or the flow field at very low Mach numbers. Then, the calculation of the velocity field lies in an unknown pressure field which is indirectly specified via the continuity equation. The pressure gradient forms a part of the source term for the momentum equation, and the resulting velocity field satisfies the continuity equation. Because of the nonlinearity and the pressure-velocity coupling, there is no direct solution for the equations, a numerical solution method is then required.

The process of numerically simulating a fluid flow problem includes three procedures, namely grid generation, discretization and a solution algorithm.

A variety of methods may be adopted to achieve discretization, of which the finite-difference, finite-volume and finite-element methods are the ones most frequently used to divide a physical domain into a sequence of subdomains known as a nodal mesh and define approximations to a continuous solution at the nodal points.
The finite-element method was initially developed to calculate stress in irregularly shaped objects and analyse structural problems, and is finding increased levels of application in fluid mechanics. In finite-element methods, a partial differential equation is reduced to a finite system of ordinary differential equations, which is then solved by matrix solution techniques. Reduction of the governing equation is based on a variational principle or the Galerkin technique which is a special case of the method of weighted residuals. Due to unstructured matrices arising from the unstructured grids used, instability of computation may be caused, and an expensive direct simultaneous solution for velocity components and pressure is required. Three-dimensional finite-element methods are very costly and storage is large.

The main advantage of the finite-element method is unlimited geometric adaptability in fitting irregular domains and in providing local grid refinement. However, the difficulties such as mathematical complexity, expensive direct solution and stability problems, have hindered its progress.

Finite-difference methods are mainly based on the assumption that truncated Taylor Series expansions of the spatial derivatives yield adequate approximations to differential equations. In the finite-difference method, the problem domain is replaced by discrete nodes. The variation of a dependent variable is approximated by a polynomial fit in the vicinity of any one node on the mesh thus satisfying the laws of transport at these discrete nodes, without any explicit reference as to how the dependent variable varies between the nodes.
The finite-difference method and finite-volume method are closely akin. The finite-volume method considers the domain as discretized into contiguous cells or control-volumes each surrounding a nodal point (see Figure 3.2). In the finite-volume method, the discrete nature of the finite-difference method is recognised and the conservation principle described by the partial differential equations is enforced on control volumes. The partial differential equation is integrated over any one finite volume to give an integro-differential equation. The Gauss-divergence theorem is used to express volume-integrals as surface-integrals. The volume- and surface-integrals are evaluated by the variation of a dependent variable over volume and its surface, which is approximated by a polynomial fit to values at nodes placed within volumes. Therefore, the finite-volume method can obtain a solution which would imply that the integrated conservation of quantities such as mass, momentum, and energy is exactly satisfied over any group of control volumes and over the whole calculation domain.

For the present research, the differential equations governing the conservation of mass, momentum, energy etc. are discretized by the finite-volume method, in which, they are first integrated over the individual computational cells and then approximated in terms of the cell-centred nodal values of the dependent variables.

For the purpose of the finite-volume discretization, it is convenient to describe all mean flow by a common form of transport equation:

$$\text{div} (\rho \bar{U} \phi - \Gamma_\phi \text{grad} \phi) = S_\phi$$

(3-34)
where $U$ is the velocity vector; $\phi$ stands for any of the dependent variables; and $\Gamma_\phi$ is the diffusive flux of $\phi$; $S_\phi$ is the net source of $\phi$.

### 3.2.2 The finite-volume discretization

#### 3.2.2.1 Finite volume equations

The finite volume formulation may be interpreted as a macroscopic application of the conservation principle to finite sub-regions of the computational domain. The procedure is independent of the coordinate system. For the sake of generality, the more convenient symbolism of the Cartesian system is employed comprising the coordinates $(x, y, z)$ with corresponding velocity components $U, V, W$.

Reference to the typical cell shown in Figure 3.3, the exact integral form of equation (3-34) is:

$$
\sum_m (\rho \bar{U} \phi - \Gamma_\phi \text{grad } \phi)_m \bar{S}^i_m = S_\phi V_p
$$

(3-35)

where, the summation is over the six faces of the cell (i.e. $m = e, w, n, s, t, b$). The expression in the brackets $(\quad)_m$ represents the total flux through faces $m$ due to convection and diffusion; and the $\bar{S}^i_m$ are surface vectors normal to the cell faces and equal in magnitude to their areas. The quantity $S_\phi$ is the average source over the cell, and $V_p$ is the cell volume.
3.2.2.2 Finite differencing schemes

Discretization is preceded by an integration of equation (3-35) over a cell to yield an integro-differential equation expressing a balance between convection and diffusion fluxes through cell faces and a cell-volume-integrated source. The cell face fluxes contain unknown cell-face values (convection), and their derivatives (diffusion), which necessitates the use of interpolation functions. In recent years, a significant amount of research effort has been directed at discretization of the combined convection and diffusion fluxes, such as the Central-differencing scheme, Up-wind scheme, Blended differencing scheme and Quadratic upstream-weighted interpolation scheme, i.e. Quick, (Leonard, 1979).

The up-wind differencing scheme is a first order scheme by which errors decrease linearly with grid spacing. The truncation error term acts as an additional numerical diffusion smearing sharp gradients of the solution.

The central differencing scheme is formally second order accurate, errors decrease by a factor of 4 when grid spacing is reduced by half. In order to obtain a bounded solution, very fine grids are required, which is too expensive for practical calculations. In the linear upwind differencing scheme, which is second order accurate, the value of the control volume face is obtained by linear extrapolation of upstream nodal values. Self-filtered central differencing and blended differencing are the schemes which combine a high-order formulation with a strategy for suppressing spatial oscillations, usually by detecting their onset and
them locally modifying the discretisation in a suitable way. The local modification is almost invariably a form of reversion to a lower-order scheme, either wholly or partially.

The quick scheme is third order accurate. It combines the accuracy of quadratic interpolation with the stability of upstream weighting. It uses an upstream based parabola to approximate the variation of the dependant variable. Nodal values from downstream appear in the equation for the face value.

The central-differencing scheme, up-wind differencing scheme, self-filtered central scheme and blended differencing scheme are implemented as alternative schemes in the STAR code, which will be introduced in the next chapter.

Patankar (1988) pointed out that lower-order schemes such as the upwind scheme, are stable and monotonic, but lead to false diffusion; high-order schemes such as quick, eliminate false diffusion but produce small oscillations in the solution and often fail to converge.

Patel et al (1987) tested eleven discretization schemes in the elliptic convective flow and heat transfer of a supersonic jet exhausting into a cold subsonic free stream. Only five of the schemes obtained convergence, such as the upwind scheme, the hybrid difference scheme (Spalding, 1972) and the upwind-in-streamline-direction scheme (Patel et al,1985) etc. The quick and the skew difference scheme (Raithby, 1976) failed to converge despite their best effects with various combinations of relaxation parameters and procedures. Between the
results of velocity and temperature obtained by the converged schemes, the differences were less than 10% and consistently less than the differences introduced by changing one of the turbulence model constants. The upwind scheme required 10% less CPU time than the upwind-in-streamline-direction scheme to achieve the same degree of convergence. Patel et al concluded that for high shear, high velocity problems, even in the presence of recirculations, the upwind scheme is the best one. It is stable, simple to understand and implement.

Vanka (1987) investigated the performance of the linear upwind scheme and compared it with the first-order upwind scheme. He observed that the second-order upwind scheme did not perform with great superiority to that of the first order variant. Shyy (1985) and Castro et al (1987) obtained contradictory conclusions. Huang et al (1985) found that all high-order schemes yielded unbounded solutions to produce the largest oscillations and performed badly in cavity flows. As described by Leschziner (1989), the tendency of these schemes to produce unphysical oscillations at high cell Peclet numbers is generally perceived as being the schemes' most serious limitation and disadvantage.

The central difference scheme can be used without fear of instability and significant wiggles. However, it is very expensive for the bounded solution and not suitable for convection dominated flow (i.e. with high grid Peclet number, $p_e = \rho U \delta x / \Gamma$, which is the ratio of the strengths of convection and diffusion). For high Peclet number flows, it performs far less satisfactorily than the first-order upwind scheme.
In the present study, the intake jet flow enters into the cylinder at high velocities (61 m/sec is the maximum velocity for steady flows, 238 m/sec for the transient flow at 3600 rpm engine speed), and separates from the valve seat and the valve lip with high shear at the edges of the jet. The flows have high grid Peclet numbers which are almost always larger than 10. For a stable, realistic and converged solution, the upwind scheme is chosen.

3.2.2.3 Solution algorithm

As mentioned above, the calculation of the flow field requires techniques to handle coupling between the momentum and continuity equations. Many schemes have been devised for the solution of the linearly coupled system in pressure and velocity. There are two groups of these methods: semi-implicit schemes and full implicit schemes.

In the first group, such as the MAC method (Harlow and Welch, 1965) and the SMAC method (Amsden and Harlow, 1970), the momentum equations are discretized explicitly, but the continuity relation can still be satisfied in an implicit manner. These methods based in the given velocity field at time k, the pressure equation is solved to get pressure at time k+1. Using the new pressure, the new velocities are solved from the explicit momentum equations. Then, other transport equations are solved. With the time advance, the procedure is repeated with all the steps.

The explicit schemes determine the values of the variables at k+1 time directly from known values at k time level. They suffer from severe
limitations on the time step size for stability when steady state solution is required, and for temporal accuracy when transient flows are calculated, especially for flows with diffusion. Therefore, these methods are prohibitively expensive for steady flows and inefficient for transient flows.

The fully implicit schemes do not suffer from time step restrictions. All the equations including the momentum equations are fully implicitly discretized. The pressure equation is satisfied simultaneously with the momentum equations through the use of iteration. The iterative schemes guess the pressure field at time \( k+1 \), and solve the momentum equations to get the new velocities, by which the pressure equation is solved. Then the other equations are solved. The steps are repeated until the solution achieves convergence. The procedure can be repeated with time advance.

Prominent methods of the fully implicit schemes are SIMPLE (Patankar and Spalding, 1972), SIMPLER (Patankar, 1980, 1981), SIMPLEST (Spalding, 1980), and PISO (Issa, 1985) etc.

SIMPLE is the original procedure among these schemes. It solves the pressure-correction equation rather than the full pressure equation. The solution is obtained by its decomposition into predictor and corrector steps. It is mainly used for steady state calculations, operating in an iterative mode. Heavy under-relaxation is needed to achieve a stable solution and to accelerate the convergence. For complex flow situations with severe mesh distortion, SIMPLE has additional parameters which can be used to better effect to procure convergence.
SIMPLER is an improved version of SIMPLE. It solves an extra pressure equation for the evaluation of pressure. The pressure-correction equation is only for correcting the velocities. It does not use guessed pressures, a pressure field is extracted from a given velocity field. SIMPLER gives faster convergence than SIMPLE, but one iteration of SIMPLER involves more computational effort.

In SIMPLEST, Spalding recommended an explicit treatment of convection and implicit treatment of diffusion in the momentum equations.

PISO is another enhancement of SIMPLE. It is applicable to both transient and steady state calculations. For steady state calculations, it can perform in either a time-marching or iterative mode, single or multi-phase flows. PISO is outlined initially by application to the incompressible flow equations, and is then extended to the implicitly discretized compressible flow equations. Issa et al (1986) applied this method to predict the incompressible flow in a duct with sudden enlargement and the compressible flow in the same case with the exception that the downstream end was closed. They demonstrated that the PISO time-marching method is many times faster than its iterative counterpart for transient flow, whether compressible or incompressible. In addition, it is stable for large time-step sizes, hence making it efficient for steady state calculations as well as transient ones.

For the present intake and in-cylinder flow, the computational domain is very complex. Mesh distortions exist. Therefore, the SIMPLE algorithm is chosen for the steady state calculations, and the PISO method is used
for calculating transient flows. These two algorithms are available in
STAR-CD and will be introduced in detail in the next chapter.
Figure 3.1 Calculations of an accelerating turbulent boundary layer
(Patel et al, 1985)

Figure 3.2 Arrangement of cells
Figure 3.3 Cell node and face labelling convention
CHAPTER 4
STAR-CD

4.1 INTRODUCTION

STAR-CD is a computational program developed by Computational Dynamics Limited for the calculation of fluid flow, heat and mass transfer, and chemical reaction in industrial and environmental circumstances.

STAR operates by solving the governing differential equations of flow physics using the Finite-volume method. STAR is capable of simulating the turbulent flow in the internal combustion engine's inlet port and cylinder with these flow phenomena: steady and transient; incompressible and compressible; heat transfer and mass transfer. A highly flexible computational mesh system is employed to permit the complex port/valve shape and moving valve/piston to be included naturally in the calculation.

4.2 MATHEMATICAL MODELLING

4.2.1 Conservation equations

Considering a moving coordinate frame run, STAR describes the mass and momentum conservation equations corresponding to (3-1) and (3-2) as follows (Computational Dynamics Ltd., 1991):
Continuity

\[ \frac{1}{\sqrt{g}} \frac{\partial}{\partial t} (\sqrt{g} \rho) + \frac{\partial}{\partial x_j} (\rho \tilde{u}_j) = 0 \quad (4-1) \]

Momentum

\[ \frac{1}{\sqrt{g}} \frac{\partial}{\partial t} (\sqrt{g} \rho u_i) + \frac{\partial}{\partial x_j} (\rho \tilde{u}_j u_i + \tau_{ij}) = \frac{\partial p}{\partial x_i} + s_i \quad (4-2) \]

Where, \( \tilde{u}_j \) - \((u_j - u_{cj})\), relative velocity between the fluid and the local coordinate frame which moves with velocity \( u_{cj} \)

- \( u_i \) - absolute fluid velocity component in direction \( x_i \)
- \( x_i \) - cartesian coordinate (\( i=1,2,3 \))
- \( p \) - piezometric pressure = \( p_s + \rho_0 g_m x_m \), where \( p_s \) is static pressure, \( \rho_0 \) is reference density, the \( g_m \) are gravitational field components and the \( x_m \) are coordinates from a datum
- \( \rho \) - density
- \( \tau_{ij} \) - stress tensor components
- \( s_i \) - momentum source components
- \( \sqrt{g} \) - determinant of metric tensor
- \( t \) - time

For turbulent flow, all the dependent variables are assumed to be their ensemble averaged values and their fluctuations (i.e. \( u_i = U_i + u_i' \), \( \rho = \rho + \rho' \), \( p = P + p' \)). Then, the time averaging conservation equations are:

Continuity

\[ \frac{1}{\sqrt{g}} \frac{\partial}{\partial t} (\sqrt{g} \rho) + \frac{\partial}{\partial x_j} (\rho \tilde{U}_j) = 0 \quad (4-3) \]
Momentum
\[
\frac{1}{\sqrt{g}} \frac{\partial}{\partial t} (\sqrt{g} \rho U_i) + \frac{\partial}{\partial x_j} (\rho \ddot{U}_j U_i + \tau_{ij}) = \frac{\partial P}{\partial x_i} + s_i
\]  \hspace{1cm} (4-4)

Where, \( \tau_{ij} = -\mu (s_{ij} + \frac{2}{3} \frac{\partial U_i}{\partial x_j} \delta_{ij}) + \rho u'_i u'_j \)  \hspace{1cm} (4-5)

\( s_{ij} \) - the rate of strain tensor, \( s_{ij} = \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \)  \hspace{1cm} (4-6)

\( \overline{u'_i u'_j} \) - the additional Reynolds stress due to the turbulent motion. \( u' \) are fluctuations about the ensemble average velocity, and the overbar denotes the ensemble averaging process. \( \mu \) is the fluid viscosity; \( \delta_{ij} \) is the "kronecker delta", it is unity when \( i=j \), and zero when \( i \neq j \).

For heat transfer, STAR incorporates the energy conservation equation for a general fluid mixture, with equal diffusivities of mass and energy, at low Mach numbers (Jones, 1980). In the present work, there is a single fluid.

\[
\frac{1}{\sqrt{g}} \frac{\partial}{\partial t} (\sqrt{g} e) + \frac{\partial}{\partial x_j} (\rho \ddot{U}_j e - F_{e,j}) = -P \frac{\partial U_i}{\partial x_i} + \tau_{ij} \frac{\partial U_i}{\partial x_j} + s_e
\]  \hspace{1cm} (4-7)

where, \( e \) - specific internal energy,
\[
e = c_v T - c_{\nu_0} T_0 + \Sigma m_k H_k
\]  \hspace{1cm} (4-8)

\( T \) - temperature

\( m_k \) - mass fraction of mixture constituent \( k \)

\( H_k \) - heat of formation of constituent \( k \)

\( \Sigma \) - summation over all mixture constituents

\( c_v \) - mean specific heat constant volume and temperature

\( c_{\nu_0} \) - reference specific heat at temperature \( T_0 \)
$s_e$ - energy source

$F_{e,j}$ - diffusional energy flux in direction $x_j$

$$F_{e,j} = -\frac{\mu}{\gamma \sigma_h} \frac{\partial e}{\partial x_j} + \rho u'_j e'$$

$\sigma_h$ is the molecular Prandtl number, $\gamma$ is the ratio of the specific heat at constant pressure and volume.

For heat transfer at a wall, STAR allows the wall temperature to be specified either on the inside or outside surfaces of the enclosure walls.

### 4.2.2 Turbulence modelling

STAR contains three alternative turbulence representations to determine the Reynolds stresses and turbulent scalar fluxes: the k-$\varepsilon$ model (Launder and Spalding, 1974), the k-$\ell$ model (Reynolds, 1976), and the directly prescribed eddy viscosity (Schlichting, 1968).

The k-$\varepsilon$ model and k-$\ell$ model both belong to the eddy-viscosity type of model, and assume the turbulent Reynolds stresses and scalar fluxes to be linked to the ensemble averaged flow properties in an analogous fashion to their laminar flow counterparts.

$$\bar{\rho u'_i' u'_j} = -\mu_{t} s_{ij} + \frac{2}{3} (\mu_{t} \frac{\partial u'_i}{\partial x_j} + \rho \kappa) \delta_{ij}$$  \hspace{1cm} (4-9)

$$\bar{\rho u'_i e'} = -\frac{\mu_{t}}{\gamma \sigma_{h,t}} \frac{\partial e}{\partial x_j}$$  \hspace{1cm} (4-10)
where, $k$ - the turbulent kinetic energy. $k = \frac{\overline{u'_i u'_i}}{2}$

$\sigma_{h,t}$ - the turbulent Prandtl

The turbulent viscosity is linked to $k$ and via:

$$\mu_t = C_{\mu} \rho k^2 / \varepsilon$$  

or the $k$ and $\ell$ via:

$$\mu_t = C_{\mu}^{1/4} \rho k^{1/2} \ell$$

The relation between $k$, and $\ell$ can be obtained by equation (4-11) and (4-12):

$$\ell = C_{\mu}^{3/4} k^{3/2} / \varepsilon$$

The $k$-$\varepsilon$ model is employed in the present work. It is also specified as a preferred option most often in STAR-CD, on the basis of generality, cost effectiveness and availability of performance data. STAR uses its particular form for the $k$-$\varepsilon$ model, which is appropriate to fully turbulent, incompressible or compressible flows (El Tahry, 1983), and allows to some extent for buoyancy effects (Rodi, 1979).

In STAR, the turbulent energy balance is represented by
\[
\frac{1}{\sqrt{g}} \frac{\partial}{\partial t} (\sqrt{g} \rho k) + \frac{\partial}{\partial x_j} (\rho \bar{U}_j k - \mu_t \frac{\partial k}{\partial x_j}) = \mu_t (P + P_B) - \rho \varepsilon - \frac{2}{3} (\mu_t \frac{\partial U_i}{\partial x_j} + \rho k) \frac{\partial U_i}{\partial x_i} 
\]

(4-14)

where, \( P = s_{ij} \frac{\partial U_i}{\partial x_i} \)

\[ P_B = \frac{g_i}{\sigma_{n_t}} \frac{1}{\rho} \frac{\partial \rho}{\partial x_i} \]

\( \sigma_k \) - empirical coefficient.

The turbulence dissipation rate is represented by

\[
\frac{1}{\sqrt{g}} \frac{\partial}{\partial t} (\sqrt{g} \rho \varepsilon) + \frac{\partial}{\partial x_j} (\rho \bar{U}_j \varepsilon - \frac{\mu_t}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x}) = C_{\varepsilon_1} \frac{\varepsilon}{k} \left[ \mu_t (P + C_{\varepsilon_3} P_B) - \frac{2}{3} (\mu_t \frac{\partial U_i}{\partial x_j} + \rho k) \frac{\partial U_i}{\partial x_i} \right] - C_{\varepsilon_2} \rho \frac{\varepsilon^2}{k} + C_{\varepsilon_4} \rho \varepsilon \frac{\partial U_i}{\partial x_i}
\]

(4-15)

where, \( \sigma_\varepsilon, C_{\varepsilon_1}, C_{\varepsilon_2}, C_{\varepsilon_3}, \) and \( C_{\varepsilon_4} \) are empirical coefficients whose values taken from references (Lauder and Spalding, 1972, Rodi, 1979, El Tahry, 1983) are given below:

<table>
<thead>
<tr>
<th>( C_{\mu} )</th>
<th>( \sigma_k )</th>
<th>( \sigma_\varepsilon )</th>
<th>( C_{\varepsilon_1} )</th>
<th>( C_{\varepsilon_2} )</th>
<th>( C_{\varepsilon_3} )</th>
<th>( C_{\varepsilon_4} )</th>
<th>( k )</th>
<th>( E )</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.09</td>
<td>1.0</td>
<td>1.20</td>
<td>1.44</td>
<td>1.92</td>
<td>0 or 1.44*</td>
<td>-0.33</td>
<td>0.42</td>
<td>9.0</td>
</tr>
</tbody>
</table>

* \( C_{\varepsilon_3} = \begin{cases} \frac{1}{1.44} & \text{when } P_B > 0 \\ 0 & \text{when } P_B = 0 \end{cases} \)
4.3 NUMERICAL SOLUTION PROCEDURE

4.3.1 Approximation process

4.3.1.1 General equations

The general vectorial form of the conservation equation is convenient for the finite-volume discretisation:

\[
\frac{1}{\sqrt{g}} \frac{\partial}{\partial t} \left( \sqrt{g} \rho \phi \right) + \text{div}(\rho \mathbf{U}^r \phi - \Gamma_\phi \text{grad} \phi) = S_\phi
\]  
\hspace{1cm} (4-16)

where, \( \mathbf{U}_r = \mathbf{U} - \mathbf{U}_c \) is the relative velocity between the fluid \( \mathbf{U} \) and the local coordinate velocity \( \mathbf{U}_c \); \( \phi \) stands for any dependent variable; \( \Gamma_\phi \) and \( S_\phi \) are the associated "diffusion" and source coefficients respectively.

An arbitrary time-averaging volume \( V_p \) bounded by moving closed surfaces \( S_j \) is shown in Figure 4.1. The (4-16) equation can be written:

\[
\frac{d}{dt} \int_{V_p} \rho \phi dV + \sum_j \int_{S_j} (\rho \mathbf{U}^r \phi - \Gamma_\phi \text{grad} \phi) \cdot d\mathbf{S} = \int_{V} S_\phi dV
\]  
\hspace{1cm} (4-17)

\[T_1 \quad T_2 \quad T_3\]

where, \( T_1, T_2, \) and \( T_3 \) represent the terms in equation (4-17) respectively, they can be discretised as follows:

1) \( T_1 = \frac{d}{dt} \int_{V_p} \rho \phi dV \)
\[ T_1 = \frac{(\rho \phi V)_{t}^{n} - (\rho \phi V)_{t}^{o}}{\delta t} \quad (4-18) \]

where, the superscripts "o" and "n" refer to "old" and "new" time levels, respectively. \( t \) is the time increment.

2) \( T_2 = \sum \int_{s_j} (\rho \bar{U}_r \phi - \Gamma_\phi \text{grad} \phi) \cdot d\bar{S} \)

\[ T_2 = \sum \int_{s_j} (\rho \bar{U}_r \phi \cdot \bar{S}) - \sum \int \Gamma_\phi \text{grad} \phi \cdot \bar{S} = \sum C_j - \sum D_j \quad (4-19) \]

where, \( C_j \) and \( D_j \) represent the convection and diffusion terms respectively. The approximation of convection terms will be discussed in the next section. The diffusion terms:

\[ D_j = \Gamma_\phi \text{grad} \phi \cdot \bar{S} = \Gamma_\phi \left[ f^j(\phi_n - \phi_p) + \sum f^k_j \delta \phi^k_j \right] \quad (4-20) \]

where, \( f^j(\phi_n - \phi_p) \) - the gradient between \( p \) and \( n \), the neighbouring cell centred node

\( f_j \) - geometrical factors

\( \Gamma_{\phi,j} \) - the face diffusivity

\( \sum f^k_j \delta \phi^k_j \) - summation over all vertex pairs on face \( j \)

3) \( T_3 = \int_{V} S_{\phi} dV \)

\[ T_3 = S_1 - S_2 \phi_p \quad (4-21) \]
4.3.1.2 Convection discretisation

The convection terms in equation (4-19) are:

\[ C_j = \langle \rho \vec{U}_r \phi \cdot \vec{S} \rangle_j \]

The \( C_j \) will be rewritten as \( C_j = F_j \phi_j \) \hspace{1cm} (4-22)

where, \( F_j = \langle \rho \vec{U}_r \cdot \vec{S} \rangle_j \) is the mass flux normal to face \( j \), \( \phi_j \) is the average value at the face.

The manner in which the convective and diffusive fluxes are expressed in terms of nodal \( \phi \) values is one of the key determining factors on accuracy, boundedness and stability to the resulting properties for both steady state and transient flow calculation. At the case of high Reynolds number flow, the choice of convective flux approximation is particular important.

There are three main classes for convective flux approximation in widespread use: "Low-order" schemes, "Higher-order" schemes, and "Filtered" or "Blended" schemes.

STAR offers five schemes for user selection: Upwind Differencing Scheme (UD), which belongs to the "Low-order" schemes; Linear Upwind Differencing Scheme (LUD) and Central Differencing Scheme (CD), belonging to the "Higher-order" scheme; Also, the Self-filtered Central Differencing Scheme (SFCD) and Blended Differencing Scheme (BD), called "Filtered " and "Blended" scheme respectively.
For the mesh shown in Figure 4.2, these schemes provide the face value \( \phi_j \) as follows:

**UD**

\[
C_{j}^{UD} = F_j \begin{cases} 
\phi_p, & \text{if } F_j \geq 0 \\
\phi_n, & \text{if } F_j < 0 
\end{cases}
\] (4-23)

**LUD**

\[
C_{j}^{LUD} = F_j \begin{cases} 
\phi_p + (\phi_p - \phi_n) f_-, & \text{if } F_j \geq 0 \\
\phi_n + (\phi_n - \phi_{n+}) f_+, & \text{if } F_j < 0 
\end{cases}
\] (4-24)

where, \( f_- \) and \( f_+ \) are linear interpolation factors

**CD**

\[
C_{j}^{CD} = F_j \left[ f_+ \phi_p + (1 - F_+) \phi_n \right]
\] (4-25)

**SFCD**

\[
C_{j}^{SFCD} = \gamma_j C_{j}^{CD} + (1 - \gamma_j) C_{j}^{UD}
\] (4-26)

where, the \( \gamma_j \) are face-related weight factors \((0 \leq \gamma_j \leq 1)\), evaluated from the local gradients in such a way that the factors are unity except when extreme appear.

**BD**

\[
C_{j}^{BD} = \gamma C_{j}^{CD/LUD} + (1 - \gamma) C_{j}^{UD}
\] (4-27)

where, \( \gamma \) is a blending factor to blend the higher order differencing with first order upwind differencing (e.g., \( \gamma = 0 \) for pure upwind differencing, \( \gamma = 1 \) for no blending).

Among these schemes, the upwind differencing scheme produces discretised equation forms which are easy to solve, and produces solutions obeying the expected physical bounds, but sometimes give rise
to "numerical diffusion"; The linear upwind differencing scheme and central differencing scheme result in less numerical diffusion than the UD scheme, but may result in equations more difficult to solve, and have solutions exhibiting non-physical spatial oscillations, which lead to "numerical dispersion"; "Filtered" or "blended" schemes combine a high-order formulation with a strategy for suppressing the spatial oscillations. Local modification is almost invariably a form of reversion to a lower-order scheme either wholly or partially. Because these schemes must base their filtering/blending practice on the evolving solution, additional nonlinearities and coupling can adversely affect their performance in an overall computational fluid dynamics calculation.

As discussed in section 3.2.2.2, the upwind differencing scheme is chosen in the present work according to the correct physical bounds on p under all conditions and high intrinsic numerical stability. In order to diminish "numerical diffusion", a finer grid is generated for the present computational domain.

4.3.1.3 Final finite volume equation

Substituting the various approximated terms back into equation (4-17), then invoking the following discretised continuity equation:

$$\frac{(\rho V)^n - (\rho V)^o}{\delta t} + \sum F_j = 0$$

(4-28)

the final form of the discrete finite volume equation in its most compact form is obtained:
Am = effects of convection and/or diffusion

\[ B_p = (pV)^o / \delta t \]

\[ A_p = \sum_m A_m + S_2 + B_p \]

The choice of the convective differencing scheme can have a strong bearing on the reliability and speed of iterative methods, through the associated coefficient matrix Am. In particular, the high-order schemes can produce matrices which are less well-conditioned for solution than the lower-order scheme. Therefore, the Upwind Differencing Scheme chosen in the present work is the best scheme in this respect.

4.3.2 Solution algorithm

STAR employs a fully implicit scheme to solve the discretised equations to ensure that the pressure equation is satisfied simultaneously with those of momentum. Currently, STAR incorporates two different implicit algorithms: the SIMPLE algorithm of Patankar and Spalding (1972), an iterative scheme, and the PISO algorithm of Issa (1985), a split-operation scheme.

Both these algorithms employ forms of predictor-corrector strategy. An equation set for pressure, which is derived by combining the finite-volume momentum and mass conservation equations, aids the velocity field to exactly satisfy the discretised continuity equation.
4.3.2.1 Finite-volume equations

Finite-volume momentum equation

\[ A_p U^n_{i,p} = H(U^n_{i,m}) + B^n_p U^n_{i,p} + S_1 + D_p (P^n_{n^+} - P^n_{n^-}) \quad (4-30) \]

Where,

\[ H(U^n_{i,m}) = \sum_m A_m U^n_{i,m} \]

\[ (P^n_{n^+} - P^n_{n^-}) \quad \text{- pressure gradient} \quad \frac{\partial P}{\partial x_i} \]

\[ D_p \quad \text{- a geometrical coefficient} \]

Finite-volume continuity equation

\[ B^n_p - B^n_p + \sum_j (\rho^n U^n_j S_j) = 0 \quad (4-31) \]

where,

\[ U_j \quad \text{- velocity normal to the cell face} \]

\[ S_j \quad \text{- face area} \]

A cell-face momentum equation, which expresses \( U_j \) in terms of the nodal velocities, and neighbouring pressures (see Figure 4.3), is:

\[ \overline{A_p} U^n_j = \overline{H(U^n_{i,m})} + \overline{B^n_p U^n_{i,p}} + \overline{S_1} + \overline{D_p (P^n_p - P^n_{n^-})} \quad (4-32) \]

where, the overbars denote a form of averaging on the nodal momentum coefficients appearing under them.
A pressure equation is obtained by substituting (4-32) into the continuity equation (4-31):

$$A_p p_p^n = \sum_m A_m p_m^n + S_1$$  \hspace{1cm} (4-33)

The "source" term, $S_1$, is a function of the nodal velocity field $U_i^n$ and $U_i^o$ and other quantities. The superscript "o" and "n" refer to "old" and "new" time level, respectively. Thus, the pressure may be calculated from equation (4-33).

4.3.2.2 SIMPLE algorithm

SIMPLE is used solely for steady calculations, operating in an iterative mode, i.e. the time derivative terms are deleted from the finite-volume equations. The procedure starts with the solution of the momentum equations with a preliminary pressure field. Following the solution of the pressure correction, the newly calculated velocities are corrected in accordance with the field of pressure perturbations. Finally, the pressure is calculated from its own equation after solving the discretisations for other $\phi$'s if necessary. The procedure is repeated until a converged solution is obtained.

What follows are the order of the execution, which starts from initial values $\phi^o$ of the variable fields.

1. Guess the pressure field $p^o$ at the start of the calculation.
2. Solve the momentum equation (4-30) with $p^o$, to obtain the provisional nodal velocity field $U_{i}^{(1)}$, and the provisional face velocities $U_{j}^{(1)}$ via equation (4-32):

$$A_{p}U_{i,p}^{(1)} = H(U_{i,m}^{(1)}) + B_{p}^{o}U_{i,p}^{o} + S_{1} + D_{p}(P_{n}^{(o)} - P_{n}^{(o)})$$  \hspace{1cm} (4-34)

$$A_{p}U_{j}^{(1)} = H(U_{i,m}^{(1)}) + B_{p}^{o}U_{i,p}^{o} + S_{1} + D_{p}(P_{p}^{(o)} - P_{n}^{(o)})$$  \hspace{1cm} (4-35)

3. Solve the pressure equation (4-33), to get the new pressure field $p^{(1)}$:

$$A_{p}P_{p}^{(1)} = \sum_{m} A_{m}P_{m}^{(1)} + S_{1}$$  \hspace{1cm} (4-36)

where, $S_{1}$ is now a function of the known nodal velocities $U_{i}^{(1)}$ and $U_{i}^{(o)}$.

4. Calculate new velocity field:

The nodal velocity $U_{i,p}^{(2)}$ can obtain via the following equation

$$A_{p}U_{i,p}^{(2)} = H(U_{i,m}^{(1)}) + B_{p}^{o}U_{i,p}^{o} + S_{1} + D_{p}(P_{n}^{(1)} - P_{n}^{(1)})$$  \hspace{1cm} (4-37)

The face velocities $U_{j}^{(2)}$ can get via equation (4-32) by $U_{i}^{(2)}$ and $p^{(1)}$.

5. Solve the discretization equation for other variables if they influence the flow field.
6. Treat the correct pressure \( P^{(1)} \) as a new guessed pressure, return to step 2, and repeat the whole procedure until a converged solution is obtained. The velocity field then is a divergence free velocity field.

4.3.2.3 PISO algorithm

PISO is applicable to both transient and steady calculations, and can perform the steady calculation in either a time-marching or iterative mode.

Starting from initial values \( \phi^o \) of the variable fields, PISO advances through a time increment \( \delta t \) in the following sequence of steps.

1. Predictor stage

The provisional nodal velocity field \( U_i^{(1)} \) is obtained via equation (4-34) by the pressure field \( P^{(o)} \) at the start of the time step. Then, the provisional face velocities \( U_j^{(1)} \) are obtained via equation (4-35).

2. First stage

Using pressure equation, the pressure field \( P^{(1)} \) can be solved via equation (4-36), as mentioned above in the section on the SIMPLE algorithm. The velocity fields \( U_i^{(2)} \) and \( U_j^{(2)} \) can be obtained from equation (4-37). The resulting solution is an approximation to that of the original equations (4-28) and (4-29).
3. Additional corrector stages

Further correctors are performed as for the first one, using the generalised equations:

\[ A_p U_{i,p}^{(q+1)} = H(U_{i,m}^{(q)}) + B_p U_{i,p}^o + S_l + D_p (P_{n-}^{(q)} - P_{n-}^{(q)}) \]  
(4-38)

\[ A_p P_{p}^{(q)} = \sum_m A_m P_{m}^{(q)} + S_l \]  
(4-39)

where, \( q \) is the corrector level (\( q=1,2,\ldots \)).

The solution from the successive stages represents increasingly better approximations to the solution of the original equation, i.e. \( U^{(q+1)} \) and \( P^{(q+1)} \) tend towards \( U^o \) and \( P^o \) with increasing \( q \).

After completion of the required number of corrector steps, the solution produced is taken as the starting field for the next time step and the sequence is repeated. If the calculation of scalar fields such as the turbulence parameters and temperature is required, then they are performed in further steps executed after the final flow corrector.

In STAR-CD, the number of correctors executed is not predetermined. It is judged from an internal measure of splitting error to enhance the accuracy and reliability of the algorithm. For calculation with moving meshes, the new position is set to the predictor stage.

For steady flow calculation, one mode of PISO is to time-march from the initial field, setting \( \delta t \) to a "large" value to accelerate the approach
to the steady state and accepting that the intermediate solutions will not be time-accurate. The alternative approach of PISO is to dispense with the time derivative terms and regard the PISO sequence as an iteration loop, to be repeatedly executed until the solution is reached. This practice is equivalent to using a very large $\delta t$, therefore, in order to avoid numerical instability, under-relaxation is introduced (Computational Dynamics Ltd., 1991).

### 4.3.3 Algorithm selection

The predictor-corrector scheme is seldom stable without heavy under-relaxation and is often slow to converge. Both SIMPLE and the iterative mode of PISO require under-relaxation. Though SIMPLE is not as stable and efficient as PISO, there are additional controls available in SIMPLE to deal with the problems involving severe mesh distortion to get better effect.

The inlet port/cylinder assembly has very complex geometry, SIMPLE is chosen for steady state calculation in the present work. The PISO method is used for transient calculation.

With SIMPLE, the pressure under-relaxation factor is set to be 0.09, the relaxation factors of velocities and turbulence parameters are all set to be 0.5.
4.4 BOUNDARY CONDITIONS

For the computational domain of the flow through the inlet port and the cylinder using STAR-CD, the boundary conditions can be described as follows: inlet, outlet, prescribed pressure, and impermeable wall.

4.4.1 Treatment of boundaries

1) Inlet

All flow properties must be known and prescribed. These characterise the inlet state of the flow whose accurate prescription profoundly influenced the predictive realism of the computation.

The turbulent kinetic energy on the inlet boundaries is taken as:

\[
k_{\text{in}} = \frac{1}{2} (I \cdot U)^2 \tag{4-40}
\]

where, \( I \) is the local relative turbulence intensity, which is assumed to be about 5% of the mean velocity at the inlet port/cylinder plenum.

The turbulence dissipation rate \( \varepsilon \) is seldom measured and may be estimated via a length-scale assumption, and specification of \( k \):

\[
\varepsilon = C_{\mu}^{3/4} k^{3/2} / \ell \tag{4-41}
\]

The turbulence quantities are provided either the turbulent kinetic energy \( k \) and its dissipation rate \( \varepsilon \), or the turbulence intensity \( I \) and the
turbulence length scale $\ell$. For the flow in a duct or a pipe, the turbulence length scale $\ell$ is bounded by a characteristic dimension of the cross-section, and is usually at least an order of magnitude smaller than this. In the present work, a uniform velocity profile is applied at the inlet plane, specified according to the measured mass flow rate. $\ell$ and $I$ are provided for the turbulence quantities: $\ell$ is 0.1 of the diameter of the inlet plane of the plenum, and $I = 5\%$.

2) Outlet

The flow should strictly be everywhere directed outwards but these conditions are unknown. STAR's practice is to extrapolate from upstream to arrive at the boundary values. This can be expressed mathematically as:

$$\frac{\partial W}{\partial z} = 0 \quad (4-42)$$

If a tendency for inflow to occur at any location on an outlet boundary. STAR sets the normal component of the cell face velocity there to zero.

3) Prescribed pressure boundaries

The pressures at the boundary cell faces are assumed known, and the velocities at these faces are linked to the local pressure gradients by special momentum equations, whose coefficients are equated to those at the cell centre. These equations, together with the continuity constraint, effectively allow the magnitude and direction of the local flow (inwards or outwards) to be calculated.
If the flow is directed inward, the boundary values of scalar quantities are ascribed user-specified values; in the case of outflow, the zero gradient assumption used in the outlet boundary procedure is employed to extrapolate from the interior.

In order to minimise round-off errors in pressure gradient calculations, the relative pressure is specified over the boundary region. The other dependant variables such as temperature, turbulence parameters need to be specified for the calculation when the flow is inwards directed.

4) Wall

Figure 4.4 shows a typical computational cell adjacent to a wall. The wall functions are used to link the fluxes through the boundary face to those at the central node p. They are also used to modify the calculation of the "source" terms in the turbulence transport equations, all of these changes being necessary to reflect the steep, non-linear variations in the flow properties through the boundary layer.

The velocity profile:

\[ U^+ = \frac{U - U_w}{(\tau_w / \rho)^{1/2}} = f_u(Y^+) \quad (4-43) \]

where, \( U_w \) - wall velocity
\[ f_u(Y^+) = \begin{cases} 
Y^+, & Y^+ \leq Y_m^+ \\
\frac{1}{\kappa} \ln(EY^+), & Y^+ > Y_m^+ 
\end{cases} \quad (4-44) \]

\( Y^+ \) - the dimensionless normal distance from the wall

\[ Y^+ = \rho C_{\mu}^{1/4} k^{1/2} Y / \mu \quad (4-45) \]

For the wall function to be effective, the dimensionless normal distance \( Y^+ \) from the wall, which measures the node location at the outer edge of the boundary layer, must ensure that it satisfies the limits, \( 12 < Y^+ < 100 \), and \( Y_m^+ \) satisfies the equation:

\[ Y_m^+ - \frac{1}{\kappa} \ln(EY_m^+) = 0 \quad (4-46) \]

The values assigned to \( \kappa \) and \( E \) are given in Table 4.1

The momentum flux is the area-integrated wall shear stress \( \tau_w \), which is linked to the velocity distribution.

The thermal energy and chemical species wall fluxes \( F_{\phi,w} \) are accorded similar treatment to the above, using the assumption of a "universal" distribution of temperature or concentration \( \phi \).

\[ \phi^+ = \frac{\rho (\phi - \phi_w) (\tau_w / \rho)^{1/2}}{F_{\phi,w}} = f_\phi (U^+) \quad (4-47) \]

\[ f_\phi (U^+) = \sigma_{\phi,t} (U^+ + P) \quad (4-48) \]
\[ P = 9.24 \left\{ \left( \frac{\sigma_\phi}{\sigma_{\phi,t}} \right)^{3/4} - 1 \right\} \left[ 1 + 0.28 \exp\left( -\frac{0.007 \sigma_\phi}{\sigma_{\phi,t}} \right) \right] \]  \hspace{1cm} (4-49)

where, \( \phi_w \) - wall value

\( \sigma_\phi \) - molecular Prandtl/Schmidt number

\( \sigma_{\phi,t} \) - turbulent Prandtl/Schmidt number

In the present work, inlet, outlet and wall boundaries are specified for the steady state; the prescribed pressure and wall boundaries are specified for the transient case.

### 4.5 COMPLETION TESTS

In interactive calculations, the criterion for convergence is:

\[ \text{Max}(R^N_\phi) < \lambda \]  \hspace{1cm} (4-50)

where, \( \lambda \) - user defined value, it is \( 10^{-3} \), in the present work.

\( R_\phi \) - normalised residual sum

\( N \) - iteration number

\[ R^N_\phi = \frac{\sum |r^N_\phi|}{M_\phi} \]  \hspace{1cm} (4-51)

the summation is over all cells in the mesh.

\[ r^N_\phi = A_p \phi_p^N - \sum_m A_m \phi_m^N - S_i \]  \hspace{1cm} (5-52)

\( M_\phi \) is a normalising factor described in Appendix A.
In the case of transient flow, PISO is used. The \( \phi \) fields at each time step represent a close approximation to the finite-volume equations. The residuals are no longer appropriate to be considered. The global rates of change, \( C^N_\phi \), is an alternative form of monitoring information:

\[
C^N_\phi = \sum (|B^n_\phi \phi^n_p| - |B^n_\phi \phi^o_p|)
\]  

(4-53)

where,  \( N \) - number of time steps  
\( \sum \) - summation over all cells

The magnitudes of the \( C_\phi \) provide a global measure of the rates of change of mass, momentum, energy etc. within the calculation domain. Therefore, it can be used for a variety of purpose including a measure of the closeness of the solution to the steady state.

4.6 STAR-CD SYSTEM STRUCTURE AND CONCLUSION

4.6.1 STAR-CD system structure

The STAR-CD Code system composes the preprocessor and postprocessor code PROSTAR, and the main analysis code STAR. The analysis code contains the particular mathematical models for turbulent flow calculation, the related boundary condition options, and the numerical solution methods, which have been introduced in the above sections.

PROSTAR is an interactive command-driven code, combining pre and post processing within a single self-contained package. It is used for
defining geometry, boundary conditions, fluid properties, and analysis controls which uniquely determine the flow problem to be solved. It also contains plotting facilities to read the various data files produced by the analysis code and format the data through standard printouts and graphical techniques into comprehensible data sets, to aid users to work in a controlled and flexible environment.

STAR is the core analysis code, which generates thermofluids predictions corresponding to the input data specifications provided by PROSTAR. STAR incorporates a numerical finite-volume procedure for solving the governing partial-differential conservation equations of mass, momentum and energy.

When either PROSTAR or STAR are used, several files are created for operating the STAR-CD system. The STAR-CD system structure and its files usage are shown in Figure 4.5.

4.6.2 Restrictions

1) Turbulence modelling

All the turbulence models employed in the STAR-CD adopt the "law of the wall" to represent the flow, heat and mass transfer within boundary layers. STAR-CD does not allow the user to have access to the core of the program so as to modify any of the sub models.
2) **Mesh arrangement**

- **Aspect ratio**
  
  The aspect ratio is limited within 10.

- **Nonorthogonality**
  
  The internal angle between two edge lines of a cell must not be less than 45°.

- **Warpage**
  
  The angle between the surface normal to the two triangular surfaces making up the quadrilateral should not be less than 45° (see Figure 4.6).

### 4.6.3 Conclusion

Despite the restrictions, it can be said that STAR-CD is a powerful CFD tool for thermofluids analysis for use in a CAE environment. Especially, its highly flexible computational mesh system, which can be body-fitted non-orthogonal; unstructured, including local refinement capability; range of cell shapes, such as hexahedra, pentahedra, and tetrahedra; and rotating or distorting, facilitates. STAR's application to the complex geometries characteristic of industrial CFD problems, such as its use in the present complex flow in the inlet port/cylinder assembly is quite appropriate.
Figure 4.1 Typical cell with centred node p and neighbour cell with centred node n

Figure 4.2 Node labelling convention for flux discretisation
Figure 4.3 Arrangement of variables and notation for PISO implementation on Cartesian mesh

Figure 4.4 Near wall cell and notation
Figure 4.5 STAR-CD system structure and file usage
Figure 4.6 Mesh distortion
CHAPTER 5

STEADY FLOW IN
THE INLET PORT AND THE CYLINDER

5.1 INTRODUCTION

The objective of the present research is to develop an understanding of the behaviour of inlet port flows and in-cylinder flows in real engines, therefore, the computational domain representing the practical port and cylinder is generated for carrying out a detailed analysis of flows both in the steady and unsteady state. This chapter is concerned with the steady flow behaviour, while the unsteady flow will be discussed in the next chapter.

For steady flows, the surface static pressures predicted using the CFD code provide a useful indicator of flow separation within the port/cylinder assembly. These are presented and compared with experimental data. The velocity fields, measured by laser Doppler anemometry in a companion study at King's College London, using a steady flow test bench with identical port/cylinder geometry to that used here, but using a liquid as the working fluid for refractive index matching, compared favourably with the predicted data.

5.2 COMPUTATION OF STEADY FLOWS

5.2.1 Flow configuration

The flow configuration is illustrated in Figure 5.1. It consists of a cylinder and a curved inlet port, in which the axis of the valve
intersects the port axis. The curved port/valve assembly locates asymmetrically to the cylinder axis as shown in Figure 5.1. For such a practical curved port/cylinder assembly, it is a tedious task to generate a suitable computational grid. Table 5.1 summarizes the dimensions of the flow configuration. Steady flow simulations were carried out for five valve lifts: 3, 4, 6, 10 and 15 mm. The corresponding ratios of the valve lift to valve diameter are 0.07, 0.09, 0.14, 0.23 and 0.35.

Table 5.1 Port-valve-cylinder parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bore</td>
<td>93.65 mm</td>
</tr>
<tr>
<td>Valve head diameter</td>
<td>43.00 mm</td>
</tr>
<tr>
<td>Valve stem diameter</td>
<td>8.00 mm</td>
</tr>
<tr>
<td>Entrance radius</td>
<td>46.00 mm</td>
</tr>
<tr>
<td>Port entrance diameter</td>
<td>46.00 mm</td>
</tr>
<tr>
<td>Maximum valve lift</td>
<td>15.00 mm</td>
</tr>
<tr>
<td>Minimum valve lift</td>
<td>3.00 mm</td>
</tr>
<tr>
<td>Ambient pressure</td>
<td>9.92x10^4 Pa</td>
</tr>
<tr>
<td>Ambient temperature</td>
<td>20°C</td>
</tr>
</tbody>
</table>

Much effort was spent in perfecting the mesh model with a body fitted coordinate representation, especially in the regions of the port bend and the vicinity of the valve, see Figure 5.2. In the port bend, the valve stem intersects the port concave wall at an angle of 30°. Special attention was paid to create cells with the internal angle larger than 45° as far as possible. Otherwise, cell distortion would result in numerical diffusion, whereby the solution would be unstable or crash due to the acceleration of the numerical discretisation errors. In the vicinity of the valve, valve seat and cylinder head, the grid lines were carefully designed to be aligned with the streamlines. The aspect ratio of cells was controlled to be less than 10, such as the cells adjacent to the valve
seat, valve lip, and cylinder head, particularly, for the cases of lower valve lifts. In the cylinder, it was difficult to create the mesh configuration, because a pole boundary condition was not available in version 2.004 of STAR-CD. The computational grid structure at the centre was designed as shown in Figure 5.3 on account of the limitations of non-orthogonality and aspect ratio. Figure 5.4 shows the whole mesh model for the steady flow simulation.

5.2.2 Boundary condition specification

1. Inlet boundaries

As already mentioned in Chapter 2, prescribing inlet conditions properly, not only the velocity components but also the turbulence quantities, is very important to achieve predictive accuracy. Hence, on inlet boundaries, the distribution of all dependent variables, except the pressure, must be prescribed. In some cases, the inlet velocity profiles and turbulence energy $k$ are provided by experimental data. In the present case, where no experimental data were available, except the mass flow rate, the velocity component normal to the inlet plane was assumed to be uniform according to the measured mass flow rate. Table 5.2 shows the measured mass flow rates at different valve lifts.

<table>
<thead>
<tr>
<th>Valve lift (mm)</th>
<th>Mass flow rate (g/sec)</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>20.69</td>
</tr>
<tr>
<td>4</td>
<td>26.21</td>
</tr>
<tr>
<td>6</td>
<td>41.94</td>
</tr>
<tr>
<td>10</td>
<td>60.70</td>
</tr>
<tr>
<td>15</td>
<td>73.67</td>
</tr>
</tbody>
</table>

Table 5.2. Measured mass flow rates
The turbulent energy $k$ on the inlet boundaries was assigned via estimates of the local relative turbulence intensity $I$ which was assumed to be 5%.

$$k_{in} = (I U_{in})^2$$ (5.1)

The turbulence dissipation rate was derived from the relation of $k$, and $\ell$.

$$\varepsilon = C_m^{3/4} k^{3/2} / \ell$$ (5.2)

where, $\ell$ - the turbulence length scale. $\ell = 0.1$ D was assumed in the present case. D is the diameter of the entrance plane of the port.

2. Outlet boundaries

The outlet boundary conditions are unknown. The distributions of the variables on the outlet plane are estimated by extrapolation in the STAR code, assuming a zero gradient along the mesh lines intersecting with the outlet plane. At these boundaries, the flow should be directed outwards everywhere. Any inwardly directed normal velocity components are reset to zero. If inwards flow persists, it will result in failure of the calculation.

In order to avoid the influence of the recirculation regions beneath the valve on the flow at the outlet plane, the length of the cylinder was increased to be six times the cylinder bore.
3. Wall boundaries

All the velocity components and the turbulence energy are specified to be zero at the impermeable wall. The near-wall region is considered to be composed of a viscous sublayer where turbulent stresses are negligible and a fully turbulent layer where viscous transport can be ignored. Wall functions are used to link the fluxes through the boundary layer to those at the central node p. The location of node p is measured in terms of the dimensionless normal distance \( Y^+ \) from the wall. \( Y^+ = 12 \) represents the location where matching between the log-law profiles in the fully turbulent regime and the linear variation corresponding to the fully laminar sublayer occurs.

The node p must lie in the range of \( 12 < Y^+ < 100 \) to ensure the wall function to be effective.

The momentum flux is the area-integrated wall shear stress \( \tau_w \).

\[
\tau_w = \frac{\rho_p \kappa c \mu^{1/4} k^{1/2} U_p}{\ln (E Y_p c \mu^{1/4} K^{1/2} / \mu)}
\]  

(5.3)

where, \( U_p \) is the resultant velocity component parallel to the wall, given by a function of form, which combines functions (4-38) and (4-40).

\[
U_p^+ = \frac{U_p - U_w}{(t_w / r)} = f_a(Y_p^+) = \begin{cases} 
Y_p^+, & Y_p^+ \leq Y_m^+ \\
\frac{1}{k} \ln (E Y_p^+), & Y_p^+ > Y_m^+ 
\end{cases}
\]  

(5.4)

\[
Y_p^+ = \rho_p c \mu^{1/4} k^{1/2} Y_p / \mu
\]  

(5.5)
where, \( \kappa \) is the von Karman's constant. \( \kappa = 0.42 \).

\( E \) is an integration constant. For a smooth wall, \( E = 9.0 \).

In the \( k \) equation, the boundary treatment is completed by the modification of the source terms in the near-wall cells, according to the turbulence generation and the total turbulence dissipation.

The mean turbulence energy generation is expressed as:

\[
\overline{p}_k = \frac{1}{Y_n} \int_{Y_v}^{Y_p} \frac{\tau_w \partial U}{\rho \partial Y} dY = \frac{\tau_w}{\rho} \frac{(U_n - U_v)}{Y_n}
\]

(5.6)

where, the subscripts \( v \) and \( n \) denote values prevailing at the edge of the viscous layer and at the outer face of the control volume.

The total turbulence dissipation is evaluated by volume integration of the wall-function of \( \varepsilon \). The mean value of \( \varepsilon \) can be obtained as

\[
\bar{\varepsilon} = \frac{1}{Y_n} \int_0^{Y_p} \varepsilon dY = \frac{2vK_v}{Y_v Y_n} + \frac{c_\mu^{3/4} K^{3/2}}{\kappa Y_n} \ln\left(\frac{Y_p}{Y_v}\right)
\]

(5.7)

Finally, the treatment of the dissipation is quite different from that of the other variables. It is not employed at boundary cells. The nodal values of the dissipation rate \( \varepsilon \) are obtained directly from the algebraic wall-function relation:

\[
\varepsilon = \frac{c_\mu^{3/4} k_p^{3/2}}{\kappa Y}
\]

(5.8)
These nodal values then serve as boundary conditions for the solution of the finite-volume equation in the internal cells.

5.2.3 Flow field predictions

5.2.3.1 Flow structure

Predicted velocity vector plots are first shown in the y-z plane which includes the port axis, indicating how the flow pattern within the curved port and the cylinder develops. Figure 5.5 illustrates the flow pattern at the valve lift of 10 mm. To aid the interpretation of the complex flow structure, it is useful to identify different regions shown in Figure 5.5.

Port Flow

The flow profiles upstream of the port bend and the valve stem are quite similar for all valve lifts, indicating that the flow in region 1 is independent of the valve lift, because of the favourable local pressure gradient, hence the flow accelerates in region 1 without separation. After passing the port bend, the flow changes its structure with variation of valve lift.

Figures 5.6(a)-(d) show the predicted mean velocity vectors at valve lifts of 3, 4, 6 and 10 mm for regions 1 to 3. The velocity vectors in region 2 are directed towards the stem at higher valve lifts, indicating separation in the wake of the stem. The separation in region 2 caused by the adverse pressure gradient is seen clearly in Figures 5.7(a)-(d) which are the static pressure maps corresponding to the flow patterns
in Figures 5.6(a)-(d), and also in Figures 5.8(a)-(d) which show the static surface pressure maps for the valve lifts of 3, 4, 6 and 10 mm.

Along the concave port wall, the flow separates from the wall then reattaches about halfway between the top of the stem and the valve seat. The larger the valve lift, the stronger the separation. This can be seen in Figures 5.7(a)-(d), the area of the adverse pressure gradient in region 2 increases with increasing valve lift, and the gradient increases as well.

In region 3, between the valve stem and the convex port wall, the flow is towards the valve stem because of the separation from the convex wall. This separation becomes stronger with increasing valve lift due to a larger zone of adverse pressure gradient along the convex wall, shown in Figures 5.7(a)-(d).

When the flow passes through the passage between the valve head and the valve seat, the flow pattern varies with the valve lift due to substantial variations of the pressure distribution along the surfaces of both the valve head and valve seat. From Figure 5.9(a), we can observe that a small zone of adverse pressure gradient exists at the leading edge of the valve head when the valve lift is 3 mm (corresponding to the valve lift/diameter ratio, \( L_v / D_v \), 0.07). The flow maintains attachment to both sides of the passage and emerges as a jet with a max. velocity of 61 m/sec, see Figure 5.10(a). At valve lift of 4 mm (\( L_v / D_v = 0.09 \)), the adverse pressure gradient region increases at the inner edge of the valve head, the flow separates from the inner edge of the valve head (see Figures 5.9(b) and 5.10(b)). At
valve lift of 6 mm ($L_v / D_v = 0.14$), Figure 5.9(c), the adverse pressure gradient region moves along the valve head to the outer edge. The flow separates from the valve head, as shown in Figure 5.10(c). When the valve lift reaches 10 mm ($L_v / D_v = 0.23$), Figures 5.9(d) and 5.10(d), the adverse pressure gradient regions increase to the whole surface of the valve head and the valve seat. The flow separates from both sides of the passage to form a free jet. At even higher valve lift 15 mm ($L_v / D_v = 0.35$), Figures 5.9(e) and 5.10(e), there is no additional flow detachment. The valve head exerts less influence on the flow than at lower valve lifts.

The predicted pressure distribution in the passage between the valve head and valve seat indicates the separation of the flow at the valve head and valve seat, of which the effect on the discharge coefficient will be introduced later.

**In-cylinder flow**

The flow passes the valve passage as a conical annular jet into the cylinder. The interaction between jet flows and cylinder walls results in a complex vortical flow pattern which is strongly dependent on the port design, valve axis orientation and the valve lift.

Here, the flow structure is first shown in y-z plane which contains the valve axis. At low valve lift 3 mm (see Figure 5.11(a) showing regions 4-5), a strong jet emerges from the valve exit into the cylinder, interacts with the cylinder head and the cylinder wall in the negative y direction to create a strong vortex in the cylinder corner (the left of
125

region 4). A jet flow is established down the cylinder wall. In the positive y direction, the high velocity jet flow emerging into the cylinder attaches to the cylinder head surface due to the low pressure region adjacent to the cylinder head caused by entrainment into the incoming jet flow, which can be seen in Figure 5.11(a). When the jet reaches the cylinder wall, it is deflected downwards, forming a vortex with clockwise rotation in the right of region 5. The flow in the cylinder mainly moves from the right towards the left to interact with the left wall jet to create a vortex with counter clockwise rotation lower down in the cylinder. As valve lift increases, the jet enters the cylinder at a larger angle with the cylinder head surface because of the separation from the surfaces of valve seat and valve head, see Figures 5.11(b)-(d). When the valve lift is 10 mm, the jet angle is about 45° with the cylinder head. The larger the valve lift, the more freely the jet expands in the right side of the cylinder. At this larger angle of entry into the cylinder, the jet is deflected by the cylinder wall further down than in the low valve lift case, forming two counter-rotating vortices: one adjacent to the cylinder head, the other beneath the valve head centred near the valve axis. In the negative y direction, the vortex in the left of region 4 exists for the four valve lifts. The vortex beneath the valve (in the left of region 5), formed by the interaction of the wall jet and the valve head, is counter clockwise due to the interaction with the flow from the right (positive y direction) of the cylinder.

In the x-z plane, the flow is close to being axisymmetric because of only 4 mm offset of the valve centre to the cylinder centre along the x direction (see Figures 5.12(a)-(d)). Unlike the case for the y-z plane, the jet flow at low valve lift of 3 mm does not attach to the cylinder
head surface. Whatever the valve lift is, the jet interacts with the cylinder walls and cylinder head surfaces to create two pairs of counter-rotating vortices in both positive and negative x directions, one pair exists in the cylinder corner, the other is beneath the valve head. But, the position of their centres varies with valve lift. With valve lift increase, the vortices in cylinder corners move downwards, and their size is increased. On the contrary, the vortices beneath the valve head are upwards with reduced size.

In x-y planes, the characteristics of "swirling" flow can be seen. Figures 5.13(a)-(c) illustrate the flow patterns in the planes which are located -25, -45, and -93.65 mm from the cylinder head surface at valve lift of 3 mm. The jet flow at 61 m/sec velocity impinges on the left wall of the cylinder because of the off-centre valve location. The cylinder wall closest to the valve impedes the flow and forces it on either side of the plane, which passes through the valve and cylinder axes, to circulate around the cylinder in opposite directions.

At z= -25 mm, Figure 5.13(a), it is shown that two counter-rotating vortices are located at either side of the plane including the valve and cylinder axes. Along the cylinder length, the interaction of the jet flow in the positive y direction and the right side of the cylinder wall (i.e. the right of region 5) affects the flow towards the left, hence the vortex pair gradually moves towards the left (see the flow structure at z= -45 mm in Figure 5.13(b)). At z= -93.65 mm, the position of the plane from the cylinder head surface is equal to the cylinder bore, the flow circulates around the cylinder on either side of the plane including the valve and cylinder axes and turns leftwards in
the negative y direction due to the entrainment of the wall jet flow. These two vortices mentioned above combine to be a kidney-shaped vortex, as shown in Figure 5.13(c). The static pressure distribution in z= -25, -45, and -93.65 mm planes in respect to the flow patterns are shown in Figures 5.15(a)-(c).

With valve lift increase, the flow is more complex. Figures 5.14(a)-(c) illustrate the flow structures at valve lift of 6 mm. The jet flow enters the cylinder at a larger angle with the cylinder head surface, hence, the impact of the jet flow on the cylinder wall closest to the valve is not as strong as at lower valve lift, neither the wall jet flow.

At z= -25 mm, Figure 5.14(a), the interaction of the wall jet flow and the closest cylinder wall produces a flow towards the valve axis and circulating around the cylinder. At higher lift, the jet flow in the positive y direction expands more freely. The downwards flow creates the higher pressure region shown in Figure 5.16(a). There are two adverse pressure gradient regions on either side of the plane containing the valve and cylinder axes: one is near the valve axis, the other is near the cylinder axis, creating two pairs of counter-rotating vortex.

At z= -45 mm, Figure 5.14(b), the wall jet flow in the negative y direction becomes weak. The flow circulating around the cylinder is affected by the interaction of the jet flow in the positive y direction and the cylinder wall, creating a large region of higher pressure in the positive y direction (see Figure 5.16(b)). There is a small adverse pressure gradient region in the negative y direction, a small clockwise vortex is then created.
Further down the cylinder, at $z = -93.65$ mm, the effect mentioned above is stronger (Figures 5.14(c) and 5.16(c)). A clockwise vortex is formed due to an adverse pressure gradient region near the cylinder wall closest to the valve.

The flow patterns shown in Figures 5.13(a)-(c) and 5.14(a)-(c) show that the in-cylinder flow is strongly dependent on the location of the valve. Even if the distance from the cylinder head is equal to the cylinder bore, the flow at that plane still shows the effect of the off-centre location of the valve.

The numerical simulation also shows that the in-cylinder flow is strongly dependent on the jet speed and the jet angle with the cylinder head, which is established by the port shape and the valve lift.

The flow in the port and the cylinder is highly three-dimensional. The complexity of the turbulent flow is illustrated in Figure 5.17. The trajectories of four particles are drawn and they indicate the irregularity of the flow in the cylinder. The four particles are set to depart uniformly from the inlet plane of the port. The trajectories become different as soon as they cross the exit plane of the port and/or when they move downstream of the valve stem. After entering the cylinder at the negative $y$ side, the particle which is located in the negative $x$ direction and close to the port axis at the inlet, moves to the positive $y$ side of the cylinder, where it moves in a vortical fashion.
**Turbulence evolution**

Turbulence is generated during the intake process in the low Reynolds number layers where the flows are impinging and accelerating respectively, and in the shear layers of the jet flow.

Figures 5.18(a)-(c) show the development of turbulence kinetic energy inside the port and in the valve passage. Turbulent kinetic energy in the port is generated near the convex wall and the concave wall, and the area upstream of the valve seat where the flow is accelerated. In the core of the port flow and adjacent the valve stem, turbulent kinetic energy is relatively lower. At the lower valve lift, 3 mm, there is no separation in the port; turbulence is generated mainly in the shear layers of the jet (see Figure 5.18(a)). With valve lift increase, separation occurs from both the convex and concave walls, the flow impinges on the valve stem from the top to the half height of the stem, as shown in Figures 5.6(a)-(d). Hence, the turbulent kinetic energy increases in these areas (Figures 5.18(b) and (c)).

Figures 5.19(a)-(c) illustrate the turbulent kinetic energy distribution in the cylinder. Higher turbulent kinetic energy is created in the shear layers of the jet flow. In early induction (at valve lift of 3 to 6 mm, Figures 5.19(a) and (b)), the turbulent kinetic energy increases with the valve lift increase due to separation of the jet from the valve head and the valve seat. Further increasing the valve lift results in the turbulent kinetic energy decaying (see Figure 5.19(c), at valve lift 10 mm).
There is little turbulence transported from the jet flow to the rest of the cylinder where the turbulent kinetic energy is very low. Both in the port and in the cylinder, the turbulence is strongly non-homogeneous.

### 5.2.3.2 Discharge coefficient

The discharge coefficient $C_d$ is a parameter that describes the effectiveness of the intake process in the engine. It is defined to determine the variation between ideal and actual mass flow rate through the inlet valve.

$$C_d = \frac{\dot{m}_a}{\dot{m}_i} \quad (5.9)$$

where, $\dot{m}_a$ is the actual mass flow rate, determined by measurement. $\dot{m}_i$ is the ideal mass flow rate, related to the upstream stagnation pressure $p_o$ and density $\rho_o$, static pressure just downstream of the valve $p$, and a reference area $A_1$.

$$\dot{m}_i = A_1 \left\{ 2p_o\rho_o \frac{\gamma}{\gamma-1} \left( \frac{p}{p_o} \right)^{\frac{\gamma}{\gamma-1}} \left[ 1 - \left( \frac{p}{p_o} \right)^{\frac{\gamma-1}{\gamma}} \right] \right\}^{\frac{1}{2}} \quad (5.10)$$

The reference area chosen here is the so-called curtain area, which is defined as:

$$A_1 = \pi D_v L_v \quad (5.11)$$

Figure 5.20 shows the variation of the discharge coefficient with the valve lift/diameter ratio both by experiment and prediction. Both
measured and predicted discharge coefficient results show three segments, which correspond to the different flow patterns through the valve passage shown in Figures 5.10(a)-(e). When $L_v/D_v$ is less than 0.09, the discharge coefficient $C_d$ increases with the valve lift. At intermediate lift, $L_v/D_v = 0.09$, $C_d$ decreases abruptly because of separation from the inner edge of the valve head. $C_d$ then increases until $L_v/D_v$ reaches 0.14. During this period, the region of the adverse pressure gradient moves to the outer edge of the valve head, but its size remains approximately constant. In the high lift range, $L_v/D_v > 0.14$, separation entails large energy loss, and results in a decrease of the discharge coefficient.

Although there are discrepancies between the measured and predicted discharge coefficient at the higher and lower valve lifts, the computational results interpret the separations of the flow in the port bend and at the valve seat and valve head, and the effects on the discharge coefficient. Separation that occurs at higher lifts caused by adverse pressure gradients appear to be correctly predicted within the limitations of the k-ε model. This clarifies our understanding of the variation of discharge coefficient with the valve lift - diameter ratio explained by Tanaka (1929), Annand and Roe (1974). The discrepancies are caused by the inability of the k-ε model to predict flow separation at higher lift, and strong flow acceleration at lower lift. At $L_v/D_v = 0.23$, the predicted discharge coefficient is underpredicted by 24%, due to underpredicted pressure. At $L_v/D_v = 0.35$, there is no further more separation than before, the static pressure downstream the valve is less underpredicted. Hence the discrepancy between the measured and predicted $C_d$ reduces to 9%. At
L_v/D_v = 0.07, C_d is overpredicted by 12% because of the overestimated pressure. The discrepancies between the measurement and the prediction will be further examined later.

5.3 COMPARISONS WITH EXPERIMENTS

5.3.1 Wall pressure measurement

5.3.1.1 Introduction

Wall pressure measurement is a conventional means of investigation of fluid flow behaviour. Harderberg and Daudel (1975) proved that wall pressure measurement is a useful and simple method to study a helical inlet port characteristics with a steady flow test bench.

As mentioned in the last section, the pressure gradient is directly related to the flow pattern in the inlet port/cylinder assembly. Therefore, a twice full size model of the port/valve cylinder assembly was built for the steady air flow tests for fixed valve lifts to provide information on surface pressure distribution. Air was induced through the model with a fan. The mass flow rate was measured by an orifice metre (see Appendix B). A honeycomb section was placed near the exit of the cylinder to ensure that the flow was uniform across the cylinder before entry to the measuring section. Figure 5.21(a) shows the measurement equipment.

The port and valve surfaces were suitably drilled to accept 1.5 mm capillary tubing flush with the surfaces to act as static pressure
tappings. There were 16 along the port circumference, and 15 along the port axial direction, connected to an inclined manometer system with kerosine as the working fluid to provide an expanded measuring scale. A photograph of the test assembly is shown in Figure 5.21(b). The static pressure was measured individually on each pressure line. During measurement, the other pressure tappings were made inoperative through use of plugs.

Isobaric diagrams were obtained for analysis with flows at valve lift 10 mm and 6 mm. Static pressures on a modified port wall surface were also measured at valve lift of 10 mm. Table 5.3 shows the conditions of the steady flow tests.

<table>
<thead>
<tr>
<th>Type of port</th>
<th>Cylinder bore (mm)</th>
<th>Valve diameter (mm)</th>
<th>Valve lift (mm)</th>
<th>Mass flow rate (g/sec)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Original port</td>
<td>93.65</td>
<td>43</td>
<td>10</td>
<td>60.70</td>
</tr>
<tr>
<td>Modified port</td>
<td>93.65</td>
<td>39</td>
<td>10</td>
<td>60.70</td>
</tr>
</tbody>
</table>

5.3.1.2 Comparisons between measured and predicted static pressures

Figure 5.22 shows the plotted wall pressure isobaric diagram. The measured pressure contours agree reasonably well with the predicted wall pressure, which is shown in Figure 5.23, except for the separation in regions 2 to 3, and near the valve seat.
Figure 5.24 illustrates the comparisons of the pressure drop in various cross-sections. Pressure drops in planes 1 to 4 are fairly well predicted. Plane 4 is a cross-section passing the top of the valve stem, at an angle of 45° to the surface of cylinder head.

Between plane 4 and 5, there is separation in the wake of the valve stem, the pressure is overpredicted by about 10%. Between plane 5 and 6, both favourable pressure gradient regions and adverse pressure gradient regions exist. The former exists as acceleration of the flow occurs, where the pressure drop is underpredicted. The latter causes flow separation. The combined effects of these two pressure gradients result in the pressure drop being overpredicted by 30%. Being upstream of the valve seat, the flow is affected by the flow separation from the valve seat and from the left port wall, the pressure drop is also overpredicted by 30%.

The discrepancies between the predicted and experimental results for surface pressures are caused for the following reasons: Firstly, the k-ε turbulence model does not respond properly to adverse pressure gradients. The k-ε model underestimates the turbulence dissipation rate near the wall, overestimating values of the kinetic energy k and shear stress $-u'v'$ which result in larger surface friction prediction in the decelerating boundary layer which causes excess momentum to diffuse into the wall boundary layer from the mainstream. Secondly, due to the production method used on the twice full size port, the cross-section area between planes 2 and 4 is approximately 8% smaller than for the computational study. Hence the section between plane 2 and 4 is narrower than the section downstream of it. The slightly
diverging section downstream of plane 4 will produce an adverse pressure gradient that would promote separation earlier than in the prediction and higher static pressure.

The predicted static surface pressure maps show the main feature of the flow through the port and the cylinder, and indicate all the separation zones caused by the adverse pressure gradient, it also shows some small recirculation zones which are not easily discernible from vector plots. At valve lift 10 mm, there is a small recirculation zone adjacent to the left port wall, which is not discernible from the vector plot (Figure 5.6(d)), but an adverse pressure gradient region is clearly seen in Figure 5.7(d).

5.3.2 Flow visualization and LDA measurements

The experimental investigation of the inlet port flow has been carried out in a companion study at King's College London under the supervision of Dr. Yianneskis. The port is made from transparent acrylic plastic. A mixture of liquids with a refractive index identical to that of the acrylic material is used to facilitate optical access to the flow field.

5.3.2.1 Flow visualization

Photographs of the visualization indicate the flow structure in the inlet port/cylinder assembly. Figures 5.25 (a) and (b) show the flow pattern in the port. Between the valve stem and the concave wall (i.e. region 2), the flow is towards the valve stem as shown in the predicted port
flow result, indicating the recirculation in the wake of the valve stem. Some bubbles exist in the region adjacent to the convex wall of the port bend (i.e. region 3), and also near the crown. These small recirculations were not reproduced in the numerical simulations because of the deficiency of the k-ε turbulence model. Upstream of the valve seat, between the left wall and the stem, the flow is strongly accelerated and towards the axis of the valve. In the cylinder, both on the y-z planes (Figures 5.25 (c) and (d)), and x-z planes (Figures 5.26 (a) to (d)), two pairs of counter-rotating vortices are shown similar to those in the predicted results. One is in the corner of the cylinder wall and the cylinder head, the other is beneath the valve head.

The visualisation video shows the same effect of the valve lift on the port flow and the in-cylinder flow as the present predictions shown in the last section. The flow upstream of the port bend is independent of the valve lift. The flow downstream of the bend is affected by the variation of the valve lift.

With valve lift increase, the recirculation between the valve stem and the concave wall of the port is stronger. But, the flow re-attaches to the concave wall earlier than at lower lift because of the effect of the stronger separation from the convex wall of the port.

At higher valve lift, the jet flow emerges from the valve exit into the cylinder at a larger angle with the cylinder head surfaces. The jet in positive y direction is more freely expanding, forming two larger vortices on each side of the jet due to the interaction of the jet flow and the right side of the cylinder wall. In the negative y direction, the
vortex in the cylinder head corner is stronger and larger. But, the vortex beneath the valve is smaller because the wall jet flow is not as strong as that at lower valve lift. In the x-z plane, the pair of the counter-rotating vortex in the cylinder head corners becomes larger with lower centre position, and the other pair beneath the valve moves upwards with smaller size (see Figures 5.25 (c-d) and 5.26).

5.3.2.2 LDA measurements

The LDA measurements were made at a series of fixed valve lifts. The liquid flow rate was adjusted to produce the same Reynolds number, 25760, in the port. Three mean velocity components and corresponding turbulence levels (r.m.s velocities) were measured in various horizontal planes to where axial distances were measured from the exit plane of the inlet port.

The working fluid used was a mixture of oil of turpentine and tetraline in the proportion 69.2 : 30.8 by volume. The refractive index of the mixture (1.489 at a temperature 21.72 °C) is identical to that of the acrylic material used for the test section. The density and kinetic viscosity of the mixture were 891 kg/m³ and 1.71x10⁻⁶ m²/s respectively.

Figure 5.27 shows the comparisons of the predicted and measured axial velocity components at ten cross-sections. Figure 5.27 (a) presents the axial velocity profiles in planes z = 45, 25, 5, -10 and -30 mm. Figure 5.27 (b) is for planes z = 35, 15, -5, -20, and -40 mm. The predicted axial mean velocity components are in good agreement with the
measurement results except the separation area which is adjacent to the left wall just ahead of the valve seat (at \( z = 5 \) and 15 mm). The present prediction did not reproduce the circulation bubbles in this area which was shown in the visualisation video. The axial velocity is overpredicted because of the inability of the \( k-\varepsilon \) model to reproduce the separation. In the region between the valve stem and the concave wall at the port bend, the predicted axial velocity is distorted with the velocity maximum being shifted to the concave wall. This phenomena has been also observed both in the experimental study of Rowe (1966) and the computational study of Patankar et al (1975) for a 180° pipe bend. Patankar et al. used the \( k-\varepsilon \) model to simulate the flow in a helically coiled pipe, at a section far enough from the entry for the pattern not to change from one section to the next. They compared their computational results with experimental data of Mori and Nakayama (1967) for Reynolds number \( 2.5 \times 10^4 \). In Patankar's results, the magnitude of the friction factor is underpredicted by 8\% (corresponding to a 8\% overestimated pressure drop). Their axial velocity component is overpredicted about 10\%. The values of the differences between prediction and measurement for pressure and velocity are the same as for the present results at the port bend which is 10\%.

Figures 5.28 (a) and (b) are the comparisons of the radial velocity components in the respective planes. Because the separation in the left port wall upstream of the valve seat is not predicted, the flow is not towards the positive \( y \) direction as strong as the flow in the experiment. Hence, the predicted radial velocity component in the port is underpredicted by 20\%, leading to a larger jet angle with the
cylinder head in positive direction, and underpredicted radial velocity component of the in-cylinder flow.

Besides the defect of the k-ε model, another reason causing the discrepancy is measurement uncertainties that cause experimental errors, for the following reasons: positions of the control volume, seeding, frequency shifting, bias effects, statistical error, Doppler broadening and count ambiguity. Suen (1992) and Cheung (1989) checked the accuracy of the LDA measurement. They estimated that the overall errors in the mean and r.m.s. velocity measurements were approximately 5% and 10% respectively.

In addition, the test section is not precisely the same size as the computational model due to the production tolerances in manufacture that are 4% smaller for the ratio of the cylinder bore to the valve diameter and 6% larger for the valve diameter, and then show that they are likely to produce higher velocities than expected.

5.4 EFFECT OF THE INLET PORT SHAPE

The overall impression conveyed by the above results is that the characteristics of the flow in the inlet port/cylinder assembly strongly depend on the port design, valve stem orientation, the valve lift and the cylinder geometry.

Earlier investigators improved the performance of port designs by trial and error processes. It was found that acceleration of the flow just ahead of the valve seat by a short convergent section in the port
followed by a divergent section, produced favourable mean flow characteristics.

Dennison et al (1931) showed an improvement of the intake flow by designing the flow passage after the valve seat as a diffuser in their tests of the valve. They proved that it was effective to use a diffusing flow passage either at the valve head and its seat or above the valve. Wood et al (1965-1966) found that an appreciable pressure recovery occurred in the divergent section of the passage between the valve and its seat. Kastner et al (1963-1964) explored the optimisation of port configuration by tapering the port to form a throat just upstream of the valve seat. They found that a diverging flow passage following the throat could improve the flow pattern through the valve, and increase the mass flow. But, the diameter of the throat should not be smaller than the valve diameter by up to 13%. The effect of the port shape on the discharge coefficient $C_o$ for different pressure drops has been shown in Figure 2.8. The original port designated as type III in the Figure, had a throat diameter equal to the valve diameter. Modified ports IIIa and IIIb also had a throat diameter equal to that of the valve. Port IIIc and IIId had throat diameters smaller than the valve diameter by 13% and 30% respectively.

Using the method of wall pressure measurement, Hardenberg and Daudel (1975) studied a helical port with progressively reduced valve diameter. They found that a reduced valve diameter to 89% of the original produced a flow with little separation, generating the lowest pressure and the highest velocity just ahead of the valve seat. Higher volumetric efficiency was achieved on engine tests.
The present study used both predicted pressure plotting and measured static wall pressure mapping to explore the optimum valve size of the present curved port. In accordance with the references above, the valve diameter in this study was reduced from 43 mm to 39 mm. With the same mass flow rate of 60.7 g/sec, the modified port flow was simulated and measured at valve lift 10 mm.

5.4.1 Prediction

Figures 5.29 and 5.30 show the pressure distributions of the modified port at valve lift 10 mm on the surface and on the y-z plane respectively. Compared to the original design (see Figures 5.7(d) and 5.8(d)), the adverse pressure gradient regions both on the concave and the convex walls are smaller. The pressure drop between the entrance plane and the exit of the port increases by about 25%, indicating the flow with higher velocity passing the valve and a better pressure recovery downstream of the valve exit. The flow in the port is rearranged, as shown in Figure 5.31. At the top of the valve stem, the velocity maximum shifts to the concave wall, then the flow moves smoothly along the passage between the stem and the port wall. On entering the cylinder, the pressure recovery is 18% higher than obtained in the original calculation. The interactions of the stronger jet flow with the cylinder head and the cylinder wall generates a larger vortex pair than in the original calculation, (see Figure 5.5).

Figures 5.32 and 5.33 illustrate the distribution of the turbulent kinetic energy. In the modified port, the region with high value of the turbulent kinetic energy moves towards the concave wall,
corresponding to the flow structure near the top of the valve stem. In
the cylinder, turbulence is transported more from the jet to the rest of
the cylinder, than in the original case. It is clearly seen the
corresponding turbulence distribution and magnitude to the trends of
the mean velocity gradients.

The discharge coefficient, which was calculated using the pressure
drop across the valve, was 16.5% higher than for the original case.
Further reduction of the valve diameter to 35 mm, resulted in the
predicted discharge coefficient being reduced by 4%.

5.4.2 Measurement

Compared with the predicted wall pressures, the measured pressures
upstream of the port bend (see Figure 5.34 and Figure 5.35) are in
good agreement with the predicted pressures. Downstream of the port
bend, the measured pressures are higher than the predicted pressures,
especially, in the area of reduced diameter part of the port. In the
cylinder, the flow in the modified port/cylinder assembly has stronger
recirculations. The measured pressure drops are larger than the
predicted pressure drops.

Comparisons of the results between the original port and the modified
port show that the errors of the wall pressure between the predicted
and the measured are of the same order (20% - 30%). The difference
is that the predicted pressure in the cylinder for the original
port/cylinder assembly is underpredicted; the predicted pressure in the
cylinder for the modified port is overpredicted. The reason causing
pressure overprediction is that smaller regions of adverse pressure gradient in the valve passage are predicted, resulting in the jet flow having a much higher velocity, consequently, generating much stronger recirculations in the cylinder.

The present computational and experimental results show that an optimum valve diameter exists in an existing port to produce a favourable flow passage which is convergent/divergent in form, accelerating the flow up to the throat of the passage, followed by pressure recovery in the divergent section. Favourable mean flow characteristics are produced, resulting in a higher discharge coefficient.

5.4.3 Closure

The flow characteristics of a curved inlet port were studied by numerical prediction using CFD and confirmed experimentally using static pressure measurement at the port/valve surfaces. The predictions show the main features of the flow through the curved port and within the cylinder. The flow is highly three-dimensional. It is strongly dependent on the valve lift and the port geometry except upstream of the port bend and the valve stem.

The static pressure measurement method provides a simple and inexpensive tool for understanding port flow behaviour and as a design aid.
The predicted static pressure plots provide detailed information of the mean flow in the port/valve cylinder, including separation that is not always apparent from vector plots.

CFD contributes to our understanding of the port/in-cylinder three-dimensional flow. It clarifies earlier experimental work, and also provides detailed information, where the LDA measurement technique may be restricted.
Figure 5.1 Layout of the curved port/valve assembly
Figure 5.2 Mesh model in the regions of the port bend and the vicinity of the valve

Figure 5.3 Grid structure at the cylinder centre
Figure 5.4 Mesh model for the steady flow
Figure 5.5 Flow pattern at valve lift of 10 mm
Figure 5.6(a) Flow in 1-3 regions at valve lift 3 mm

Figure 5.6(b) Flow in 1-3 regions at valve lift 4 mm
Figure 5.6(c) Flow in 1-3 regions at valve lift 6 mm

Figure 5.6(d) Flow in 1-3 regions at valve lift 10 mm
Figure 5.7(a) Static pressure in 1-3 regions at valve lift 3 mm

Figure 5.7(b) Static pressure in 1-3 regions at valve lift 4 mm
Figure 5.7(a) Static pressure in 1-3 regions at valve lift 3 mm

Figure 5.7(b) Static pressure in 1-3 regions at valve lift 4 mm
PROSTAR 2.0
PRESSURE
N/M^2
LOCAL MX=-47.89
LOCAL MY=-3563.

-47.89
-299.0
-90.1
-30.2
-10.3
-1.56
-1.08
-0.25
-0.30
-0.59
-0.98
-1.37
-1.77
-2.17
-2.57
-2.96
-3.35
-3.74
-4.13
-4.53

PROSTAR 2.0
PRESSURE
N/M^2
LOCAL MX=114.8
LOCAL MY=-3664.

-114.8
-98.8
-82.9
-67.0
-51.1
-35.2
-19.3
-4.4
-0.1
-1.5
-2.9
-4.3
-5.7
-7.2
-8.6
-10.1
-11.5
-13.0
-14.4
-15.8
-17.2
-18.7
-20.1
-21.6
-23.0
-24.5
-26.0
-27.5
-29.0
-30.4
-31.9
-33.4
-34.9
-36.4
Figure 5.7(c) Static pressure in 1-3 regions at valve lift 6 mm

Figure 5.7(d) Static pressure in 1-3 regions at valve lift 10 mm
Figure 5.7(c) Static pressure in 1-3 regions at valve lift 6 mm

Figure 5.7(d) Static pressure in 1-3 regions at valve lift 10 mm
Figure 5.8(a) Surface pressure in 1-3 regions at valve lift 3 mm

Figure 5.8(b) Surface pressure in 1-3 regions at valve lift 4 mm
Figure 5.8(a) Surface pressure in 1-3 regions at valve lift 3 mm

Figure 5.8(b) Surface pressure in 1-3 regions at valve lift 4 mm
Figure 5.8(c) Surface pressure in 1-3 regions at valve lift 6 mm

Figure 5.8(d) Surface pressure in 1-3 regions at valve lift 10 mm
Figure 5.8(c) Surface pressure in 1-3 regions at valve lift 6 mm

Figure 5.8(d) Surface pressure in 1-3 regions at valve lift 10 mm
Figure 5.9(a) Pressure distribution in valve passage at valve lift 3 mm

Figure 5.9(b) Pressure distribution in valve passage at valve lift 4 mm
Figure 5.9(a) Pressure distribution in valve passage at valve lift 3 mm

Figure 5.9(b) Pressure distribution in valve passage at valve lift 4 mm
Figure 5.9(c) Pressure distribution in valve passage at valve lift 6 mm

Figure 5.9(d) Pressure distribution in valve passage at valve lift 10 mm
Figure 5.9(c) Pressure distribution in valve passage at valve lift 6 mm

Figure 5.9(d) Pressure distribution in valve passage at valve lift 10 mm
PROSTAR 2.0

PRESSURE

LOCAL MX=931.4
LOCAL MN=3779.

PROSTAR 2.0

PRESSURE

LOCAL MX=2111
LOCAL MN=5029.
Figure 5.9(e) Pressure distribution in valve passage at valve lift 15 mm
Figure 5.9(e) Pressure distribution in valve passage at valve lift 15 mm
PROSTAR 2.0
PRESSURE
N/M^2
LOCAL MX=2448
LOCAL MN=3787.

-2448
-2540
-2653
-2770
-2895
-2919
-3013
-3107
-3201
-3296
-3380
-3464
-3578
-3673
-3767
Figure 5.10(a) Flow pattern in valve passage at valve lift 3 mm

Figure 5.10(b) Flow pattern in valve passage at valve lift 4 mm
Figure 5.10(c) Flow pattern in valve passage at valve lift 6 mm

Figure 5.10(d) Flow pattern in valve passage at valve lift 10 mm
Figure 5.10(e) Flow pattern in valve passage at valve lift 15mm
Figure 5.11(a) Flow pattern in the cylinder at valve lift 3 mm

Figure 5.11(b) Flow pattern in the cylinder at valve lift 4 mm
Figure 5.11(c) Flow pattern in the cylinder at valve lift 6 mm

Figure 5.11(d) Flow pattern in the cylinder at valve lift 10 mm
Figure 5.12(a) Flow pattern in the cylinder at valve lift 3 mm

Figure 5.12(b) Flow pattern in the cylinder at valve lift 4 mm
Figure 5.12(c) Flow pattern in the cylinder at valve lift 6 mm

Figure 5.12(d) Flow pattern in the cylinder at valve lift 10 mm
Figure 5.13(a) Flow in z=-25mm at valve lift 3 mm

Figure 5.13(b) Flow in z=-45mm at valve lift 3 mm
Figure 5.13(c) Flow in $z=-93.5\text{mm}$ at valve lift 3 mm

Figure 5.14(a) Flow in $z=-25\text{mm}$ at valve lift 6 mm
Figure 5.14(b) Flow in z=-45 mm at valve lift 6 mm

Figure 5.14(c) Flow in z=-93.5 mm at valve lift 6 mm
Figure 5.15(a) Pressure distribution in $z=-25$ mm at valve lift 3 mm

Figure 5.15(b) Pressure distribution in $z=-45$ mm at valve lift 3 mm
Figure 5.15(a) Pressure distribution in z=-25 mm at valve lift 3 mm

Figure 5.15(b) Pressure distribution in z=-45 mm at valve lift 3 mm
Figure 5.15(c) Pressure distribution in $z= -93.5$ mm at valve lift 3 mm

Figure 5.16(a) Pressure distribution in $z= -25$ mm at valve lift 6 mm
Figure 5.15(c) Pressure distribution in $z=-93.5$ mm at valve lift 3 mm

Figure 5.16(a) Pressure distribution in $z=-25$ mm at valve lift 6 mm
Figure 5.16(b) Pressure distribution in $z=-45$ mm at valve lift 6 mm

Figure 5.16(c) Pressure distribution in $z=-93.5$ mm at valve lift 6 mm
Figure 5.16(b) Pressure distribution in $z=-45$ mm at valve lift 6 mm

Figure 5.16(c) Pressure distribution in $z=-93.5$ mm at valve lift 6 mm
Figure 5.18(a) Turbulent kinetic energy in 1-3 regions at valve lift 3 mm

Figure 5.18(b) Turbulent kinetic energy in 1-3 regions at valve lift 6 mm
Figure 5.18(a) Turbulent kinetic energy in 1-3 regions at valve lift 3 mm

Figure 5.18(b) Turbulent kinetic energy in 1-3 regions at valve lift 6 mm
Figure 5.18(c) Turbulent kinetic energy in 1-3 regions at valve lift 10 mm

Figure 5.19(a) Turbulent kinetic energy in the cylinder at valve lift 3 mm
Figure 5.18(c) Turbulent kinetic energy in 1-3 regions
at valve lift 10 mm

Figure 5.19(a) Turbulent kinetic energy in the cylinder
at valve lift 3 mm
Figure 5.19(b) Turbulent kinetic energy in the cylinder at valve lift 6 mm

Figure 5.19(c) Turbulent kinetic energy in the cylinder at valve lift 10 mm
Figure 5.19(b) Turbulent kinetic energy in the cylinder at valve lift 6 mm

Figure 5.19(c) Turbulent kinetic energy in the cylinder at valve lift 10 mm
Figure 5.20 Variation of the discharge coefficient with the ratio of valve lift to valve diameter
Figure 5.21(b) The test assembly
Figure 5.23 Predicted wall pressure
Figure 5.22 Measured wall pressure

Figure 5.23 Predicted wall pressure
Figure 5.25 Visualization of flow pattern in the y-z plane
Figure 5.26 Visualization of flow pattern in the x-z plane
Figure 5.27(a) Comparison of the axial velocity component in planes $z = 45, 25, 5, -10$ and $-30$ mm
Figure 5.27(b) Comparison of the axial velocity component
in planes $z = 35, 15, -5, -20, \text{ and } -40 \text{ mm}$
Figure 5.28(a) Comparison of the radial velocity component in planes $z = 45, 25, 5, -10$ and $-30$ mm
Figure 5.28(b) Comparison of the radial velocity component in planes z = 35, 15, -5, -20, -40 mm
Figure 5.29 Surface pressure map of the modified port/valve assembly

Figure 5.30 Static pressure distribution in the modified port/valve assembly
Figure 5.29 Surface pressure map of the modified port/valve assembly

Figure 5.30 Static pressure distribution in the modified port/valve assembly
Figure 5.31 Flow structure in the modified port and the cylinder
Figure 5.32 Distribution of the turbulent kinetic energy in the modified port

Figure 5.33 Distribution of the turbulent kinetic energy in the cylinder
Figure 5.32 Distribution of the turbulent kinetic energy in the modified port

Figure 5.33 Distribution of the turbulent kinetic energy in the cylinder
Figure 5.34 Measured wall pressure for the modified port

Figure 5.35 Predicted wall pressure for the modified port
Figure 5.35 Predicted wall pressure for the modified port
CHAPTER 6
COMPUTATION OF UNSTEADY FLOW

Predictions of transient flow for the complete induction and compression strokes were performed. The mesh coordinates change by a subroutine NEWXYZ, which is written in Fortran 77 to control the mesh motion as a function of valve and piston movements. The simulation begins at 2° BTDC, and ends at TDC of compression.

6.1 FLOW CONFIGURATION

Figure 6.1 shows the computational domain for the port/valve and cylinder assembly, corresponding at crank angle 90° ATDC. The mesh model contains 45,140 cells including the inlet plenum not shown in figure 6.1. Table 6.1 gives further details of engine geometry, and the operational speeds 1000 rpm and 3600 rpm for which results will be presented.

<table>
<thead>
<tr>
<th>Engine speed</th>
<th>3600/1000 rpm</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bore</td>
<td>93.65 mm</td>
</tr>
<tr>
<td>Stroke</td>
<td>93.65 mm</td>
</tr>
<tr>
<td>Connecting rod length</td>
<td>150.00 mm</td>
</tr>
<tr>
<td>Clearance at TDC</td>
<td>12.50 mm</td>
</tr>
<tr>
<td>Compression ratio</td>
<td>8.5</td>
</tr>
<tr>
<td>Intake valve opening</td>
<td>2.0° BTDC</td>
</tr>
<tr>
<td>Intake valve closure</td>
<td>28.0° ABDC</td>
</tr>
<tr>
<td>Maximum valve lift</td>
<td>12.00 mm</td>
</tr>
<tr>
<td>Valve diameter</td>
<td>43.00 mm</td>
</tr>
</tbody>
</table>

Table 6.1
6.2 BOUNDARY CONDITIONS AND MOVING MESH

During the induction stroke, boundaries are specified as two types: pressure boundary condition and wall boundary condition. When the intake valve is closed, all the boundaries of the in-cylinder domain are wall boundaries.

6.2.1 Boundary conditions

Pressure boundaries

A constant pressure boundary condition was specified at the inlet plane of the plenum during the induction stroke, where a pressure of zero relative to ambient is declared. The turbulence intensity and turbulence length scale were chosen to provide the turbulence quantities during the variation of the valve lift. A turbulent intensity of 5% of the instantaneous mean flow velocity at the plenum, and a length scale of 10% of the diameter of the port entrance plane is assumed. If the flow is inwardly directed, the boundary values of scalar quantities such as the turbulence parameters are ascribed as the specified values; while in the case of outflow, STAR-CD extrapolates the value of these variables from the upstream direction.

Wall boundaries

At all solid surfaces, the "law of the wall" conditions were considered, which has been described in Chapter 5, and surfaces are assumed to be isothermal.
6.2.2 Moving mesh

A moving mesh expands and contracts in relation to the valve lift in the areas which are adjacent to the port exit, between the fixed port wall and the moving valve. In the cylinder, the moving mesh consists of two zones, the first, between the cylinder head and the moving valve, and the second, between the valve head and the piston.

During the compression stroke, following inlet valve closure, the port is not considered in the computational domain. The moving mesh contracts between the cylinder head and the piston.

The new time level vertex coordinates are specified in a program - subroutine NEWXYZ. During movement, the mesh is controlled to avoid extremes of cell deformity: cell's aspect ratio is less than 10; nonorthogonality is not less than 45°; and the warpage is not less than 45° either.

6.3 Computational results

6.3.1 Flow structure

Induction

During the induction period, the velocity vector field pattern of the flow in the port upstream of the valve stem in the YZ plane shows little variation with crank angle, though magnitude increases with valve lift. It has been found that it is better to study the flow separation process in
the port in the region downstream of the valve stem and in the vicinity of the valve seat by mapping the static pressure distribution at the port/valve and cylinder surfaces, a method first reported by Hardenberg and Daudel (1975) in their experimental studies of intake ports using a steady flow test bench.

The static pressure at the surface will be influenced by the mean flow and clearly indicates zones of separation in a manner not always apparent from vector plots, without a large number of sectional diagrams. This can be seen from Figures 6.2(a) to (d), for valve lift of 6mm and 10 mm and engine speeds of 1000 rpm and 3600 rpm. Little difference in the general pressure pattern is observed in the port upstream of the valve stem, though velocity magnitude increases with valve lift and speed. In all cases, a recirculation zone is observed in the wake of the stem, at its junction with the valve port surface. However, the size of this zone increases with valve lift for the lower engine speed, and appears to be controlled by the overall pressure drop between the intake plenum and the engine cylinder, which decreases with increased opening of the valve and downward movement of the piston, and increases with engine speed. This can been seen in Figures 6.2(a)-(d).

In the cylinder, there is a strong interaction of the valve jet flow and the cylinder and piston surfaces. Figures 6.3(a) and (b) show the flow patterns in the YZ plane at crank angle 34° ATDC when valve lift is 4 mm for engine speeds of 1000 rpm and 3600 rpm respectively. At this very low lift, there is strong interaction of the valve jet flow with the cylinder head, and a strong jet flow down the cylinder wall and
along the piston surface created by the valve flow in the negative Y direction. The jet flow from the valve in the positive Y direction at lower engine speed almost attaches to the cylinder head surface because of the low pressure zone created adjacent to the head due to entrainment into the jet, the cylinder wall being sufficiently far not to cause downward deflection of the jet. These phenomena are also observed when the crank angle is 46° ATDC and the valve lift is 6 mm (see Figure 6.4(a)). As the valve opens, the jet trajectory moves towards the cylinder axis. However, for the same valve lifts of 4 mm and 6 mm, and the higher engine speed of 3600 rpm, the jet attachment is not as strong as for the 1000 rpm case, due here to the lower pressure in the cylinder causing movement of the jet away from the cylinder head.

To present the flow patterns in the cylinder continuously and clearly, the following presentation adopts a uniform grid structure and a fixed geometry scale for the flow structure figures since 46° ATDC to TDC. From Figure 6.4(b), the development of large scale vortex structures around the edges of the jet flow can be seen. The structure of the vortex flow at 76° ATDC for the 3600 rpm case (see 6.4(d)), is similar to that shown in 6.4(b), but the strength (circulation) of the vortices are much higher, as flow velocities through the valve have increased by a factor of about 4.

In the XZ plane, at 34° ATDC and 46° ATDC, (see Figures 6.5(a) to (b), and Figures 6.6(a) and (c)), there are much higher velocities of the jet flow at the valve exit and no attachment of the jet to the cylinder head. The reason is the presence of the cylinder wall, causing the jet to
deflect towards the cylinder axis, and create two sets of vortices, a large pair beneath the valve, and a pair at the cylinder head. In the lower speed 1000 rpm case, Figure 6.5(a) and Figure 6.6(a), the inlet valve jet flow is deflected towards the cylinder axis, more than in the high speed case, Figure 6.5(b) and Figure 6.6(c), because the jet flow velocity is approximately 4 times lower at 1000 rpm engine speed. Hence, there is less air entrainment from the region adjacent to the cylinder head, and pressure in this region is higher than in the 3600 rpm case. In Figure 6.6(b), the development of a strong vortex structure beneath the valve and near the cylinder head is shown, the former is very dependent on the presence of the piston. Little difference in the qualitative structure of the flow exists at the valve lift of 10 mm between 1000 rpm and 3600 rpm (Figures 6.6(b) and 6.6(d)), though the magnitudes of velocities are greater for the higher speed by a factor of about 4. Also there is little difference in the flow pattern at the highest valve lift 12 mm when the crank angle is 112° ATDC (Figures 6.7(a) to (d)).

At 172° ATDC when the piston is reaching maximum displacement, and the intake valve is beginning to close (8 mm lift), Figures 6.8(a) to (d), show a highly complex flow structure in the cylinder which appears to be controlled by the exit jet flow, the piston, and cylinder surfaces, particularly, at the lower engine speed. Where in Figure 6.8(a), the weak inlet jet flow in the positive Y direction is deflected under the valve to create a vortex. This flow is created partially by the wall jet flows along the cylinder and further deflected by the piston, shown in Figure 6.8(b), and the higher pressure at the cylinder head where a "dead zone" region appears to exist, which is shown in Figure 6.8(a).
Compared with the high speed case, presented in Figures 6.8(c) and 6.8(d), the flow is much more complex, in that deflection of the wall jet by the piston, which is shown in Figure 6.8(b) and 6.8(d), appears to create a secondary large vortex structure in the lower part of the cylinder in the 1000 rpm case, due to the much reduced momentum of the inlet jet flow, so that local velocities are about 8 times lower than at 3600 rpm. This seems to arise from the fact that at the lower engine speed, the air flow follows piston motion more closely, as at 172° ATDC the piston speed is very low. For higher engine speed, the momentum of the induced air mass causes it to continue entering the cylinder despite low piston motion. This can clearly be seen from contours of pressure across the intake valve - Figures 6.9(a) to 6.9(d) - where in the 1000 rpm case, very little change in pressure across the valve is observed compared with the 3600 rpm case.

During the early compression stroke (4° ABDC), once again, the air flow is observed to be following the piston motion for the low engine speed case, so that flow reversal through the intake port is apparent in Figures 6.10(a) and 6.10(b). Whereas, at 3600 rpm, an inflow to the cylinder is shown (see Figures 6.10(c) and 6.10(d)). At the higher engine speed, the flow structure in the YZ and XZ planes is now similar to that at 1000 rpm and 12° CA earlier which is 172° ATDC, but the flow velocities are higher.

At 16° ABDC and 1000 rpm, in the last phase of inlet valve closure, the effect of piston motion combined with the cylinder/piston wall flow in the XZ plane, is to cause a strong axial gas motion, which combined
with the outflow process through the valve produces what appears to be a deformation of the vortex motion under the valve in the YZ plane - Figure 6.11(a), whereas in the XZ plane the vortices under the valve appear to be displaced laterally towards the cylinder wall due to piston motion, which cause a downward flow towards the piston.

A secondary pair of vortices adjacent to the piston surface identified earlier at 172° ATDC is present, but in a weaker form. In the case of the higher engine speed, the outflow in the YZ plane, shown in Figure 6.11(c), has not as yet combined with the piston generated gas motion and wall flows to deform the vortex system under the valve, while in plane XZ illustrated in Figure 6.11(d), the pattern of flow is similar to that at 1000 rpm, but velocities are about three times as high.

**Compression**

The period of the compression stroke is from just after inlet valve closure to top dead centre (30° ABDC to TDC). The flow at 30° ABDC, illustrated in Figures 6.12(a) to (d), shows a general similarity of structure to that at 16° ABDC. However, in the low speed case, the effect of piston motion on axial components of flow is apparent, (see Figure 6.12(a)), as there is a reduction in flow velocities adjacent to the cylinder head. In the XZ plane, shown in Figure 6.12(b), a stratification of the flow in the cylinder occurs with a pair of vortices adjacent to the piston and a much weaker vortex flow at the top of the cylinder. At the higher engine speed, the flow in the YZ plane (Figure 6.12(c)), is little changed from that at 16° ABDC. However, in the XZ
plane, Figure 6.12(d), the displacement of the vortex pair upwards from the piston surface, due to the stronger flow near the cylinder head causes a flow down the cylinder wall and along the piston surface.

In the period 60° ABDC to TDC, the flow on the YZ plane clearly shows the "spin up" of the vortex structure and its evolution into a more complex form. Figures 6.13(a) to (e) show results for 1000 rpm in the YZ plane, Figures 6.14(a) to (e) are for 3600 rpm in the YZ plane. The small vortex in the left side of the cylinder head corner (in the negative y direction) disappears when the crank angle is 120° ABDC. The vortex in the right side of the cylinder head corner has shifted towards the piston surface as the piston approaches TDC. In the lower engine speed case, a pair of the jet-driven vortex persists during the whole compression. At higher engine speed, the pair of vortex broke at TDC due to stronger interaction with the piston motion.

In the XZ plane, in the same crank angle range to TDC, the flow stratifies and a spin up of the weaker vortex structure occurs. The flow on the XZ plane is shown for 1000 rpm and 3600 rpm respectively in Figures 6.15(a) to (e) and 6.16(a) to (e). During the period of 60° ABDC to 120° ABDC, the flow stratification is clearly shown. Towards the end of the compression process, large scale rotating flow structures are shown in Figures 6.15(d) and (e), and Figures 5.16(d) and (e).

6.3.2 Flow at the valve exit

The flow passing the valve exit as a jet, enters into the cylinder at a
velocity about 10 to 20 times the mean piston speed for the 1000 rpm engine speed case, and about 15 times for the 3600 rpm engine speed case.

Figure 6.17 and Figure 6.18 show the mean velocity components and turbulence intensities resolved radially at the valve exit on the y-z plane and the x-z plane respectively, which are normalised with mean piston speed. For both engine speed cases, the mean velocity profile varies obviously during early induction (20° to 60°), but the variation becomes smaller in mid induction when the crank angle is from 60° to 120°. Bigger gradients of the normalised radial mean velocity are presented on the XZ plane in the 1000 rpm case at crank angles of 34° and 46°. It explains once again the reason for the stronger jet attachment to the cylinder head at low lift for the 1000 rpm engine speed than the 3600 rpm engine speed.

Differences of the normalised turbulence intensity profiles exist between these two engine speed cases during early induction. Though the value of the peak turbulence intensity is similar, the ratio of the peak value to the lowest one is different. It is 2.0 for the 1000 rpm case, 2.6 for the 3600 rpm case. That means the average normalised turbulence intensity in the 1000 rpm case is higher. The location, at which the turbulence is highest, corresponds with the location of maximum gradients of the mean velocity. Therefore, the radial mean velocity gradients at 1000 rpm are bigger than those at 3600 rpm, causing a stronger vortex flow structure. During mid induction, the ratio tends to be similar, which is about 1.6 for both cases, but the profile shape is different. On the YZ plane, the normalised turbulence
intensity profiles are similar both in shape and magnitude.

6.3.3 Evaluation of turbulent field

During the induction process, the flow is presented only on the YZ plane from crank angles 46° ATDC to 4°ABDC, as this clearly demonstrates the generation of turbulence by the jet flow at the valve exit. Figures 6.19(a) to (e) show contours of turbulent kinetic energy for the crank angles of 46° ATDC to 4°ABDC at engine speed 1000 rpm. Figures 6.20(a) to (e) are for the higher engine speed case. During the first half of induction (see Figures 5.19(a) to (b) and Figures 5.20(a) to (b)), the development of shear layers at the edge of the jet flow is clearly seen, with a much less turbulent central core. The width of the shear layer can be seen to increase with increasing valve lift. The strong wall jet along the cylinder and piston surfaces nearest the valve axis is also clear. The generation of turbulence at the edges of the jet flow and the large scale vortex structures that it creates can be clearly observed in Figures 6.4(b), 6.19(b) and 6.20(b). Turbulence in the jet region decays rapidly during the second half of induction, with lower decay rates in the rest of the cylinder (see Figures 6.19(c) to (e) and Figures 6.20(c) to (e)). In the lower engine speed case, turbulence decay occurs earlier than in the higher engine speed case. During the whole induction stroke, the turbulent field shows a highly non-isotropic and non-homogeneous character.

Moving to the compression period, the results of turbulent kinetic energy from crank angle 120° ABDC to TDC in the YZ plane for 1000 rpm, are shown in Figures 6.21(a) to (c). The generation of high levels
of turbulence where there is interaction between the vortex flows in the upper part of the cylinder, is generated by the valve flow, and the upward axial flow created by a combination of piston movement and upward deflection of wall jet type flows. This is clearly shown in Figure 6.21(a) at 120° ABDC and Figure 6.13(c).

At 30° BTDC there has been an increase in the turbulent kinetic energy, due to dissipation at the edges of the vortex flow which has a high rotation speed because of its reduced size. Finally, at TDC, there is a large reduction of the turbulent kinetic energy as dissipation increases, and the ordered vortex structure, appears to be re-ordered into a larger complex flow structure. The same general arguments may be applied to the turbulent kinetic energy in the XZ plane, (see Figures 6.22(a) to (c)), and also to the results at 3600 rpm, though here magnitudes are higher (Figures 6.23(a) to (c) and Figures 6.24(a) to (c)). From the results it can be concluded that the turbulent field becomes more isotropic with progression of the compression process, but at TDC the turbulence is still not homogeneous. Figures 6.25(a) and (b) show the variation of the turbulent kinetic energy and the dissipation with variation of crank angle at engine speeds 1000 rpm and 3600 rpm respectively, for 5 points in the cylinder, at the mid plane of the instantaneous displacement volume. This clearly shows the effect of the incoming jet flow in the period up to maximum valve lift at 112° ATDC, followed by a period between valve closure and 60° BTDC when little change in turbulent kinetic energy occurs - though vortex spin up in this period is quite significant. In the period of 60° BTDC to 30° BTDC there is significant increase in the turbulent kinetic energy due to interaction of the vortex flows and the piston generated flow. In
the final period of the compression stroke, there is sharp decrease in the turbulent kinetic energy due to increased dissipation, and the approximate equality of magnitude at points A, B, C and D indicating the near isotropic nature of the turbulence. At TDC, the turbulence intensity normalised with mean piston speed is approximately 0.5 for both engine speeds.

6.3.4 Comparison with steady flow

Figure 6.26 shows the comparisons of the mean velocity and turbulence intensity components at the inlet valve exit, which were resolved radially in the YZ and XZ planes between the steady flow and engine simulations. The mean velocities and turbulence intensities for the unsteady flow were normalised with mean piston speed. The steady flow results were normalised with the mean flow velocity based on the cylinder diameter for a given valve lift, and mass flow rate corresponding to a fixed pressure drop across the inlet plenum and exit from the cylinder. The non-dimensionalised mean velocity profiles are not similar between the steady flow and engine simulations, both in magnitude and profiles shape. There is greater asymmetry about the valve axis in the flow profiles and their magnitude between the engine and steady flow cases.

Comparing the static pressure at the surface of the port/valve and cylinder assembly under steady flow condition for valve lift of 6mm and 10 mm shown in Figure 5.8, the region of flow separation in the wake of the stem is smaller in the case of engine simulation, due to the higher overall pressure drop between the plenum and cylinder as
mentioned earlier. It should also be noticed that there is a significant asymmetry in the pressure distribution at the valve seat in the engine case compared with the steady flow tests, due in the engine to flow interaction with the piston surface.

The asymmetric structure of the in-cylinder flow about the valve axis is particularly noticeable for the high valve lift of 10 mm, shown in Figure 5.11 and Figure 5.12, and is caused by the presence of the piston deflecting the strong wall jet flow from the valve exit nearest the cylinder wall, see Figure 6.4(b). This results in a different flow structure at the valve exit in the positive Y direction. In the engine simulation the effect of speed change is small on the magnitude and profiles of the mean velocity components, and the small differences that are observed, are at the lower valve lifts of 4 and 6 mm. In regard to turbulence intensity profiles, these are generally similar in all cases, showing a higher intensity at the edges of the jet flow through the valve.

6.4 DISCUSSION

The results presented above show the strong interaction that exists between the jet flows created at the exit of the valve, which are highly asymmetric, because of the vicinity of the valve to part of the cylinder wall in the negative Y direction. This wall jet flow is extremely strong during the valve opening. It interacts with the piston, to create a three-dimensional; vortex flow situation in the cylinder which evolves further, during the induction process, and is enhanced during compression creating higher turbulence levels, finally to break down to
a more complex flow structure at TDC, creating at this point a near isotropic turbulence field, which, however, is not homogeneous. The features of the flow observed here coincide with general features observed in the literature reviewed. However, unlike the study of Echian and Hoult (1979), the observations here indicate a forced vortex flow structure during induction, consistent with the observations of Khalighi (1990).

There is a difference from El Tahry et al in the observations of the second phase of the induction process (60° to 120°) in regard to agreement between steady flow and engine simulations, where strong effects of the piston and valve location are found to create highly complex flow structures as discussed earlier.

The results in the present work show the behaviour of jet "flapping" as Suen (1992) had observed. However, jet attachment/detachment phenomena are unstable processes, and therefore in the engine experiments are likely to produce the periodic impairment of induction flow, cannot be predicted in the present model of the flow structure. Echian and Hoult also observed the bistable "flapping" of the intake jet flow and identified it as a possible cause of large cycle to cycle variation. The author would concur these observations, because a steady turbulent jet flow is intermittent in its large scale structure (Hinze, 1975), a similar situation must also exist in the engine inlet jet flow.
Figure 6.1 The computational domain
Figure 6.2(a) Surface pressure for the valve lift of 6 mm
and engine speed 1000 rpm

Figure 6.2 (b) Surface pressure for the valve lift of 10 mm
and engine speed 1000 rpm
Figure 6.2(a) Surface pressure for the valve lift of 6 mm and engine speed 1000 rpm

Figure 6.2 (b) Surface pressure for the valve lift of 10 mm and engine speed 1000 rpm
Figure 6.2(c) Surface pressure for the valve lift of 6 mm and engine speed 3600 rpm

Figure 6.2(d) Surface pressure for the valve lift of 10 mm and engine speed 3600 rpm
Figure 6.2(c) Surface pressure for the valve lift of 6 mm and engine speed 3600 rpm

Figure 6.2(d) Surface pressure for the valve lift of 10 mm and engine speed 3600 rpm
### PROSTAR 2.1

**7.FEB-04**

**PRESSURE**

<table>
<thead>
<tr>
<th>LOCAL MX = 3322</th>
<th>LOCAL MN = -3265E+05</th>
</tr>
</thead>
<tbody>
<tr>
<td>X</td>
<td>Y</td>
</tr>
</tbody>
</table>

#### Values

<table>
<thead>
<tr>
<th>Value</th>
<th>Color</th>
</tr>
</thead>
<tbody>
<tr>
<td>-3322</td>
<td>Red</td>
</tr>
<tr>
<td>-6417</td>
<td>Orange</td>
</tr>
<tr>
<td>-9607</td>
<td>Yellow</td>
</tr>
<tr>
<td>-1170E+06</td>
<td>Green</td>
</tr>
<tr>
<td>-1380E+06</td>
<td>Blue</td>
</tr>
<tr>
<td>-1590E+06</td>
<td>Purple</td>
</tr>
<tr>
<td>-1790E+06</td>
<td>Purple</td>
</tr>
<tr>
<td>-2000E+06</td>
<td>Blue</td>
</tr>
<tr>
<td>-2210E+06</td>
<td>Blue</td>
</tr>
<tr>
<td>-2420E+06</td>
<td>Blue</td>
</tr>
<tr>
<td>-2630E+06</td>
<td>Blue</td>
</tr>
<tr>
<td>-2840E+06</td>
<td>Blue</td>
</tr>
<tr>
<td>-3050E+06</td>
<td>Blue</td>
</tr>
<tr>
<td>-3260E+06</td>
<td>Blue</td>
</tr>
</tbody>
</table>
Figure 6.3(a) Flow pattern at 34° ATDC for engine speed 1000 rpm

Figure 6.3(b) Flow pattern at 34° ATDC for engine speed 3600 rpm
Figure 6.4(a) Flow pattern at 46° ATDC for engine speed 1000 rpm

Figure 6.4(b) Flow pattern at 76° ATDC for engine speed 1000 rpm
Figure 6.4(c) Flow pattern at 46° ATDC for engine speed 3600 rpm

Figure 6.4(d) Flow pattern at 76° ATDC for engine speed 3600 rpm
Figure 6.5(a) Flow pattern at 34° ATDC for engine speed 1000 rpm

Figure 6.5(b) Flow pattern at 34° ATDC for engine speed 3600 rpm
Figure 6.6(a) Flow pattern at 46° ATDC for engine speed 1000 rpm

Figure 6.6(b) Flow pattern at 76° ATDC for engine speed 1000 rpm
Figure 6.6(c) Flow pattern at 46° ATDC for engine speed 3600 rpm

VELOCITY MAGNITUDE
M/SEC
LOCAL MX= 238.9
LOCAL MN= 0.7477

Figure 6.6(d) Flow pattern at 76° ATDC for engine speed 3600 rpm
Figure 6.7(a) Flow pattern in the y-z plane at 112° ATDC for 1000 rpm engine speed

Figure 6.7(b) Flow pattern in the x-z plane at 112° ATDC for 1000 rpm engine speed
Figure 6.7(c) Flow pattern in the y-z plane at 112° ATDC for 3600 rpm engine speed

Figure 6.7(d) Flow pattern in the x-z plane at 112° ATDC for 3600 rpm engine speed
Figure 6.8(a) Flow pattern in the y-z plane at 172° ATDC for 1000 rpm engine speed

Figure 6.8(b) Flow pattern in the x-z plane at 172° ATDC for 1000 rpm engine speed
Figure 6.8(c) Flow pattern in the y-z plane at 172° ATDC for 3600 rpm engine speed

Figure 6.8(d) Flow pattern in the x-z plane at 172° ATDC for 3600 rpm engine speed
Figure 6.9(a) Pressure distribution in the y-z plane at 172° ATDC for 1000 rpm engine speed

Figure 6.9(b) Pressure distribution in the x-z plane at 172° ATDC for 1000 rpm engine speed
Figure 6.9(a) Pressure distribution in the y-z plane at 172° ATDC for 1000 rpm engine speed

Figure 6.9(b) Pressure distribution in the x-z plane at 172° ATDC for 1000 rpm engine speed
Figure 6.9(c) Pressure distribution in the y-z plane at 172° ATDC for 3600 rpm engine speed

Figure 6.9(d) Pressure distribution in the x-z plane at 172° ATDC for 3600 rpm engine speed
Figure 6.9(c) Pressure distribution in the y-z plane at 172° ATDC for 3600 rpm engine speed

Figure 6.9(d) Pressure distribution in the x-z plane at 172° ATDC for 3600 rpm engine speed
Figure 6.10(a) Flow pattern in the y-z plane at $4^\circ$ ABDC for engine speed 1000 rpm

Figure 6.10(b) Flow pattern in the x-z plane at $4^\circ$ ABDC for engine speed 1000 rpm
Figure 6.10(c) Flow pattern in the y-z plane at 4° ABDC for engine speed 3600 rpm

Figure 6.10(d) Flow pattern in the x-z plane at 4° ABDC for engine speed 3600 rpm
Figure 6.11(a) Flow pattern in the y-z plane at 16° ABDC for engine speed 1000 rpm

Figure 6.11(b) Flow pattern in the x-z plane at 16° ABDC for engine speed 1000 rpm
Figure 6.11(c) Flow pattern in the y-z plane at 16° ABDC for engine speed 3600 rpm

Figure 6.11(d) Flow pattern in the x-z plane at 16° ABDC for engine speed 3600 rpm
Figure 6.12(a) Flow pattern in the y-z plane at 30° ABDC for engine speed 1000 rpm

Figure 6.12(b) Flow pattern in the x-z plane at 30° ABDC for engine speed 1000 rpm
Figure 6.12(c) Flow pattern in the y-z plane at 30° ABDC for engine speed 3600 rpm

Figure 6.12(d) Flow pattern in the x-z plane at 30° ABDC for engine speed 3600 rpm
Figure 6.13(a) Flow pattern in the y-z plane at 60° ABDC for engine speed 1000 rpm

Figure 6.13(b) Flow pattern in the y-z plane at 90° ABDC for engine speed 1000 rpm
Figure 6.13(c) Flow pattern in the y-z plane at 120° ABDC for engine speed 1000 rpm

Figure 6.13(d) Flow pattern in the y-z plane at 30° BTDC for engine speed 1000 rpm

Figure 6.13(e) Flow pattern in the y-z plane at TDC for engine speed 1000 rpm
Figure 6.14(a) Flow pattern in the y-z plane at 30° ABDC for engine speed 3600 rpm

Figure 6.14(b) Flow pattern in the y-z plane at 60° ABDC for engine speed 3600 rpm
Figure 6.14(c) Flow pattern in the y-z plane at 120° ABDC for engine speed 3600 rpm

Figure 6.14(d) Flow pattern in the y-z plane at 30° BTDC for engine speed 3600 rpm

Figure 6.14(e) Flow pattern in the y-z plane at TDC for engine speed 3600 rpm
Figure 6.15(a) Flow pattern in the x-z plane at 30° ABDC for engine speed 1000 rpm

Figure 6.15(b) Flow pattern in the x-z plane at 60° ABDC for engine speed 1000 rpm
Figure 6.15(c) Flow pattern in the x-z plane at 120° ABDC for engine speed 1000 rpm

Figure 6.15(d) Flow pattern in the x-z plane at 30° BTDC for engine speed 1000 rpm

Figure 6.15(e) Flow pattern in the x-z plane at TDC for engine speed 1000 rpm
Figure 6.16(a) Flow pattern in the x-z plane at 30° ABDC for engine speed 3600 rpm

Figure 6.16(b) Flow pattern in the x-z plane at 60° ABDC for engine speed 3600 rpm
Figure 6.16(c) Flow pattern in the x-z plane at 120° ABDC for engine speed 3600 rpm

Figure 6.16(d) Flow pattern in the x-z plane at 30° BTDC for engine speed 3600 rpm

Figure 6.16(e) Flow pattern in the x-z plane at TDC for engine speed 3600 rpm
At 1000 rpm engine speed

Lν = 4 mm

Lν = 6 mm

Lν = 10 mm

Figure 6.17 Normalised radial mean velocity and turbulence intensity on the y-z plane at the valve exit

○ - normalised radial mean velocity component   ● - normalised turbulence intensity

At 3600 rpm engine speed
$L_v = 4\text{ mm}$

$L_v = 6\text{ mm}$

$L_v = 10\text{ mm}$

At 1000 rpm engine speed

At 3600 rpm engine speed

Figure 6.18 Normalised radial mean velocity and turbulence intensity on the x-z plane at the valve exit

$\circ$ - normalised radial mean velocity component

$\bullet$ - normalised turbulence intensity
Figure 6.19(a) Turbulent kinetic energy at 46° ATDC for engine speed 1000 rpm

Figure 6.19(b) Turbulent kinetic energy at 76° ATDC for engine speed 1000 rpm
Figure 6.19(a) Turbulent kinetic energy at 46° ATDC for engine speed 1000 rpm

Figure 6.19(b) Turbulent kinetic energy at 76° ATDC for engine speed 1000 rpm
PROSTAR 2.1
2-FEB-94
TURB KINETIC ENERGY
M**2/S**2
LOCAL MX= 34.19
LOCAL MN=0.1819

PROSTAR 2.1
2-FEB-94
TURB KINETIC ENERGY
M**2/S**2
LOCAL MX= 45.97
LOCAL MN=0.1941
Figure 6.19(c) Turbulent kinetic energy at 112° ATDC
for engine speed 1000 rpm

Figure 6.19(d) Turbulent kinetic energy at 172° ATDC
for engine speed 1000 rpm
Figure 6.19(c) Turbulent kinetic energy at 112° ATDC for engine speed 1000 rpm

Figure 6.19(d) Turbulent kinetic energy at 172° ATDC for engine speed 1000 rpm
Figure 6.19(e) Turbulent kinetic energy at $4^\circ$ ABDC for engine speed 1000 rpm

Figure 6.20(a) Turbulent kinetic energy at $46^\circ$ ATDC for engine speed 3600 rpm
Figure 6.19(e) Turbulent kinetic energy at 4° ABDC for engine speed 1000 rpm

Figure 6.20(a) Turbulent kinetic energy at 46° ATDC for engine speed 3600 rpm
Figure 6.20(b) Turbulent kinetic energy at 76° ATDC
   for engine speed 3600 rpm

Figure 6.20(c) Turbulent kinetic energy at 112° ATDC
   for engine speed 3600 rpm
Figure 6.20(b) Turbulent kinetic energy at 76° ATDC for engine speed 3600 rpm

Figure 6.20(c) Turbulent kinetic energy at 112° ATDC for engine speed 3600 rpm
Figure 6.20(d) Turbulent kinetic energy at 172° ATDC
for engine speed 3600 rpm

Figure 6.20(e) Turbulent kinetic energy at 4° ABDC
for engine speed 3600 rpm
Figure 6.20(d) Turbulent kinetic energy at 172° ATDC
for engine speed 3600 rpm

Figure 6.20(e) Turbulent kinetic energy at 4° ABDC
for engine speed 3600 rpm
TURB KINETIC ENERGY

PROSTAR 2.1
17-FEB-94

M"2/S"2

LOCAL MX= 402.0
LOCAL MN= 4.219

PROSTAR 2.1
17-FEB-94

M"2/S"2

LOCAL MX= 205.6
LOCAL MN= 2.692
Figure 6.21(a) Turbulent kinetic energy at 120° ABDC
for engine speed 1000 rpm

Figure 6.21(b) Turbulent kinetic energy at 30° BTDC
for engine speed 1000 rpm
Figure 6.21(a) Turbulent kinetic energy at 120° ABDC for engine speed 1000 rpm

Figure 6.21(b) Turbulent kinetic energy at 30° BTDC for engine speed 1000 rpm
Figure 6.21(c) Turbulent kinetic energy at TDC
for engine speed 1000 rpm

Figure 6.22(a) Turbulent kinetic energy at 120° ABDC
for engine speed 1000 rpm
Figure 6.21(c) Turbulent kinetic energy at TDC for engine speed 1000 rpm

Figure 6.22(a) Turbulent kinetic energy at 120° ABDC for engine speed 1000 rpm
PROSTAR 2.1
16-FEB-94
TURB KINETIC ENERGY
M^2/S^2
LOCAL MX= 4.110
LOCAL MN=0.1901E-01

PROSTAR 2.1
16-FEB-94
TURB KINETIC ENERGY
M^2/S^2
LOCAL MX= 4.367
LOCAL MN=0.1086
Figure 6.22(b) Turbulent kinetic energy at 30° BTDC
for engine speed 1000 rpm

Figure 6.22(c) Turbulent kinetic energy at TDC
for engine speed 1000 rpm
Figure 6.22(b) Turbulent kinetic energy at 30° BTDC for engine speed 1000 rpm

Figure 6.22(c) Turbulent kinetic energy at TDC for engine speed 1000 rpm
Figure 6.23(a) Turbulent kinetic energy at 120° ABDC
for engine speed 3600 rpm

Figure 6.23(b) Turbulent kinetic energy at 30° BTDC
for engine speed 3600 rpm
Figure 6.23(a) Turbulent kinetic energy at 120° ABDC for engine speed 3600 rpm

Figure 6.23(b) Turbulent kinetic energy at 30° BTDC for engine speed 3600 rpm
Figure 6.23(c) Turbulent kinetic energy at TDC
for engine speed 3600 rpm

Figure 6.24(a) Turbulent kinetic energy at 120° ABDC
for engine speed 3600 rpm
Figure 6.23(c) Turbulent kinetic energy at TDC for engine speed 3600 rpm

Figure 6.24(a) Turbulent kinetic energy at 120° ABDC for engine speed 3600 rpm
Figure 6.24(b) Turbulent kinetic energy at 30° BTDC
for engine speed 3600 rpm

Figure 6.24(c) Turbulent kinetic energy at TDC
for engine speed 3600 rpm
Figure 6.24(b) Turbulent kinetic energy at 30° BTDC for engine speed 3600 rpm

Figure 6.24(c) Turbulent kinetic energy at TDC for engine speed 3600 rpm
Figure 6.25(a) Evolution of turbulent kinetic energy and dissipation rate at 1000 rpm.
Figure 6.25(b) Evolution of turbulent kinetic energy and dissipation rate at 3600 rpm.
$L_v = 4 \text{ mm}$

$L_v = 6 \text{ mm}$

$L_v = 10 \text{ mm}$

Figure 6.26 Normalised radial mean velocity and turbulence intensity at the valve exit

--- steady flow,  
----- transient flow at 1000 rpm engine speed,  
----- transient flow at 3600 rpm engine speed
7.1 CONCLUSIONS

The research efforts presented herein have focused on the simulation of turbulent flow in the curved inlet port and the cylinder under both steady and unsteady states using a Computational Fluid Dynamics method. The predicted flow characteristics were confirmed experimentally with static pressure measurements at the port/valve surfaces and LDA measurements of the velocity components. The primary aims have been to interpret the complex port/cylinder flow behaviour and provide detailed information for engine design. In this regard it has been confirmed with CFD what has been known from empirical design that the acceleration of the flow into valve curtain by using a valve slightly smaller in diameter than that of the port, increases the discharge coefficient and reduces flow separation.

The major findings emerging from the present study have been discussed in Chapters 5 and 6 in relation to the flow states. The major conclusions drawn from the work are presented below.

7.1.1 Predicted flow features in the port and the cylinder

The present numerical predictions provide a large amount of information to build a clear picture of port/cylinder flow structures.
Flow in the inlet port

- Steady flow

The flow upstream of the valve stem and the curved port bend is independent of the valve lift, because the flow is in a favourable condition without separation.

There is a separation region between the valve stem and the concave port wall, caused by the adverse pressure gradient. The separation is affected by the valve lift: the higher the valve lift, the larger the separation region. Separation also exists in the area adjacent to the convex port wall. This separation becomes stronger with increasing valve lift due to a larger zone of adverse pressure gradient.

- Transient flow

The flow pattern upstream of the valve stem and the port bend is hardly affected by the valve and piston movements. Downstream of the stem and the port bend, the flow is very much affected by both the valve and piston motions. The size of the separation zone in the wake of the valve stem increases with increased opening of the valve and downward movement of the piston, and decreases with the engine speed.

Compared with the steady flow at the same valve lift, the separation zone in the valve stem wake is smaller due to increase of the overall pressure drop between the intake plenum and the engine cylinder. The
higher the engine speed, the smaller the separation zone. It is caused by higher overall pressure drop at the higher engine speed.

Flow through the valve passage and at the valve exit

- **Steady flow**

The flow pattern passing through the valve passage varies with the valve lift due to substantial variation of the pressure distribution along the surfaces of both the valve head and valve seat.

At low valve lifts \((L_v/D_v < 0.07)\), the flow remains attached to both sides of the valve passage, and emerges as a jet with higher speed at the valve exit, attaching to the cylinder head surface. The discharge coefficient \(C_d\) increases with the valve lift. At intermediate lifts \((L_v/D_v = 0.09)\), the flow separates from the inner edge of the valve head, and \(C_d\) decreases abruptly. In the range of \(L_v/D_v = 0.09 - 0.14\), the flow separation is found at the outer edge of the valve head, but the size of the separation region remains approximately constant, and hence \(C_d\) increases with the valve lift. In addition, the jet detaches from the cylinder head surface. At high lifts, \(L_v/D_v > 0.14\), the flow separates from the valve head because of the increased adverse pressure gradient region which extends all over the valve head surface. When the valve lift reaches \(L_v/D_v = 0.023\), the flow separates from both sides of the valve passage to form a free jet, and results in a decrease of the discharge coefficient.
• **Transient flow**

The transient flow patterns in the valve passage are similar to those at steady state. However, there are significant differences in the non-dimensionalised mean velocity profiles at the valve exit, both in terms of magnitude and profile shape. There is greater asymmetry about the valve axis in the profiles and their magnitude in the transient case than for the steady flow, due to the presence of the piston. The piston deflects the strong wall jet flow from the valve exit nearest the cylinder wall (in the negative y direction), and results in a different flow structure at the valve exit in the positive y direction.

The effect of engine speed change is small on the non-dimensionalised magnitude and profiles of the mean velocity components. During the early phase of the induction process, small differences are observed.

**In-cylinder flow**

• **Steady flow**

A highly three-dimensional vortical flow pattern is created by the interaction of the intake jet with the cylinder surfaces. In the y-z plane, two pairs of counter-rotating vortices are formed during intermediate and high valve lifts: one adjacent to the cylinder head, the other beneath the valve head centred near the valve axis. At lower lift, there is no vortex in the right side of the cylinder head (in the positive y direction), due to the attachment of the jet to the cylinder head surface. In the x-z plane, two pairs of counter-rotating vortices exist with all
The position and size of the vortex are dependent on the jet speed and the jet angle with the cylinder head, which is affected by the port geometry, its location relative to the cylinder, and the valve lift.

- **Transient flow**

Because of the downward motion of the piston, the jet flow along the cylinder wall closest to the valve, has very high velocities. The strong wall jet interacts with the piston, to create a highly three-dimensional recirculation flow in the cylinder, which evolves further during the induction process and is enhanced during compression, creating higher turbulence levels. This large scale flow structure is highly dependent on the port geometry and its location relative to the cylinder.

At high engine speeds, the large scale flow structure created by the inlet valve jet and its interaction with the piston and cylinder surfaces is dominant, and the piston motion is of little consequence. At lower engine speeds, the large scale flow structure is also observed, but the gas flow also responds to piston motion, causing stratification of the in-cylinder flow late in the induction stroke.

The vortex structure in the flow persists into the compression stroke, and is augmented, before a final rearrangement of the flow structure at TDC, when the large-scale circulation finally breaks down to a more complex small-scale turbulence.
The turbulent kinetic energy is created principally at the edges of the inlet valve jet and cylinder wall jet flows during the early induction stroke. Turbulent kinetic energy levels are high in the late compression period, around 30° BTDC when rapid rearrangement of the vortex flow structure occurs. At TDC is approached, the turbulent kinetic energy levels decay sharply due to increased viscous dissipation. At TDC, the turbulence field attains a state that is nearly isotropic, but not homogeneous. The normalised turbulence intensity with mean piston speed is approximately 0.5.

The attachment/detachment of the inlet valve jet flow to/from the cylinder head has a strong effect on the in-cylinder flow structure, though the unstable nature of this phenomenon cannot be modelled here.

There are significant differences between the results from steady flow and transient simulations, reflecting the strong influence of the valve and piston movements.

7.1.2 Effect of port geometry

The present study shows, by both numerical simulation and static pressure measurements, that a reduced valve diameter of about 11% of the original forms a convergent/divergent flow passage between the valve head and the valve seat, which accelerates the flow up to the throat of the passage, followed by a better pressure recovery in the divergent section, and achieves a higher discharge coefficient.
This research clarified earlier studies of port designs by trial and error processes, and interpreted the flow behaviour in the modified valve passage by prediction using CFD method.

7.1.3 Mapping of the static pressure distribution

Mapping the static pressure distribution at the port/valve and the cylinder surfaces is an effective method to study flow separation processes in the region downstream of the valve stem and in the vicinity of the valve seat, because the static pressure will be influenced by the mean flow and clearly indicates zones of separation in a manner not always apparent from vector plots, without a large number of sectional diagrams.

The predicted static pressure plots presented in this thesis provide detailed information of the mean flow features in the port and the cylinder, including all the separation regions. Those plots illustrated for the modified port case, show how separation regions decreased in size, and how the favourable mean flow characteristics were produced within the convergent/divergent flow passage.

The static pressure measurements confirmed the computational results. It is a simple and inexpensive tool for understanding port flow behaviour.

7.1.4 CFD contributions

All the conclusions mentioned above demonstrate the contribution of
CFD to the production of detailed information concerning the flow behaviour of the port/valve/cylinder assembly not possible in the past.

The steady flow computational results were verified by LDA experimental data and there was generally acceptable agreement between them, giving confidence in the present work.

Because the LDA measurement technique is restricted by the complex engine geometry and operating conditions, CFD is becoming more important in the engine design process. Not only more details we can obtain, but also some experimental error involved we are enable to correct.

7.2 RECOMMENDATIONS

From the work presented in this thesis, we see clear pictures of details of the recirculation patterns within the practical generic port and the cylinder, and the influences of particular parameters of the port geometry on flow structures. We are confident that, using the computational technique, our aim to optimise the port design for obtaining high volumetric efficiency over a wide engine speed range and appropriate flow motion can be achieved.

Facing the engine industry today, the challenges we have to meet are low emission and high performance. Four valve engines are therefore introduced to be the mid 1990s engines in the automotive field. The key characteristic of using two inlet valves is the large intake flow area, consequently, a high volumetric efficiency, even at high speeds.
Four valve petrol engines usually have a pent-roof combustion chamber. Tumble (barrel swirl) is generated within the cylinder during the induction process and leads to a higher level of turbulence at the end of compression, resulting a fast burn combustion system. Effects of tumble are as follows: reduced ignition delay, reduced burn duration, lower levels of cycle-by-cycle variation, and a greater tolerance to exhaust gas recirculation, leading to scope for reducing part load fuel consumption and brake specific emission. But, too much tumble will lead to an increase in the unburnt hydrocarbon emissions and an increase in the brake specific fuel consumption (de Bore et al, 1990).

To produce the required levels of air flow and tumble is the challenge of inlet port/valve design. There are many possible ways to arrange the two ports, such as: one helical port and one straight port; one straight port and one curved port; two helical ports. When the shapes of the ports are decided, there are still several ways to arrange their positions and orientations.

Furthermore, in order to achieve the features of high engine performance, such as flat torque characteristics with favourable low end behaviour and high power output, low engine - out emission under steady and transient operating conditions, a system for controlling the burn rate is required. This is because a fast burn combustion system will just benefit the engine at part load. For the engine at full load, if the combustion is too fast, there will be increased levels of combustion noise. Hence to obtain high volumetric efficiency over a wide speed range is desirable. The most effective way is to
realise a variable charge motion, which can be achieved using air flow reduction passing one of the two intake valves at part load by port throttling or port disablement. Port throttling can be employed using a control valve or by the provision of exhaust gas recirculation.

CFD techniques now offer the possibility of predicting the fluid flow with definitive designs of inlet port/valve assemblies, port throttling or port disablement for producing the charge motion to be adopted according to engine speed and load.

With rapid development of powerful digital super computers, and computational codes, the prediction can be more accurate. For example, the latest released version of STAR-CD, (version 2.2), has addressed a recently-developed Re-normalisation Group Model (RNG) which is a derivation of the k-ε model, and has been shown to achieve more accurate results for certain cases with flow separations, such as the flow over a two-dimensional square-shaped bluff body (Boysan, 1993). Also, the Two-layer turbulence model is installed. Researchers now have several choices to obtain more accurate predictions. In addition, combustion models are installed, and multi-phase flows with solid particles are able to be calculated. Therefore, much more accurate predictions of the fluid flow for the whole engine cycle are feasible now. The ultimate goal of using CFD as a basic tool in advanced engine design process will be achieved soon.
REFERENCES

Aita, S., Tabbal, A., Munck, G., Montmayeur, N., Takenaka, Y.,
Aoyagi, Y., and Obana, S.,
"Numerical Simulation of Swirling Port-Valve-Cylinder Flow in

Amann, C. A.,
"Classical Combustion Diagnostics for Engine Research", SAE Paper

Amsden, A. A., and Harlow, F. H.,
"The SMAC method: A Numerical Technique for Calculating
Incompressible Fluid Flows", Los Alamos Sci. Lab. Report, LA -
4370, 1970.

Anand, W. J. D., and Roe, G. E.,
Gas Flow in the Internal Combustion Engine, Yeovil: Foulis,
1974.

Arcoumanis, C., Begleris, P., Gosman, A. D., and Whitelaw, J. H.,
"Measurements and Calculations of the Flow in a Research Diesel

Arcoumanis, C., and Whitelaw, J. H.,
"Fluid Mechanics of Internal Combustion Engines _ A Review",
Argueyrolles, B., Taghavi, R., and Zellat, M.,

Barnes-Moss, H. W.,

Bicen, A. F., Vafidis, C., and Whitelaw, J. H.,

Bradshaw, P.,

Bradshaw, P.,

Brandstätter, W., Johns, R. J. R., and Wigley, G.,

Boussinesq, J.,
Boysan, H. F.,

Castro, I. P. and Bradshaw, P.,

Castro, I. P., and Jones, J. M.,

Chen, K. Y.,

Cheung, R. S. W., Nadarajah, S., Tindal, M. J., and Yianneskis, M.,

De Boer, C. D., Johns, R. J. R., Grigg, D. W., Train, B. M., Denbratt, I. P., and Linna, J-R,
Demirdzic, I., Peric, M., and Yianneskis, M.,
"Numerical Predictions of the Mean Flow and Turbulence in an
Axisymmetric Port and Their Assessment Against Experimental Data",
Proceeding of the xixth International Symposium on Heat and Mass
Transfer in Gasoline and Diesel Engines, Dubrovnik, Yugoslavia, 1987.

Dennison, E. S., Kuchler, T. C., and Smith, D. W., "Experiments on
Engrs. (Oil Gas Pwr.), 53-6, pp. 79-97, 1931.

Dupont, A.,
"Using a Three-Dimensional Aerodynamic Code as a Guide for
Choosing Between Various S. I. Engine Chamber Designs",
Proceedings of the Second International Conference on Super-

El Tahry, S. H.,
"A Numerical Study on the Effects of Fluid Motion at Inlet-Valve
Closure on Subsequent Fluid Motion in a Motored Engine",

El Tahry, S. H.,
"k-ε Equation for Compressible Reciprocating Engine Flows",

El Tahry, S. H.,
"Application of a Reynolds Stress Model to Engine-Like Flow
Calculations", in Flows in Internal Combustion Engines - II,
El Tahry, S. H.,

El Tahry, S. H., Khalighi, B., and Kuziak, W. R.,

Ekchian, A., and Hoult, D. P.,

Errera, M. P.,

Errera, M. P., Labbe, J., and Jerot, A.,

Gibson, M. M., and Rodi, W.,
Gosman, A. D., Jones, R. J. R., and Watkins, P. A.,

Gosman, A. D., Tsui, Y. Y., and Vafidis, C.,

Gosman, A. D., and Ahmed, A. M. Y.,

Harlow, F. H., and Welch, J. E.,

Hanjalic, K., and Launder, B. E.,

Hanjalic, K., and Launder, B. E.,
Hardenberg, H. O., and Daudel, H.,

Haworth, D. C., El Tahry, S. H., Huebler, M. S., and Chang, S.,

Hendricks, C. J., and Brighton, J. A.,

Henriot, S., Le Coz, J. F., and Pinchon, P.,

Heywood, J. B.,

Hinze, J. O.,
Huang, P. G., Launder, B. E., and Leschziner, M. A.,

Hunter, D. U.,

Issa, R. I.,

Issa, R. I., Gosman, A. D., and Watkins, A. P.,

Isshiki, Y., Shimamoto, Y., and Wakisaka, T.,
Jones, W. P.,

Johns, W. P., and Launder, B. E.,

Johns, R. J. R.,

Kastner, L. J., William, T. J., and White, J. B.,


Khalighi, B.,
Lam, C. K. G., and Bremhorst, K. A.,
"A Modified Form of the k-\(\varepsilon\) Model for Predicting Wall Turbulence",

Launder, B. E., and Spalding, D. B.,

Launder, B. E., and Sharma, B. I.,
"Application of Energy-Dissipation Model of Turbulence to Calculation of Flow Near a Spinning Disc",

Launder, B. E., and Spalding, D. B.,
"The Numerical Computation of Turbulent Flows",

Launder, B. E., Reece, G. J., and Rodi, W.,
"Progress in the Development of a Reynolds-Stress Turbulence Closure",

Leonard, B. P.,
"A Stable and Accurate Convective Modelling Procedure Based on Quadratic Upstream Interpolation",
Leschziner, M. A., and Rodi, W.,

Leschziner, M. A.,

Naitoh, K., Fujii, H., Urushihara, T., and Takagi, Y., Kuwahara, K.,

Naser, J. A.,

Norris, H. L., and Reynolds, W. C.,

Patankar, S. V., and Spalding, D. B.,
Patankar, S. V., Pratap, V. S., and Spalding, D. B.,
"Prediction of Turbulent Flow in Curved Pipes", J. Fluid Mech.,

Patankar, S. V.,
**Numerical Heat Transfer and Fluid Flow**, Hemisphere,

Patankar, S. V.,
"A Calculation Procedure for Two-Dimensional Elliptic Situations",

Patankar, S. V.,
"Recent Developments in Computational Heat Transfer",

Patel, V. C., Rodi, W., and Scheuerer, G.,
"Turbulence Models for Near-Wall and Low Reynolds Number Flows:

Patel, M. K., and Markatos, N. C., and Cross, M.,
"Method of Reducing False-Diffusion Errors in Convection-Diffusion

Patel, M. K., and Markatos, N. C.,
"An Evaluation of Eight Discretization Schemes for Two-dimensional
Convection-Diffusion Equations", International Journal for Numerical
Patel, M. K., Cross, M., and Markatos, N. C., Mace, A. C. H.,

Reynolds, W. C.,

Reynolds, W. C.,

Rodi, W.,

Rodi, W., and Scheuerer, G.,

Rogallo, R. S., and Moin, P.,

Schlichting, H.,
Computational Dynamics Limited,

Shyy, W.,

Simpson, R. L., and Wallace, D. B.,

Spalding, D. B.,

Suen, K. O.,

Sugiura, S., Yamada, T., Inoue, T., Morinishi, K., and Satofuka, N.,
Tabaczynski, R. J.,

Taghavi, R., and Dupont, A.,

Tanaka, K.,

Tindal, M. J., Williams, T. J., and Aldoory, M.,

Uzkan, T., Borgnakke, C., and Morel, T.,

Vanka, S. P.,
Warsi, Z. V. A.,

Wilcox, D. C., and Rubesin, W. M.,

Wolfshtein, M. W.,

Wood, G. B., Hunter, D. U., Taylor, E. S., and Taylor, C. F.,

Yamada, T., Inoue, T., Yoshimatsu, A., Hiramatsu, T., and Konishi, M.,
"In-Cylinder Gas Motion of Multi-Valve Engine - Three Dimensional Numerical Simulation", SAE Paper 860465, 1986.

Yap., C.,
### APPENDIX A

#### RESIDUAL NORMALISATION FACTORS

<table>
<thead>
<tr>
<th>Equation</th>
<th>Variable</th>
<th>Normalisation Factor $M_\phi$</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Continuity</strong></td>
<td>$p$</td>
<td>$M_p \equiv \sum \dot{m}_i$</td>
</tr>
<tr>
<td><strong>Momentum</strong></td>
<td>$u,v,w$</td>
<td>$M_{u,v,w} \equiv \sum \dot{m}_i \left( \bar{u}_i^2 + \bar{v}_i^2 + \bar{w}_i^2 \right)^{1/2}$</td>
</tr>
<tr>
<td><strong>Turbulence Energy</strong></td>
<td>$k$</td>
<td>$M_k \equiv \sum \dot{m}_i \left( \bar{u}_i^2 + \bar{v}_i^2 + \bar{w}_i^2 \right)$</td>
</tr>
<tr>
<td><strong>Turbulence Dissipation</strong></td>
<td>$\varepsilon$</td>
<td>$M_\varepsilon \equiv \frac{M_k}{(L/V_{\text{nom}})}$</td>
</tr>
<tr>
<td><strong>Energy</strong></td>
<td>$h$</td>
<td>$(a)$ Zero wall heat transfer</td>
</tr>
<tr>
<td></td>
<td></td>
<td>$M_h \equiv \sum \dot{\bar{m}}_m \left( \bar{c}_p \bar{T}_m + \sum C_j H_j \right)<em>i \equiv E</em>{\text{in}}$</td>
</tr>
<tr>
<td></td>
<td></td>
<td>$(b)$ Finite wall flux</td>
</tr>
<tr>
<td></td>
<td></td>
<td>$M_h \equiv \max { (E_{\text{in}} - E_{\text{ref}}, \ (E_{\text{out}} - E_{\text{ref}}) }$</td>
</tr>
<tr>
<td></td>
<td></td>
<td>$(c)$ Specified $\Delta T$</td>
</tr>
<tr>
<td></td>
<td></td>
<td>$M_h \equiv \sum \dot{\bar{m}}_m \left( \bar{c}_p \right)_i \Delta T$</td>
</tr>
<tr>
<td></td>
<td></td>
<td>which override (a) and (b) if $\Delta T \neq 0$</td>
</tr>
<tr>
<td><strong>Chemical Species</strong></td>
<td>$c_j$</td>
<td>$M_{c_j} \equiv \max \left{ \sum \dot{\bar{m}}_i \max { \bar{C}_1, \bar{C}_2, \bar{C}_3 }_i, \ (\sum \dot{\bar{m}}_i \max { \bar{C}_1, \bar{C}_2, \bar{C}_3 }_i, \ (\sum \dot{\bar{m}}_i \max { \bar{C}_1, \bar{C}_2, \bar{C}_3 }_i \right}$</td>
</tr>
</tbody>
</table>
In the above table, the subscript \( i \) denotes boundary \( i \).

\[ \dot{m}_{in} \equiv \text{total mass flow through boundary } i \]

\[ \sum_{in}, \sum_{out} \equiv \text{summations over all inflow and outflow boundaries respectively} \]

\[ \bar{\phi}_i \equiv \text{average value of } \phi \text{ over boundary } i \]

\[ E_{out} \equiv \sum_{out} \dot{m}_i (\bar{C}_p \bar{T} + \sum_j C_j H_j)_i \]

\[ E_{ref} \equiv \sum_{in} \dot{m}_i (\bar{C}_p \bar{T}_{ref} + \sum_j C_j H_j)_i \]

\[ L \equiv \text{specified characteristic length of flow domain} \]

\[ V_{nom} \equiv \text{specified mean inlet velocity} \]
APPENDIX B
CALCULATION OF MASS FLOW RATE THROUGH THE TEST SECTION

The mass flow rate through the test section in the rig is measured by an orifice meter. Figure 8.1 shows the orifice plate. The dimensions of the orifice plate used in the rig are given below:

<table>
<thead>
<tr>
<th>Dimension</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Diameter of orifice</td>
<td>d = 41.88 mm</td>
</tr>
<tr>
<td>Diameter of pipe</td>
<td>D = 63.10 mm</td>
</tr>
<tr>
<td>Angle of bevel</td>
<td>F = 40°</td>
</tr>
<tr>
<td>Thickness of the plate</td>
<td>E = 6.06 mm</td>
</tr>
<tr>
<td>Thickness of the orifice</td>
<td>e = 1.293 mm</td>
</tr>
<tr>
<td>The upstream spacing of pressure tapping</td>
<td>l₁ = 64 mm</td>
</tr>
<tr>
<td>The downstream spacing of pressure tapping</td>
<td>l₂ = 30.5 mm</td>
</tr>
<tr>
<td>Diameter ratio</td>
<td>B = d/D</td>
</tr>
</tbody>
</table>

1.1 DISCHARGE COEFFICIENT OF ORIFICE PLATE

The discharge coefficient of the orifice plate is obtained by the Stolz equation:

\[
C = 0.5959 + 0.0312\beta^{2.1} - 0.1840\beta^4 + 0.029\beta^{2.5}\left[\frac{10^6}{Re_D}\right] + 0.0900L_1\beta^4(1 - \beta^4)^{-1} - 0.0337L_2\beta^3
\]

(8.1)

where, L₁ = l₁ / D, L₂ = l₂ / D
when \( L_1 \geq 0.4333 \), use 0.0390 for coefficient of \( \beta^4(1 - \beta^4)^{-1} \).

The Reynolds number refers to both the upstream condition of the fluid and the upstream diameter of the pipe, i.e. \( \text{Re}_D = \frac{U_1D}{v_1} \). Then, the discharge coefficient of the present orifice meter is 0.606.

### 1.2 MASS FLOW RATE

The mass flow rate is determined by the following formula in the British Standard (BS1042).

\[
q_m = CEe \frac{\pi}{4} d^2 \sqrt{2\Delta p \rho}
\]  

(8.2)

Here, \( E \) is velocity of approach factor

\[
E = (1 - \beta^4)^{1/2} = \frac{D^2}{\sqrt{D^4 - d^4}}
\]  

(8.3)

\( \varepsilon \) is expandability factor

\[
\varepsilon = 1 - (0.41 + 0.35\beta^4) \frac{\Delta p}{k_p}
\]  

(8.4)

Formula (8.4) is applicable only if \( \frac{p_2}{p_1} \geq 0.75 \).

Because the test section in the rig is twice the size of the practical one, the mass flow rate through the orifice meter should be twice the value of the practical mass flow rate. The pressure drop across the orifice
meter which is determined by formula (8.2), is used to ensure the expected value of the mass flow rate.

Figure 8.1 Orifice plate