Scavenging Flow in a Two-Stroke Diesel Engine

Master Thesis

By

Eythor Sigurdsson s052837
May 2nd 2011

Section for Fluid Mechanics
Department of Mechanical Engineering
Technical University of Denmark

Supervisor
dr. Jens Honore Walther

Technical University of Denmark
Abstract

Optimization of large two-stroke marine diesel engines is an important task due to high fuel consumption and emissions of these engines. The engine cycle is a complex system where several processes contribute and one of these processes is the scavenging process, the air exchange process of the engine. The present study contains a numerical investigation on the scavenging process performed by a full engine cycle CFD simulation, where the flow field in the scavenging system, engine cylinder and exhaust valve is studied. The characteristic parameters of the scavenging process are estimated and the mixing of cylinder gasses studied.

A CFD model was constructed where a simplified real engine geometry was discretized, representing a full scale engine model. The motion of the piston was simulated by compressing the cells in the cylinder volume and the opening and closing of the scavenging ports was controlled by adopting Arbitrary Sliding Interfaces (ASI). For controlling the air exchange process the exhaust valve was modelled and the motion of the valve controlled by implementing the actual movement of the real engine valve.

The numerical investigation consists of two parts. In the first part a numerical validation was performed where solver parameters and settings were studied and their influence on the model predictions examined. Several meshes with different number of cells were studied and the convergence of the solution examined. Turbulence models were compared using Reynolds Averaged Navier Stokes (RANS) simulations, closed by the standard $k-\varepsilon$ model, the $k-\varepsilon$ RNG model and the $k-\omega$ SST model. The $k-\varepsilon$ RNG model and the $k-\omega$ SST were predicting very similar results.

In the second part the results from the model simulations are compared to available experimental results. The total mass of delivered air through the engine estimated by the CFD model is in good agreement with a experimentally estimated total mass. A reasonable agreement is between the estimated cylinder pressure from the CFD model and the cylinder pressure obtained from an experimental investigation. The scavenging characteristics obtained indicate a efficient scavenging process with a low degree of mixing of cylinder gasses.
Preface

The aim of this work was to construct a CFD model to perform simulations on a full engine cycle for a two-stroke engine. The main methods and results obtained from this work are presented in the report. A wide range of subjects has been treated in this work, which in some cases has limited the details of the results. Treating a wide range of subjects has also opened for a wide range of future subjects that could be performed using the CFD model constructed in this work.

I want to thank especially my supervisor, dr. Jens Honore Walther for great inspiration, support and guidance throughout the project. My wife Nanna I also want to thank for her support and endless patience.
# Contents

## List of Tables

### iv

## List of Figures

### viii

## Nomenclature

### xiii

## 1 Introduction

1.1 Background ........................................... 1
1.2 Thesis Statement ...................................... 3
    1.2.1 Project Specification ............................. 3
1.3 Basics of a Uniflow Scavenged Two-Stroke Engine ...... 4
1.4 Methods of Scavenging ................................ 5
    1.4.1 Two-Stroke Cycle ................................. 6
1.5 The 4T50MX Test Engine ................................. 7
    1.5.1 Measurement Results ............................. 10
1.6 Model Numbers ....................................... 14

## 2 Theory

2.1 Scavenging Process .................................... 15
    2.1.1 Scavenging Parameters ............................ 15
    2.1.2 Scavenging Models ................................. 17
2.2 Governing Equations .................................. 21
2.3 Turbulence Modeling ................................... 24
2.4 Passive Scalar Modeling ............................... 28
2.5 Piston Movement ...................................... 28
    2.5.1 Piston Velocity ................................. 30
## Contents

3 Numerical Method 33
3.1 Solution Algorithm .................................................. 33
3.2 Temporal and Spatial Discretization ................................. 34
3.3 Time Step Size .......................................................... 34
3.4 Boundary Conditions .................................................... 35
3.5 Initial Conditions ....................................................... 37
3.6 Fluid Properties .......................................................... 37
3.7 Arbitrary Sliding Interfaces ............................................ 37
3.7.1 Events ................................................................. 38
3.8 User Subroutines ........................................................ 38
3.9 Parallelization ............................................................ 40

4 Mesh generation 43
4.1 Basic Mesh - 57k Mesh .................................................. 43
4.2 Refinement Step One - 116k Mesh .................................... 49
4.3 Refinement Step Two - 174k Mesh .................................... 51

5 Numerical Validation 55
5.1 Engine Cycle Dependency .............................................. 55
5.2 Comparing Discretization Schemes and Time Steps .................. 61
5.3 Heat Release Volume ..................................................... 65
5.4 Grid Resolution Study ................................................... 72
5.5 Comparing Turbulence Models ......................................... 75

6 Results 81
6.1 Flow Visualization ....................................................... 81
6.2 Mass Flow Rate .......................................................... 97
6.3 Pressure Profile .......................................................... 98
6.4 Tangential Velocity & Swirl Ratio ..................................... 99
6.5 Cylinder Momentum ...................................................... 100
6.6 Mixing Process .......................................................... 102
6.7 Scavenging Parameters ................................................ 105
6.8 Experimental Comparison .............................................. 107
6.9 Effect of Scavenging Port Depth ..................................... 109
6.10 Effect of Wall Temperatures ......................................... 114

7 Discussion and Conclusion 119
7.1 Discussion ............................................................... 119
7.2 Conclusion ............................................................... 122
7.3 Future Work ............................................................ 122
7.4 Acknowledgments ....................................................... 123

Bibliography 125
Appendices

Appendix A  Coefficients and Constants for the $k - \omega$ SST turbulence model

Appendix B  Thermodynamical Properties

Appendix C  Event file

Appendix D  User Subroutines

Appendix E  Setfile for Parallelization

Appendix F  Study of $y^+$ Values
List of Tables

1.1 Characteristic geometrical parameters of the 4T50MX test engine. . . . . . 8
1.2 Characteristic dynamics of the 4T50MX test engine at full load. . . . . . 9
2.1 Coefficients and constants for the $k - \varepsilon$ turbulence models . . . . 27
4.1 Boundary conditions on cylinder volume and cover. ...................... 45
4.2 Boundary conditions on the exhaust valve. ................................. 47
4.3 Number of cells in basic mesh. ............................................. 49
4.4 Number of cells in mesh constructed in refinement step one. .......... 51
4.5 Number of cells in mesh constructed in refinement step 2. ............ 53
5.1 Pressure at TDC and maximum pressure for five consecutive engine cycles. 57
5.2 Total mass flow and relative difference to cycle two in total mass flow through the scavenging port. ......................................................... 59
5.3 Angular momentum at TDC in four consecutive engine cycles. .......... 61
5.4 Total mass of scavenging air delivered for all turbulence models. .... 76
5.5 Scavenging efficiency predicted by the different turbulence models. ... 77
6.1 Comparing cylinder masses and scavenging efficiency for the standard and the reduced scavenging port depth. ................................. 112
A.1 Coefficients and constants for the $k - \omega$ SST turbulence model ..... 133
<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.1</td>
<td>The evolution of the thermal efficiency ($\eta_t$) in the two stroke engines [24].</td>
</tr>
<tr>
<td>1.2</td>
<td>Cross sectional view of a large two-stroke uniflow scavenged diesel engine [17].</td>
</tr>
<tr>
<td>1.3</td>
<td>Different methods of scavenging [25].</td>
</tr>
<tr>
<td>1.4</td>
<td>Sequence of events for a full two-stroke cycle [1].</td>
</tr>
<tr>
<td>1.5</td>
<td>The measured pressure in the scavenging box over one engine cycle, as a function of CAD ATDC.</td>
</tr>
<tr>
<td>1.6</td>
<td>The measured pressure in the engine cylinder over one engine cycle, as a function of CAD ATDC.</td>
</tr>
<tr>
<td>1.7</td>
<td>The valve lift profile as a function of CAD ABDC.</td>
</tr>
<tr>
<td>1.8</td>
<td>The estimated heat released during combustion as a function of CAD ATDC.</td>
</tr>
<tr>
<td>1.9</td>
<td>Area of opening for the scavenging ports and the exhaust valve along with the pressure difference from engine cylinder to exhaust receiver as a function of CAD ABDC.</td>
</tr>
<tr>
<td>2.1</td>
<td>Prediction of scavenging efficiency for the different scavenging models presented.</td>
</tr>
<tr>
<td>2.2</td>
<td>Piston and crank mechanism.</td>
</tr>
<tr>
<td>2.3</td>
<td>Piston position above BDC as a function of CAD.</td>
</tr>
<tr>
<td>2.4</td>
<td>Piston velocity as a function of CAD.</td>
</tr>
<tr>
<td>3.1</td>
<td>Parallel performance compared to idealized performance of a single processor run.</td>
</tr>
<tr>
<td>4.1</td>
<td>The computational domain showing all regions of the mesh.</td>
</tr>
<tr>
<td>4.2</td>
<td>Bottom of cylinder volume mesh (left) and top of cylinder volume mesh (right) with the piston in BDC.</td>
</tr>
<tr>
<td>4.3</td>
<td>Upper Left: Exhaust valve mesh. Upper Right: Exhaust valve mesh and the top of the cylinder volume mesh (red). Lower: Closed exhaust valve.</td>
</tr>
<tr>
<td>4.4</td>
<td>Left: Scavenging port. Right: Alignment of scavenging box, scavenging port and cylinder volume, seen from below.</td>
</tr>
<tr>
<td>Figure</td>
<td>Description</td>
</tr>
<tr>
<td>--------</td>
<td>-------------</td>
</tr>
<tr>
<td>4.5</td>
<td>Refined scavenging port compared to original scavenging port.</td>
</tr>
<tr>
<td>4.6</td>
<td>Results of cylinder volume refinements performed in refinement step one.</td>
</tr>
<tr>
<td>4.7</td>
<td>Refinements made on the scavenging port based on y+ values from the 116k mesh.</td>
</tr>
<tr>
<td>4.8</td>
<td>Refined cylinder volume showing the result of refinements conducted in refinement step 2.</td>
</tr>
<tr>
<td>5.1</td>
<td>Pressure on cylinder cover for five consecutive cycles.</td>
</tr>
<tr>
<td>5.2</td>
<td>Mass flow rate through scavenging ports for 5 consecutive cycles.</td>
</tr>
<tr>
<td>5.3</td>
<td>Tangential velocity profiles in TDC for four consecutive cycles.</td>
</tr>
<tr>
<td>5.4</td>
<td>Mass flow rate profiles for different discretization schemes and C_r numbers as a function of CAD ABDC.</td>
</tr>
<tr>
<td>5.5</td>
<td>Comparing tangential velocity profiles at TDC for different discretization schemes and Courant numbers.</td>
</tr>
<tr>
<td>5.6</td>
<td>Axial velocity profile at a distance of one diameter above piston surface.</td>
</tr>
<tr>
<td>5.7</td>
<td>Illustration of heat release volume in the top of the cylinder volume, with piston positioned in TDC.</td>
</tr>
<tr>
<td>5.8</td>
<td>Temperature [K] distribution inside engine cylinder before opening of intake port for constant volume (left) and for variable volume (right).</td>
</tr>
<tr>
<td>5.9</td>
<td>Comparison of density $\left[ \frac{kg}{m^3} \right]$ variations in the lower half of the engine cylinder at $-30$ CAD ABDC for constant and variable heat release volumes.</td>
</tr>
<tr>
<td>5.10</td>
<td>Comparing the pressure inside the cylinder for the different heat release volumes.</td>
</tr>
<tr>
<td>5.11</td>
<td>Comparing total mass and trapped fresh air mass in cylinder under the scavenging process for constant (CV) and moving (VV) heat release volume.</td>
</tr>
<tr>
<td>5.12</td>
<td>Comparison of axial velocity component [m/s] at $-30$ CAD ABDC for constant and variable heat release volumes.</td>
</tr>
<tr>
<td>5.13</td>
<td>Tangential velocity profiles for the different heat release volumes at $-10$ CAD ATDC, 20 mm above the piston surface.</td>
</tr>
<tr>
<td>5.14</td>
<td>Comparing mass flow rate through scavenging ports with total mass of scavenging air and pressure acting on cylinder cover for different grid sizes during the scavenging process.</td>
</tr>
<tr>
<td>5.15</td>
<td>Comparing cylinder pressure for the 57k,116k and the 174k meshes.</td>
</tr>
<tr>
<td>5.16</td>
<td>Comparing tangential velocity profiles for different grid resolutions, 20 mm above the piston surface in TDC.</td>
</tr>
<tr>
<td>5.17</td>
<td>Comparing angular momentum for the 57k,116k and the 174k meshes.</td>
</tr>
<tr>
<td>5.18</td>
<td>Comparing mass flow rate and pressure profiles for the different turbulence models.</td>
</tr>
<tr>
<td>5.19</td>
<td>Total mass of cylinder charge and retained mass of scavenging air for the different turbulence models.</td>
</tr>
<tr>
<td>5.20</td>
<td>Comparing tangential velocity profiles at TDC for the different turbulence models.</td>
</tr>
<tr>
<td>5.21</td>
<td>Angular momentum over the engine cycle for the different turbulence models.</td>
</tr>
<tr>
<td>5.22</td>
<td>Turbulent kinetic energy for the different turbulence models.</td>
</tr>
<tr>
<td>6.1</td>
<td>Velocity magnitude in m/s, showing the gas flow in the whole engine, including scavenging port, engine cylinder and exhaust valve.</td>
</tr>
</tbody>
</table>
6.2 Radial velocity component in m/s, showing the gas flow in the whole engine, including scavenging port, engine cylinder and exhaust valve. ........................................... 83
6.3 Tangential velocity component in m/s, showing the gas flow in the whole engine, including scavenging port, engine cylinder and exhaust valve. .............. 84
6.4 Axial velocity component in m/s, showing the gas flow in the whole engine, including scavenging port, engine cylinder and exhaust valve. ................. 85
6.5 Velocity components $v_r$, $v_\theta$ and $v_z$ in m/s showing the flow field within the engine cylinder at BDC. ................................................................. 87
6.6 Flow in lower half of the cylinder showing the entrance effect from the scavenging port for several piston positions. ......................................................... 89
6.7 Flow through the exhaust valve for several piston positions. ...................... 91
6.8 Mach number and fluid density in the exhaust valve region, 15 CAD after opening of the exhaust valve. ................................................................. 92
6.9 Horizontal x-y plane at a height of half the port height. ............................. 93
6.10 Flow through a horizontal (x-y) plane of the scavenging port at half the port height, with given velocity magnitude of the out of plane velocity component given in m/s. ............................................. 94
6.11 Temperature ($K$) of the fluid for several piston positions. ...................... 96
6.12 Mass flow rate profile and pressure profiles from scavenging box and engine cylinder. ................................................................. 97
6.13 Cylinder cover pressure through one engine cycle along with the cylinder cover pressure shown from EVC. ................................................................. 98
6.14 Tangential velocity profiles for three different positions above the piston surface 10 CAD before TDC. ................................................................. 99
6.15 Angular and axial momentum in the engine cylinder over one engine cycle, from TDC to TDC. ................................................................. 101
6.16 Total mass of cylinder charge along with the retained fresh mass and the total delivered fresh mass. ................................................................. 103
6.17 Mass concentration of the passive scalar given with piston position ABDC. .... 104
6.18 Scavenging efficiency obtained from CFD model compared to theoretical scavenging models. ................................................................. 105
6.19 Charging efficiency obtained from CFD model compared to theoretical scavenging models. ................................................................. 106
6.20 Schematic view of the experimental test rig. ............................................ 107
6.21 Comparison of experimental results and CFD results for three positions upstream the piston surface with scavenging port 50% closed. ..................... 108
6.22 Flow through a horizontal (x-y) plane of the scavenging port with reduced depth at half the port height, with given velocity magnitude of the out of plane velocity component given in m/s. ............................................. 111
6.23 Angular momentum of the gas in the cylinder volume for standard and reduced scavenging port depths. ......................................................... 113
6.24 Tangential velocity profiles at TDC for the standard and the reduced scavenging port depths. ................................................................. 114
6.25 Temperature [K] distribution in the cylinder volume for adiabatic model and model with constant wall temperatures, with piston at TDC. .......... 115
6.26 Radial temperature (T [K]) profile for adiabatic model and model with constant
wall temperatures, with piston at TDC. .............................................. 116
6.27 Temperature [K] distribution in the cylinder volume for adiabatic model and model
with constant wall temperatures, with piston at TDC. .......................... 116
6.28 Tangential velocity $V_\theta$ [m/s] profile for adiabatic model and model with constant
wall temperatures, with piston at TDC. .............................................. 117

B.1 Molecular viscosity as a function of temperature. ................................. 134
B.2 Specific heat as a function of temperature. .......................................... 135
B.3 Thermal conductivity as a function of temperature. .............................. 135

E.1 Scavenging box and scavenging port for the 174k mesh. ...................... 167
E.2 Cylinder volume and scavenging port for the 174k mesh. ...................... 168
E.3 Cylinder volume and scavenging port for the 174k mesh. ...................... 169
E.4 Exhaust valve and exhaust diffuser for the 174k mesh. ......................... 170

F.1 Dimensionless wall coordinate $y^+$ shown for comparison of the 116k grid and the
174k grid. ......................................................................................... 172
Nomenclature

\( \dot{V} \) Volume flow
\( \vec{r} \) Position vector
\( \vec{v} \) Velocity vector
\( \vec{v}_b \) Cell face velocity

\( \Gamma \) Diffusivity of a scalar
\( \gamma \) Specific heat ratio
\( \Delta \bar{P} \) Engine pressure difference
\( \Delta t \) Time step size
\( \Delta x \) Grid cell size
\( \delta_{ij} \) Kronecker delta
\( \epsilon \) Turbulent dissipation
\( \eta_{ch} \) Charging efficiency
\( \eta_m \) Mass fraction
\( \eta_{sc} \) Scavenging efficiency
\( \eta_{rt} \) Retaining efficiency
\( \eta_{tr} \) Trapping efficiency
\( \theta \) Crankshaft angle
\( \theta_{sc} \) Scavenging port angle
\( \Lambda \) Delivery ratio
\( \mu \) Viscosity
\( \mu_t \) Turbulent viscosity
\( \nu \) Reference kinematic viscosity
\( \nu_t \) Turbulent kinematic viscosity
\( \rho \) Fluid density
\( \rho_b \) Burned gas density
\( \rho_0 \) Reference density
\( \sigma \) Prandtl number
\( \tau_{ij} \) Fluid stress tensor
\( \phi \) Scalar quantity
List of Figures

\( \omega \) Specific turbulent dissipation
\( \omega_s \) Swirl velocity

\( A \) Atwood number
\( A_{cyl} \) Cross-sectional area of engine cylinder
\( a \) Crankshaft radius
\( a_R \) Substance specific constant
\( B \) Cylinder bore
\( b_i \) Body forces
\( b_R \) Substance specific constant
\( CR \) Compression ratio
\( C_r \) Courant number
\( CV \) Constant volume
\( c \) Speed of sound
\( c_p \) Specific heat
\( D_t \) Turbulent mass diffusivity
\( d_{EV} \) Exhaust valve diameter
\( E_p \) Parallel efficiency
\( Eu \) Euler number
\( Fr \) Froude number
\( F_{h, j} \) Diffusional energy flux
\( G \) Linear momentum
\( g \) Gravity
\( h \) Enthalpy
\( h_p \) Scavenging port height
\( k \) Turbulent kinetic energy
\( k_{th} \) Thermal conductivity
\( K \) Short-circuiting factor
\( L \) Connecting rod length
\( L_z \) Angular momentum
\( l \) Turbulent length scale
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$l_p$</td>
<td>Scavenging port length</td>
</tr>
<tr>
<td>$m$</td>
<td>Mass</td>
</tr>
<tr>
<td>$m_{sc}$</td>
<td>Experimental scavenging mass</td>
</tr>
<tr>
<td>$M_{Cyh}$</td>
<td>Total mass of cylinder charge</td>
</tr>
<tr>
<td>$M_{Dref}$</td>
<td>Reference mass</td>
</tr>
<tr>
<td>$M_{Retscav}$</td>
<td>Retained scavenging mass</td>
</tr>
<tr>
<td>$M_{Totscav}$</td>
<td>Total delivered scavenging mass</td>
</tr>
<tr>
<td>$n$</td>
<td>Engine speed</td>
</tr>
<tr>
<td>$n_p$</td>
<td>Number of processors</td>
</tr>
<tr>
<td>$n_{port}$</td>
<td>Number of scavenging ports</td>
</tr>
<tr>
<td>$P$</td>
<td>Pressure</td>
</tr>
<tr>
<td>$P_c$</td>
<td>Critical pressure</td>
</tr>
<tr>
<td>$\overline{P}_{exh}$</td>
<td>Cycle average exhaust receiver pressure</td>
</tr>
<tr>
<td>$P_{max}$</td>
<td>Cycle maximum pressure</td>
</tr>
<tr>
<td>$\overline{P}_{sc}$</td>
<td>Cycle average scavenging box pressure</td>
</tr>
<tr>
<td>$P_{TDC}$</td>
<td>Pressure at TDC</td>
</tr>
<tr>
<td>$p$</td>
<td>Normal and shear stress turbulent generation</td>
</tr>
<tr>
<td>$p_b$</td>
<td>Buoyancy forces</td>
</tr>
<tr>
<td>$q$</td>
<td>Fraction of fresh air charge zone</td>
</tr>
<tr>
<td>$q_h$</td>
<td>Energy source</td>
</tr>
<tr>
<td>$q_\phi$</td>
<td>Scalar source or sink</td>
</tr>
<tr>
<td>$R$</td>
<td>Universal gas constant</td>
</tr>
<tr>
<td>$Re$</td>
<td>Reynolds number</td>
</tr>
<tr>
<td>$R_i$</td>
<td>Individual gas constant</td>
</tr>
<tr>
<td>$R_s$</td>
<td>Swirl ratio</td>
</tr>
<tr>
<td>$r$</td>
<td>Radius</td>
</tr>
<tr>
<td>$S$</td>
<td>Mean velocity strain rate</td>
</tr>
<tr>
<td>$S_{ij}$</td>
<td>Mean strain</td>
</tr>
<tr>
<td>$S_t$</td>
<td>Engine stroke</td>
</tr>
<tr>
<td>$Sc_t$</td>
<td>Turbulent Schmidt number</td>
</tr>
<tr>
<td>$St$</td>
<td>Strouhal number</td>
</tr>
</tbody>
</table>
List of Figures

\begin{flushleft}
\begin{tabular}{ll}
\text{\textit{s}} & Piston position \\
\text{\textit{T}} & Temperature \\
\text{\textit{TI}} & Turbulence intensity \\
\text{\textit{T}_c} & Critical temperature \\
\text{\textit{T}_{sc}} & Temperature of scavenging air \\
\text{\textit{T}_0} & Reference temperature \\
\text{\textit{T}_1} & Temperature at the end of the scavenging period \\
\text{\textit{t}} & Time \\
\text{\textit{t}_c} & Time per engine cycle \\
\text{\textit{t}_p} & Wall clock time for parallel run \\
\text{\textit{t}_s} & Wall clock time for single processor run \\
\text{\textit{t}_{sc}} & Time per scavenging port opening \\
\text{\textit{U}_b} & Bulk velocity \\
\text{\textit{U}_{sc}} & Scavenging port velocity \\
\text{\textit{u}} & Flow velocity \\
\text{\textit{\bar{u}_i}} & Mean flow velocity \\
\text{\textit{u}_i'} & Velocity fluctuations \\
\text{\textit{V}} & Volume \\
\text{\textit{V}_c} & Compression volume \\
\text{\textit{V}_m} & Molar volume \\
\text{\textit{V}_r} & Radial velocity component \\
\text{\textit{V}_\theta} & Tangential velocity component \\
\text{\textit{V}_z} & Axial velocity component \\
\text{\textit{VV}} & Variable volume \\
\text{\textit{w}} & Scavenging port width \\
\text{\textit{x}} & Displacement factor \\
\text{\textit{y}} & Short-circuiting factor \\
\text{\textit{y}^+} & Dimensionless wall distance \\
\text{ABDC} & After Bottom Dead Center \\
\text{ATDC} & After Top Dead Center \\
\end{tabular}
\end{flushleft}
<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>BDC</td>
<td>Bottom Dead Center</td>
</tr>
<tr>
<td>CAD</td>
<td>Crank Angle Degree</td>
</tr>
<tr>
<td>EVC</td>
<td>Exhaust Valve Closing</td>
</tr>
<tr>
<td>EVO</td>
<td>Exhaust Valve Opening</td>
</tr>
<tr>
<td>IPC</td>
<td>Intake/scavenging Port Closing</td>
</tr>
<tr>
<td>IPO</td>
<td>Intake/scavenging Port Opening</td>
</tr>
<tr>
<td>TDC</td>
<td>Top Dead Center</td>
</tr>
</tbody>
</table>
CHAPTER 1

Introduction

In the current thesis a three-dimensional model of a two-stroke marine diesel engine is constructed and the methods and results are presented. In current chapter the background of the two-stroke engine is reviewed along with the working process and the different methods for scavenging the engine. In chapter 2 the theory of the scavenging process is studied and the governing equations are introduced. Chapter 3 contains a discussion of the numerical method and solver parameters for the model and the generation of the numerical mesh is discussed in chapter 4.

A numerical validation of the model is performed in chapter 5 where the solution stability and accuracy are studied. In chapter 6 the results predicted by the model are compared to experimental findings. The effect of altering the scavenging port geometry is studied and a preliminary study of the effect of wall temperatures is performed. A general discussion and conclusion of the model results are performed in chapter 7.

1.1 Background

The concept of two-stroke cycle engines has been used in many different configurations since it was invented by Sir Dugald Clerk in England at the end of the 19th century. The form of the engine as it is known today, with crankcase compression for the induction process and intake ports controlled by the piston, was patented by Joseph Day in England in 1891 [5]. The Day engine is the forerunner of the simple two-stroke engine as it is known today. Today the two-stroke engines can be found in different sizes in numerous applications, varying from a chainsaw to the largest container ships.

The large low speed two-stroke marine diesel engine is the dominant type for marine propulsion of the cargo vessels built in the last few decades [27]. Due to increased transportation, increasing oil prices and regulations on emissions the need of increasing
the efficiency has become significant. One of the major progress in increasing the thermal efficiency was the introduction of the turbo charger in the 1950’s [28]. Using a turbocharger made it possible to extract some of the energy of the exhaust gas and add it to the fresh air charge used for scavenging and combustion in next engine cycle. Increased pressure in the scavenging ports also lead to more efficient scavenging process. The introduction of the turbocharger led to a reduction in the size of the engines and therefore fuel consumption. The evolution of the thermal efficiency is shown in figure 1.1.

As seen in figure 1.1 the thermal efficiency, $\eta_t$, is not significantly improving until the late 1950’s, when the turbocharger was introduced. In modern engines, hydraulics are used to control the exhaust valve opening (EVO) and closing (EVC) making it possible to control the pressure inside the engine cylinder at scavenging port opening. This adjustment possibility makes it possible to optimise the air exchange process of the engine, called the scavenging process.

An important process in terms of engine efficiency is the scavenging process. Improving the scavenging process and thereby the overall efficiency of the engine leads to a lower fuel consumption and emission, which is one of the major concerns in modern engine design. Improvements of the scavenging process can be achieved by experimental or numerical investigations using a full scale engine or engine models for optimizing the engine geometry and operating parameters. The study of the in-cylinder flow driven by the scavenging process is also necessary for understanding the flow prior to fuel injection in the engine cycle.

Performing experimental investigations on a operating engine is expensive and therefore it
is desirable to develop CFD models capable of predicting the operating characteristics of
the engine.

1.2 Thesis Statement

The main purpose of this work is to construct a CFD model capable of simulating the
scavenging process in the two stroke engine. The purpose is to use the CFD model to
gain knowledge on the physics of the whole engine cycle and the scavenging process in the
present engine and also to open for new possibilities in engine design and optimization in
future engines.

1.2.1 Project Specification

This study will consider a full cycle simulation using a full scale CFD model constructed
using geometrical dimensions from a uniflow scavenged two-stroke marine diesel engine.
The model will consist of the main components of the engine and the geometry will be
simplified sufficiently to construct a working model.

Numerical validation of the CFD model should be performed as the available experimental
data is limited. Several turbulence models will be considered and the results predicted
are compared and evaluated as a part of the numerical validation. The goal is to study the
scavenging process into detail, where the important parameters of the scavenging process
will be evaluated and compared to the findings of other numerical and experimental studies.
Due to high compression of the cylinder gas the thermodynamics will also be considered in
the simulations.
1.3 Basics of a Uniflow Scavenged Two-Stroke Engine

A cross-sectional view of a uniflow scavenged two-stroke low speed marine diesel engine is shown in figure 1.2.

![Cross-sectional view of a large two-stroke uniflow scavenged diesel engine](image)

**Figure 1.2:** Cross sectional view of a large two-stroke uniflow scavenged diesel engine [17].

Air is delivered to the scavenging receiver from the turbocharger, which is driven by the exhaust gas. The scavenging receiver is a large container that works as a buffer for the scavenging air before it is passed to the scavenging box. Each cylinder in the engine has its own individual scavenging box to supply the air to the scavenging ports. From the scavenging box the air is delivered to the cylinder through angled scavenging ports which create a swirling flow inside the cylinder. The combustion products are forced out of the cylinder through the exhaust valve by the fresh air blown in through the scavenging ports. The combustion products are blown from the cylinder through the exhaust valve into the exhaust receiver. The exhaust receiver is very large compared to the exhaust ducts so the high velocity in the exhaust duct is reduced significantly due to the large volumetric change, while the pressure is still high in order to drive the turbocharger.
The air exchange process in the two-stroke engine is very sensitive to the design of the scavenging flow system as the air exchange process is controlled through flow related processes and not by the piston as in the four-stroke engine [25].

### 1.4 Methods of Scavenging

Several types of scavenging methods can be used in order to remove the combustion products from the cylinder, however the basic principle remains the same. The most common methods are shown in figure 1.3.

![Figure 1.3: Different methods of scavenging [25].](image)

Figure 1.3 (a) shows one of the simplest methods of scavenging, the cross scavenging concept. The intake port is on one side of the cylinder and the exhaust port on the other side. On the piston top is a deflector plate which directs the incoming flow towards the cylinder head and away from the exhaust port. Using this deflector plate prevents the intake air to enter the exhaust duct directly, also called for short-circuiting. The geometry of the cross scavenging method is rather simple and is typically used in low cost engines [25].

Figure 1.3 (b) shows the uniflow scavenging method. In the uniflow scavenging method the intake ports are placed in the bottom of the cylinder liner while an exhaust valve is placed in the cylinder head. Using the exhaust valve increases the complexity and the weight of the engine while the efficiency of the engine is higher compared to cross and loop scavenging [5]. One of the advantages with uniflow scavenging is that the timing of the
exhaust valve is not dependent on the piston position and the valve can be altered with
the intake port opening (IPO) and closing (IPC) in order to adjust the in-cylinder pressure
for optimum scavenging performance. The uniflow flow scavenged method is typically used
in large two stroke marine diesel engines.

Figure 1.3 (c) shows the loop scavenging method. This method is similar to the cross
scavenging as the intake port and the exhaust ports are both in the cylinder walls. The
main difference is that in loop scavenging the intake and exhaust ports are located at the
same side of the cylinder and the inertia of the incoming air is used to direct the air away
from the exhaust ports. This geometry improves the displacement of the combustion gases
compared to cross scavenging and reduces the risk of short circuiting. The piston has a
flat top which leads to a more compact combustion chamber and more efficient combustion
process [5].

1.4.1 Two-Stroke Cycle

The cycle of a two-stroke uniflow scavenged engine consists of a sequence of events which
are highly coupled in order to create a successive motion of the crankshaft. Figure 1.4
shows the sequence of events for a full two-stroke cycle.

![Sequence of events for a full two-stroke cycle](image)

In figure 1.4 (a) the piston is at the bottom dead center (BDC) and the scavenging ports
and the exhaust valve are fully open. Pressurized air from the turbocharger enters the
cylinder through the scavenging ports and forces the burned gas through the exhaust valve.
This is referred to as the scavenging process. The scavenging process is discussed in more
details in section 2.1. The piston starts to move upwards, partially covering the scavenging
ports.

In figure 1.4 (b) the piston has covered the scavenging ports and the exhaust valve has
closed the exhaust duct. The cylinder is now filled with a mixture of fresh air and burned gas from the previous engine cycle. The air mixture is compressed by the piston motion causing the pressure and temperature to rise to more than 100 bars and over 700 °C.

In figure 1.4 (c) the piston is at top dead center (TDC). Fuel is injected to the cylinder under high pressure. Due to high pressure difference the fuel evaporates when it enters the cylinder causing the fuel to heat up very quickly and self ignite. The expansion of the fuel-air mixture due to combustion leads to a rapid pressure increase causing the piston to move downwards in what is called the power stroke.

In figure 1.4 (d) the piston is moving downwards in the cylinder and the energy from the fuel is extracted. The exhaust valve is opened and the combustion products begin to leave the cylinder through the exhaust valve due to the pressure difference through the exhaust valve. When the pressure in the cylinder has decreased sufficiently the piston uncovers the scavenging ports and the pressurized fresh air is blown into the cylinder, removing the remaining combustion products.

1.5 The 4T50MX Test Engine

The CFD model constructed in this thesis is based on data from the 4T50MX test engine located at the research facility at the headquarters of MAN Diesel & Turbo A/S in Copenhagen. At the time of construction the engine was designed to operate at higher ratings and firing temperatures than any two-stroke engine [30]. The 4T50MX engine is a uniflow scavenged two-stroke diesel engine with a nominal performance of 7500 kW at 123 rpm, which are full load conditions. The characteristic geometrical parameters of the engine are given in table 1.1.
### Table 1.1: Characteristic geometrical parameters of the 4T50MX test engine.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Symbol</th>
<th>Units</th>
<th>–</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cylinders</td>
<td>−</td>
<td>4</td>
<td>−</td>
</tr>
<tr>
<td>Bore</td>
<td>$B$</td>
<td>m</td>
<td>0.5</td>
</tr>
<tr>
<td>Connecting Rod</td>
<td>$L$</td>
<td>m</td>
<td>2.885</td>
</tr>
<tr>
<td>Stroke</td>
<td>$S_t$</td>
<td>m</td>
<td>2.2</td>
</tr>
<tr>
<td>Compression volume</td>
<td>$V_c$</td>
<td>$m^3$</td>
<td>0.02653</td>
</tr>
<tr>
<td>Compression ratio</td>
<td>$CR$</td>
<td>–</td>
<td>17.28</td>
</tr>
<tr>
<td>Exhaust valve diameter</td>
<td>$d_{EV}$</td>
<td>m</td>
<td>0.27</td>
</tr>
<tr>
<td>Scavenging ports</td>
<td>$n_{port}$</td>
<td>–</td>
<td>30</td>
</tr>
<tr>
<td>Scavenging port height</td>
<td>$h_p$</td>
<td>m</td>
<td>0.21</td>
</tr>
<tr>
<td>Scavenging port width</td>
<td>$w_p$</td>
<td>m</td>
<td>0.04</td>
</tr>
<tr>
<td>Scavenging port depth</td>
<td>$l_p$</td>
<td>m</td>
<td>0.05</td>
</tr>
<tr>
<td>Scavenging port angle</td>
<td>$\theta_{sc}$</td>
<td>deg</td>
<td>20</td>
</tr>
</tbody>
</table>

The compression volume ($V_c$) is defined as the volume of the cylinder when the piston is at TDC. The compression ratio ($CR$) gives the ratio between the compression volume and the displaced volume of the engine, given with the piston in BDC.

The dynamics of the engine are considered using experimental results based on measurements performed on the 4T50MX test engine under full load conditions. A short description of the measurement results is given in section 1.5.1. The characteristic dynamics of the engine obtained from these measurements are given in table 1.2.
1.5 The 4T50MX Test Engine

Table 1.2: Characteristic dynamics of the 4T50MX test engine at full load.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Symbol</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>Engine speed</td>
<td>$n$</td>
<td>rpm</td>
</tr>
<tr>
<td>Time p. cycle</td>
<td>$t_c$</td>
<td>s</td>
</tr>
<tr>
<td>Time p. scavenging</td>
<td>$t_{sc}$</td>
<td>s</td>
</tr>
<tr>
<td>Bulk velocity</td>
<td>$U_b$</td>
<td>m/s</td>
</tr>
<tr>
<td>Swirl velocity</td>
<td>$\omega_s$</td>
<td>1/s</td>
</tr>
<tr>
<td>Burned gas density</td>
<td>$\rho_b$</td>
<td>kg/m$^3$</td>
</tr>
<tr>
<td>Reference Temperature</td>
<td>$T_{sc}$</td>
<td>K</td>
</tr>
<tr>
<td>Reference density</td>
<td>$\rho_0$</td>
<td>kg/m$^3$</td>
</tr>
<tr>
<td>Reference viscosity</td>
<td>$v$</td>
<td>m$^2$/s</td>
</tr>
<tr>
<td>Pressure Scavenging box</td>
<td>$P_{sc}$</td>
<td>bar</td>
</tr>
<tr>
<td>Pressure Exhaust rec.</td>
<td>$P_{exh}$</td>
<td>bar</td>
</tr>
<tr>
<td>Pressure difference</td>
<td>$\Delta P$</td>
<td>bar</td>
</tr>
<tr>
<td>Mass scavenging air</td>
<td>$m_{sc}$</td>
<td>kg</td>
</tr>
</tbody>
</table>

| − |

Time during scavenging ($t_{sc}$) is defined as the time it takes for the piston to open for the scavenging ports and close them again.

Bulk velocity ($U_b$) is defined as the volume flow through the engine, $\dot{V}$, divided by the cross sectional area of the cylinder, $A_{cyl}$. The volume flow is found as follows,

$$\dot{V} = \frac{m_s c}{\rho_0 t_{sc}}$$  \hfill (1.5.1)

giving the bulk velocity as

$$U_b = \frac{\dot{V}}{A_{cyl}}$$  \hfill (1.5.2)

The swirl velocity $\omega_s$ is estimated as follows.

$$\omega_s = U_{sc} \sin(\theta_{sc})$$  \hfill (1.5.3)
where \( U_{sc} \) is the flow velocity through the scavenging port estimated using equation 1.5.2 replacing the cylinder cross sectional area with the cross sectional area of the port.

The reference temperature \( T_{sc} \) is a time averaged value measured in the scavenging box of the engine. The burned gas density \( \rho_b \) is the density of the burned gas at the time of port opening and the reference density \( \rho_0 \) is the density of the gas in the scavenging box. The pressure in scavenging box \( P_{sc} \) and pressure in exhaust receiver \( P_{exh} \) are measured quantities.

1.5.1 Measurement Results

The pressure measurements obtained in the scavenging box and exhaust receiver, giving the parameters \( P_{sc} \) and \( P_{exh} \) will be used as boundary conditions for the CFD model. The estimated heat released during combustion will be added to the CFD model in order to expand the cylinder gas and simulate the pressure increase from combustion. The exhaust valve position estimated with measurements will be used for defining the time dependent position of the exhaust valve in the CFD model.

The pressure measured in the scavenging box is used as an inlet boundary condition for the CFD model. The pressure in the scavenging box includes a time varying pressure through the whole engine cycle, shown in figure 1.5.

![Pressure vs CAD ATDC](image)

**Figure 1.5:** The measured pressure in the scavenging box over one engine cycle, as a function of CAD ATDC.

As seen in figure 1.5 the pressure varies depending on the crank angle degree (CAD) after TDC (ATDC) which can have a significant effect on the volumetric flow through the scavenging port compared to a constant pressure value. The time average value of the
scavenging box pressure is \( P_{sc} = 3.64 \text{ bar} \).

For the outlet pressure boundary in the CFD model the measured pressure in the exhaust receiver is used. Only the time averaged pressure is available for the exhaust receiver, with a value of \( P_{exh} = 3.28 \text{ bar} \).

In order to verify the in-cylinder pressure estimated by the CFD model the in-cylinder pressure from the experiment is used. Figure 1.6 shows the measured in-cylinder pressure as a function of CAD.

![Figure 1.6: The measured pressure in the engine cylinder over one engine cycle, as a function of CAD ATDC.](image)

The pressure is measured at a point in the cylinder cover. From bottom dead center (BDC) to exhaust valve closing (EVC) the pressure is close to constant as both scavenging ports and exhaust valve are open. From EVC to top dead center (TDC) the pressure is increasing due to compression until it reaches TDC. Fuel is injected and combustion occurs, leading to a pressure increase, until it reaches a maximum around 11\(^\circ\) after TDC. Due to expansion of the cylinder volume the pressure decreases until the compression stroke starts in the next cycle.

A profile of the exhaust valve lift is shown in figure 1.7, where the lift is measured as the distance from the cylinder cover to the edge of the exhaust valve.
The valve starts to open at 73.5 CAD before the BDC and is fully open at 32 CAD before the BDC, with a maximum valve lift of 66.2 mm. The valve closing starts at 38 CAD after bottom dead center (ABDC) and the valve is fully closed at 83 CAD ABDC. The heat released during combustion is shown in figure 1.8.

The maximum heat that is released occurs approximately at 10° after top dead center (ATDC) and is slowly decreasing until 58° ATDC where it becomes zero and heat from
combustion is no longer added to the system.

The mass of scavenging air flowing through the scavenging ports in each engine cycle has been estimated. The composition of the flue gas is measured along with the system oil. As the amount of oil injected in each engine cycle is well known the emission components are used for estimating the mass flow. The total mass of scavenging air per cycle is 2.069 kg and this value is used for evaluation of the mass flow predicted by the CFD model.

Based on the measurement results from the engine test and the geometry of the engine given in table 1.1, the opening area of the exhaust valve and the scavenging port along with the pressure difference from the engine cylinder to the exhaust receiver is shown in figure 1.9.

![Figure 1.9](image)

**Figure 1.9:** Area of opening for the scavenging ports and the exhaust valve along with the pressure difference from engine cylinder to exhaust receiver as a function of CAD ABDC.

The opening area of the scavenging port is not equal to the cross-sectional area of the ports as the piston is shifted with 55 mm. The result of this is that the piston is not in flush with the lower edge of the ports at BDC.

During exhaust valve opening where the scavenging ports are closed the high pressure difference from engine cylinder to the exhaust receiver is forcing the residual gas through the exhaust valve into the exhaust receiver. At scavenging port opening the pressure difference is at minimum and the flow through the exhaust valve is forced by the incoming air from the scavenging ports. When scavenging ports are fully closed the exhaust valve is still fully open and the remaining residual gas leaves the engine cylinder. The pressure difference is increasing as the exhaust valve is closing due to compression of the cylinder gasses.
1.6 Model Numbers

Several dimensionless numbers can be used to characterize the engine and the scavenging process. The parameters used for calculating the dimensionless numbers are reference parameters given in table 1.2. To characterize the flow in the engine cylinder the Reynolds number giving the ratio of inertial forces to viscous forces is as,

\[ Re = \frac{U_b B}{\nu} = 2.5 \times 10^6 \]  

Using the cylinder diameter as the characteristic length scale shows that the flow is highly turbulent.

The Strouhal number gives the non-dimensional frequency of the engine by relating the engine speed to the bulk velocity of the flow and is given as.

\[ St = \frac{nB}{U_b} = 3.9 \times 10^{-2} \]  

The Euler number is an important parameter for the scavenging process as it relates the pressure forces to the inertial forces and is given as.

\[ Eu = \frac{\Delta P}{\rho_0 U_b^2} = 13.0 \]  

At this Euler number the pressure difference is dominating the flow conditions.

As the density of the burned gas is low compared to the density of the scavenging air, a large density difference across the scavenging ports at the time of port opening. This is described by the Atwood number defined as.

\[ A = \frac{\rho_0 - \rho_b}{\rho_0 + \rho_b} = 0.48 \]  

The Froude number gives the ratio of inertial to buoyancy forces, defined both in terms of axial and radial forces. The Froude number is given as follows,

\[ Fr_a = \frac{U_b^2}{g B} = 141.0 \quad \text{and} \quad Fr_r = \frac{\omega^2 B}{g} = 2.28 \]  

Compared to the axial Froude number, the radial Froude number is showing a stronger effect of buoyancy force in terms of the radial direction. The inertial force is though still larger compared to the buoyancy force.
In this chapter the characteristic dimensionless numbers for the engine, the theory of the scavenging process along with the governing equations for the numerical model are introduced. The motion of the piston in terms of CAD is derived along with the velocity of the piston.

2.1 Scavenging Process

In this section the scavenging parameters and scavenging models are introduced in order to quantify and model the scavenging process of the engine. The scavenging process is the air exchange process of the engine where the burned gas is replaced with fresh air which is compressed and used for the combustion process in the next cycle. The basic parameters of the scavenging process are introduced and are followed up by the scavenging models developed to predict the scavenging process.

2.1.1 Scavenging Parameters

The scavenging parameters describe the process of gas exchange within the engine cylinder. The delivery ratio is defined as the mass of air that is delivered during the scavenging process, \( M_{\text{Totscav}} \), divided by a reference mass, \( M_{\text{Dref}} \). The reference mass is defined as the mass that is required to fill the displaced volume of the cylinder at scavenging box conditions \((P_{sc}, T_{sc})\). Displaced volume is defined as the cross-sectional area of the cylinder times the stroke. The delivery ratio is as follows,

\[
\Lambda = \frac{M_{\text{Totscav}}}{M_{\text{Dref}}}
\] (2.1.1)
The charging efficiency $\eta_{ch}$ is defined as the mass of scavenged air, $M_{Retscav}$ that is retained within the cylinder volume at any instant divided by the reference mass $M_{Dref}$. The charging efficiency becomes,

$$\eta_{ch} = \frac{M_{Retscav}}{M_{Dref}} \quad (2.1.2)$$

The scavenging efficiency $\eta_{sc}$ is defined as the retained scavenging mass $M_{Retscav}$, divided by the total mass of the cylinder charge, $M_{Cych}$. The total mass of the charge is a sum of the mass of fresh air, the mass of burned gas and the mass of unburned fuel from previous cycle.

$$\eta_{sc} = \frac{M_{Retscav}}{M_{Cych}} \quad (2.1.3)$$

The scavenging efficiency can therefore be used to define the effectiveness of the scavenging process.

The retaining efficiency $\eta_{rt}$ gives the ratio of $M_{Retscav}$ and $M_{Totscav}$ and indicates the fraction of fresh air charge that has been delivered and retained in the cylinder.

$$\eta_{rt} = \frac{M_{Retscav}}{M_{Totscav}} \quad (2.1.4)$$

At the closing of the scavenging ports and exhaust valve the retaining efficiency becomes the trapping efficiency, $\eta_{tr}$, which defines the ratio of trapped air charge $M_{Trscav}$ compared to the total delivered fresh air charge $M_{Totscav}$.

$$\eta_{tr} = \frac{M_{Trscav}}{M_{Totscav}} \quad (2.1.5)$$

As seen from the above equations some of these parameters can be related and found as a function of another parameter. The charging efficiency can be defined in terms of the delivery ratio and the retaining efficiency,

$$\eta_{ch} = \Lambda \eta_{rt} \quad (2.1.6)$$

These parameters will be used in section 6.7 to quantify the effectiveness of the scavenging process modelled by the CFD model.
2.1 Scavenging Process

2.1.2 Scavenging Models

Several mathematical models exist for predicting the scavenging process. In past studies the scavenging process has been idealized as a combination of three sub-processes. These sub-processes are:

- Displacement, where burned gas is displaced with fresh air charge.
- Mixing, where the fresh air charge mixes with the burned gas.
- Short-circuiting, where a stream of fresh air charge enters the exhaust duct without mixing with the burned gas [8].

Even though these models cannot be used to analyse the flow field through the scavenging ports and engine cylinder into detail or study the effects of different geometries, they are used to analyse the operating characteristics of the scavenging process. These mathematical models are generally divided into three categories, single-phase, multi-zone and multi-dimensional models [24], where phase refers to a time period where the scavenging process is assumed to occur.

Simple Single-Phase Models

The simple single-phase, one or two zone models assume that the scavenging process occurs under only one phase, which could be displacement or mixing of gasses and never a combination of these two.

Perfect Displacement Model

The first model discussed is the perfect displacement model, which is used to estimate the upper bound of the scavenging process, where parameters such as scavenging efficiency and charging efficiency are overestimated. This model is a so called single-phase, two-zone model which refers to a process where the burned gas is displaced by the fresh air charge and no mixing of gasses takes place. This process consists of two zones, the fresh charge zone and the burned gas zone. For the model calculations it is assumed that the process occurs under a constant cylinder volume and pressure, no heat or mass is allowed to cross the interface between fresh charge and burned gas and the cylinder walls are assumed adiabatic. The relationship between delivery ratio and charging efficiency during perfect displacement can be found as follows [12],

\[
\eta_{ch} = \begin{cases} 
\Lambda & \text{for } \Lambda < \rho_i/\rho_0 \\
\rho_i/\rho_0 & \text{for } \Lambda \geq \rho_i/\rho_0 
\end{cases} 
\] (2.1.7)

where \(\rho_i\) is the density of the fresh charge at cylinder pressure and \(\rho_0\) is the density of the fresh air charge at reference conditions. Equation (2.1.7) states that as long as the volume of fresh air charge is smaller than the cylinder volume the charging efficiency equals the delivery ratio as all the fresh charge is retained. When the volume of fresh air
charge exceeds the cylinder volume and all the burned gas is removed from the cylinder the charging efficiency takes the value of the density difference and the cylinder is fully displaced.

The scavenging efficiency has also been defined in terms of the perfect displacement definition and is given as follows [12],

$$\eta_{sc} = \begin{cases} 1 + \frac{\rho_b}{\rho_i} \left( \frac{\Lambda \rho_i}{\rho_0} - 1 \right) & \text{for } \Lambda < \frac{\rho_i}{\rho_0} \\ 1.0 & \text{for } \Lambda \geq \frac{\rho_i}{\rho_0} \end{cases}$$ (2.1.8)

**Perfect Mixing Model**

The perfect mixing model is another single-phase model, based on similar assumptions as the perfect displacement model. The perfect mixing model is a single zone model where it is assumed that the fresh air charge mixes instantaneously with the burned gas, forming a homogeneous mixture inside the cylinder volume. The perfect mixing is assumed to occur under constant cylinder pressure and volume and the walls are assumed adiabatic. The two gases involved have to follow the ideal gas law and are assumed to have the same molecular weights and their specific heat is identical and constant. The charging efficiency for the perfect mixing model is given as follows [12],

$$\eta_{ch} = 1 - \exp(-\Lambda)$$ (2.1.9)

and the scavenging efficiency is given by the following [12],

$$\eta_{sc} = \frac{T_1}{T_{sc}} [1 - \exp(-\Lambda)]$$ (2.1.10)

where $T_1$ is the temperature of the cylinder contents at the end of the scavenging period and $T_{sc}$ is the temperature of the incoming fresh air charge. From equation (2.1.9) it is seen that the charging efficiency is only dependent on the delivery ratio, while the scavenging efficiency given in equation (2.1.10) is dependent on the temperature ratio between the burned gas and the fresh air charge [12].

As the perfect displacement model defines the upper bound of the scavenging process, the perfect mixing model generally defines the lower bound of the scavenging process. Even though these models do not predict the actual values of the scavenging parameters, they can be used as a reference and for comparison [12].

**Multi-Phase Multi-Zone Models**

It has been shown by visual observations in motored engines that the scavenging process is not a continuous process that occurs under one-single phase as the simple models introduced
2.1 Scavenging Process

In the previous section assumes [12]. In general the scavenging process is assumed to occur in several distinct phases and in two or more distinct zones. In previous section two phases were used, the displacement and mixing phases. For the extended models an additional phase, the short-circuiting phase, is included in the model calculations. These models often require some empirical model constants that depend on both operating conditions and geometry of the cylinder and scavenging ports. The empirical model constants have been estimated from experimental measurements.

Maekawa Three-Zone Model

One of the first multi-zone models developed was the model introduced by Maekawa in 1957 [24]. In this model it is assumed that the scavenging process only occurs under one single phase and in three distinct zones, the fresh air charge zone, the mixing zone and the short-circuiting zone. It is assumed that the scavenging process occurs under constant cylinder volume, pressure and temperature. The fresh air and burned gas have identical density and specific heat and no mass is allowed to cross the interface between zones. Two constants are used in this model, \( K \) which represents the amount of short-circuiting (\( K = 1 \) equals no short circuiting) and \( q \) which represents the amount of fresh air charge zone (\( q = 1 \) equals no existence of fresh air charge zone). The entering fresh air charge is split into three streams where the first stream is directly lost into the exhaust system by short circuiting and the mass fraction for that stream is defined as \((1 - K)\dot{m}_i\), the mass fraction for the fresh air charged stream that is mixed with the burned gas is \(qK\dot{m}_i\) and the mass fraction that forms the last stream of fresh air charge that forms the fresh air charge zone is \((1 - q)K\dot{m}_i\). The charging efficiency from the Maekawa model is as follows from [12],

\[
\eta_{ch} = \begin{cases} 
K(1 - q)\Lambda + [1 - K(1 - q)\Lambda]\eta_m & \text{for } \Lambda < [K(1 - q)]^{-1} \\
1.0 & \text{for } \Lambda \geq [K(1 - q)]^{-1}
\end{cases} 
\]  \quad (2.1.11)

where \( \eta_m \) is the mass fraction of the charge in the mixing zone and is as follows from [12].

\[
\eta_m = q\left\{1 - \exp\left[\frac{-K\Lambda}{1 - K(1 - q)\Lambda}\right]\right\} 
\]  \quad (2.1.12)

The values of the constants \( K \) and \( q \) are typically set to 0.90 and 0.75, respectively [12]. This model has been shown to give some realistic values for the charging and scavenging efficiency [12] but no direct application is available for uniflow scavenging engines. The scavenging models introduced in the following subsection is partly based on the Maekawa model.

Benson and Brandham Two-Phase Three-Zone Model

A modified version of the Maekawa model is the Benson and Brandham model [4]. This model is a two-phase three-zone model based on similar assumptions as the Maekawa model but instead of having a fresh air charge zone, a pure burned gas zone is assumed.
2.1 Scavenging Process

closest to the exhaust ports in phase one. In phase one the cylinder volume also consists
of a mixing zone adjacent to the scavenging ports and a short-circuiting zone. In phase
two it is assumed that the mixing zone has overtaken the pure burned gas zone and the
model only consists of mixing and short-circuiting. Initially it is assumed that the mixing
zone has a mass of \((1 - x)m_0\) and the burned gas zone has a mass of \(xm_0\). The fresh air
charge entering the cylinder volume is assumed to split into two substreams where the
short-circuiting stream has the mass fraction \(ym_i\) and the mixing zone fraction is \((1 - y)m_i\).
The charging efficiency is defined as follows from [4],

\[
\eta_{ch} = \begin{cases} 
(1 - y)\Lambda & \text{for } \Lambda \leq x(1 - y) \\
1 - (1 - x)\exp[x - (1 - y)\Lambda] & \text{for } \Lambda > x(1 - y)
\end{cases}
\] (2.1.13)

where \(x\) is a displacement factor and \(y\) is a short-circuiting factor which gives the amount
of fresh air that is diverted into the exhaust duct without mixing with the burned gas. For
this model the scavenging efficiency is equal to the charging efficiency.

The study of Benson and Brandham on the influence of charging efficiency on engine
performance aimed at comparing equation 2.1.13 to a full model simulation for a uniflow
scavenging marine propulsion engine with a single exhaust valve. Results of this study have
shown that the model is in a good agreement with the full engine simulation [4], especially
at a high displacement factor. For well designed scavenging systems the displacement factor
is assumed 0.75 or larger [4].

The Benson and Brandham model has been modified by several authors to include special
design of engines and operations, such as opposed piston engines where a variation in
cylinder volume and pressure has been taken into account [24].

The scavenging efficiency estimated by the simple scavenging models introduced in the
previous subsections with the given model constants are shown and compared in figure 2.1.
2.2 Governing Equations

The gas flow through the engine is governed by the Navier-Stokes equations for the conservation of both mass and momentum. While the heat transfer is governed by the enthalpy conservation equation for fluid mixture.

The mass conservation (continuity) equation for compressible fluid flows is given in Cartesian coordinates using the Einstein convention as [10],

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_j) = 0$$

(2.2.1)

where $\rho$ is the fluid density, $t$ is the time, $x_j$ is a Cartesian coordinate and $u_j$ is the fluid velocity in direction $x_j$. 

Figure 2.1: Prediction of scavenging efficiency for the different scavenging models presented.

As mentioned earlier the perfect displacement model gives the upper boundary for the scavenging process as no mixing of gasses occurs under this process. The perfect mixing model gives the lower boundary for the scavenging process as it assumes the gasses are mixed perfectly. In between these models the Maekawa model is predicting similar results as the perfect mixing model, while the Benson and Brandham model with a different shape gives a value of the scavenging efficiency in between the other models. The results from these models will only be used as a comparison to the results predicted by the CFD model and no validation of these models will be undertaken.
The momentum conservation equations for compressible fluid flow is given as [10],

\[ \frac{\partial \rho u_i}{\partial t} + \frac{\partial (\rho u_j u_i)}{\partial x_j} = \frac{\partial \tau_{ij}}{\partial x_j} - \frac{\partial P}{\partial x_j} \delta_{ij} + \rho b_i \] (2.2.2)

in which \( b_i \) represents the body forces acting on the control volume and \( \tau_{ij} \) is the fluid stress tensor. Assuming a Newtonian fluid, the stress tensor can be written as [10],

\[ \tau_{ij} = \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \left( \frac{2}{3} \mu \frac{\partial u_i}{\partial x_i} \right) \delta_{ij} \] (2.2.3)

where \( P \) is the pressure, \( \mu \) is the dynamic viscosity of the fluid and \( \delta_{ij} \) is Kronecker delta (unity when \( i = j \) else zero). When dealing with turbulent flow the dependent variables, such as \( u \) and \( p \) assume their ensemble average values. This is discussed in more details in section 2.3.

When referring to the entire system of flow equations the Navier-Stokes equations are supplied with an equation of state and the energy equation, giving a complete set of flow equations [2]. The energy equation, governing the fluid heat transfer is given as [6],

\[ \frac{\partial \rho h}{\partial t} + \frac{\partial (\rho u_j h + F_{h,j})}{\partial x_j} = \frac{\partial P}{\partial t} + u_j \frac{\partial P}{\partial x_j} + \tau_{ij} \frac{\partial u_i}{\partial x_j} + q_h \] (2.2.4)

where \( F_{h,j} \) is the diffusional energy flux in direction \( x_j \) given by following [6],

\[ F_{h,j} = -k_{th} \frac{\partial T}{\partial x_j} \] (2.2.5)

where \( k_{th} \) is the thermal conductivity, and \( T \) is the temperature. In equation (2.2.4), \( h \) is the static enthalpy given by following,

\[ h = \tau_p T - c_p^0 T_0 \] (2.2.6)

where \( \tau_p \) is the mean constant-pressure specific heat at temperature \( T \), \( c_p^0 \) is the reference specific heat at reference temperature \( T_0 \).

For density calculations the equation of state used is the ideal gas law, given as [18],

\[ PV = m R_i T \] (2.2.7)

where \( V \) the volume, \( m \) the mass, \( R_i \) the individual gas constant. Density is gained from the volume and the mass as \( \rho = \frac{m}{V} \).
The transport of a scalar quantity is governed by the generic transport equation, analogous to the previous equations. The conservation equation is given as \[10\],
\[
\frac{\partial \rho \phi}{\partial t} + \frac{\partial (\rho u_j \phi)}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \Gamma \frac{\partial \phi}{\partial x_j} \right) + q_\phi
\] (2.2.8)

where \( \Gamma \) is the diffusivity for the scalar quantity \( \phi \) and \( q_\phi \) is the source or sink of the scalar quantity.

As the focus of this study is to simulate a reciprocating engine the movement of the computational grid with respect to time has to be taken into account. An additional equation, called the space conservation law is solved simultaneously with the mass, momentum, and energy conservation equations. The space conservation law relates the change in cell volume with the cell face velocities \( v_b \) [6], and is given on integral form as \[10\],
\[
\frac{d}{dt} \int_{\Omega} d\Omega - \int_{S} \vec{v}_b \cdot \vec{n} dS = 0
\] (2.2.9)

When the location of the grid is known as a function of time the solution to the Navier-Stokes equations poses no new problems due to the movement of boundaries as the convective fluxes are calculated using the relative velocity \( \vec{v} - \vec{v}_b \) [10].

**Real Gas vs. Ideal Gas**

The high pressures occurring in an engine cycle cause the ideal gas law assumption to deviate from the properties of a real gas as the ideal gas law does not compensate for the molecular forces. The difference in the state quantities of an ideal gas is compared to the state quantities of a real gas in order to evaluate the difference in the pressure provided by the CFD model compared to the experimental value. The equation of state used to describe the real gas behaviour is the Redlich-Kwong equation, given as \[13\],
\[
P = \frac{RT}{V_m - b_R} - \frac{a_R}{\sqrt{T}V_m(V_m + b_R)}
\] (2.2.10)

where \( V_m \) is the molar volume defined as the molar mass divided by density, \( R \) is the universal gas constant, \( T \) is temperature and \( a_R \) and \( b_R \) are substance specific constants given as \[13\],
\[
a_R = \frac{0.42748R^2T_{c}^{2.5}}{P_c}
\] (2.2.11)
\[
b_R = \frac{0.08662RT_{c}}{P_c}
\] (2.2.12)
where \( T_c \) and \( P_c \) are the critical temperature and pressure, respectively. The ideal gas law is given in equation (2.2.7).

The pressure of the gas found by using the Redlich-Kwong equation is compared to a pressure obtained by the ideal gas law at a temperature of 870 K and a density of 55 kg/m³ which is the state of the air with the piston positioned in TDC. The pressure obtained by the Redlich-Kwong equation is 151.9 bar compared to a pressure of 142.09 bar obtained by the ideal gas law, a difference of 6.5%. From the assumption of ideal gas in the numerical model a pressure of 6.5% lower compared to the experimental value is expected.

### 2.3 Turbulence Modeling

The Navier-Stokes equations presented in section 2.2 can be solved using different approaches to the turbulence quantities. One approach is to solve the discretized equations directly as they form a closed set of equations, this is called Direct Numerical Simulation (DNS). In DNS the whole range of spatial and temporal scales of turbulence within the flow are resolved which is very computationally demanding and requires very fine mesh resolution. Instead of solving the Navier-Stokes equations directly several other approaches have been made, such as the Reynolds Averaged Navier-Stokes equations (RANS) and Large Eddy Simulations (LES). The turbulent approach used in this study are the RANS equations where the governing equations are averaged using the Reynolds decomposition. In Reynolds decomposition it is assumed that the instantaneous velocity can be divided into a mean part \( \bar{u} \) and a fluctuating part \( u' \).

\[
\bar{u}_i = \bar{u} + u'
\]

Applying the Reynolds decomposition to all of the quantities in the Navier-Stokes equations gives the Reynolds Averaged Navier-Stokes equations where an extra term has appeared in the stress term. This additional term is called the Reynolds stresses, \( \rho u' \bar{u}' \). Due to this additional stress term the number of unknowns in the set of equations is now more than the number of equations and this is referred to as the closure problem of turbulence [26]. In order to close the RANS equations the Reynolds stresses have to be modelled using a turbulence model.

For the energy flux \( F_{h,j} \) given in equation (2.2.5) the term \( \overline{\rho u' h'} \) is added on the right hand side to account for the turbulent fluctuations. The solution of the turbulent part can be found as follows [6]:

\[
\overline{\rho u' h'} = -\frac{\mu_t}{\sigma_{h,t}} \frac{\partial \bar{h}}{\partial x_j}
\]

where \( \sigma_{h,t} \) is the turbulent Prandtl number of enthalpy, set to 0.9 for all turbulence models.
Several types of turbulence models have been developed for the solution of the closure problems and in this study three turbulence models are used, where the effects of these models are compared and evaluated. The turbulence models that are used are the two equation standard $k-\varepsilon$ model [15], the $k-\varepsilon$ RNG model [31] where RNG stands for Renormalization Group and the $k-\omega$ SST model [21]. These two equation turbulence models calculate the Reynolds stresses using the Boussinesq hypothesis [29] where the turbulent kinetic energy $k$ and the turbulent eddy viscosity $\mu_t$ are related to the Reynolds stresses. When considering the $k-\varepsilon$ models the turbulent eddy viscosity is calculated using the turbulent kinetic energy and the turbulent dissipation $\varepsilon$ [6], while for the $k-\omega$ model the turbulent viscosity is calculated using the turbulent kinetic energy and the specific turbulent dissipation $\omega$. The calculation of the turbulent kinetic energy and the turbulent dissipation for turbulence models is described in the following subsection. The turbulent viscosity for the two $k-\varepsilon$ turbulence models is given as follows [6],

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

where $C_\mu$ is a model constant for the turbulence models, introduced in next section. The turbulent viscosity for the $k-\omega$ SST model is as follows [6],

$$\mu_t = \rho \frac{a_1 k}{\max(a_1 \omega, \Omega^* F_2)}$$

where $a_1$ is a constant equal 0.31, $\Omega^*$ is a model coefficient and $F_2$ is a function given in appendix A.

**Standard $k-\varepsilon$ turbulence model**

The implementation of the $k-\varepsilon$ turbulence model in STAR-CD has taken into account the compressibility of the working fluid for the calculation of the turbulent kinetic energy $k$ and the turbulent dissipation $\varepsilon$. [6]. The turbulent kinetic energy transport equation is as follows [6],

$$\frac{\partial (\rho k)}{\partial t} + \frac{\partial}{\partial x_j} \left[ \rho u_j k - \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] = \mu_t (p + p_B) - \rho \varepsilon - \frac{2}{3} \left( \mu_t \frac{\partial u_i}{\partial x_j} + \rho k \right) \frac{\partial u_i}{\partial x_j}$$

where $\mu_t$ is the turbulent viscosity, $\sigma_k$ is the turbulent Prandtl number, $p$ is the turbulent generation by normal and shear stresses given as

$$p = \frac{\partial u_i}{\partial x_j} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)$$
and \( p_B \) accounts for the buoyancy forces and is given as:

\[
p_B = \frac{-g_i}{\sigma_k} \frac{\partial \rho}{\partial x_i} \tag{2.3.7}
\]

The second term on the right hand side of equation 2.3.5 accounts for the viscous dissipation and the third and last term accounts for the compressibility effects.

The equation for the turbulent dissipation in the standard \( k-\varepsilon \) model is as follows [6],

\[
\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial}{\partial x_j} \left[ \rho u_j \varepsilon - \left( \mu + \frac{\mu_t}{\sigma_{\varepsilon}} \right) \frac{\partial \varepsilon}{\partial x_j} \right] = C_{\varepsilon_1} \varepsilon \left[ \mu_t \rho - \frac{2}{3} \left( \mu_t \frac{\partial u_i}{\partial x_i} + \rho k \right) \frac{\partial u_i}{\partial x_i} \right] + C_{\varepsilon_2} \mu_t p_B - C_{\varepsilon_3} \varepsilon \frac{\varepsilon^2}{k} + C_{\varepsilon_4} \rho \varepsilon \frac{\partial u_i}{\partial x_i} - \frac{C_{\mu} \eta^3 \left( 1 - \eta / \eta_0 \right) \rho \varepsilon^2}{1 + \beta \eta^3} \tag{2.3.8}
\]

where \( C_{\varepsilon_1}, C_{\varepsilon_2}, C_{\varepsilon_3} \) and \( C_{\varepsilon_4} \) are model coefficients and \( \sigma_{\varepsilon} \) the turbulent Prandtl number whose values are given in table 2.1.

**k-\( \varepsilon \) RNG turbulence model**

The \( k-\varepsilon \) RNG model is a modified version of the standard \( k-\varepsilon \) model. The transport equation for the turbulent kinetic energy is the same as for the standard \( k-\varepsilon \) model, while the equation for the turbulent dissipation is modified using a Renormalization Group method (RNG). The equation for the turbulent dissipation is as follows [6],

\[
\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial}{\partial x_j} \left[ \rho u_j \varepsilon - \left( \mu + \frac{\mu_t}{\sigma_{\varepsilon}} \right) \frac{\partial \varepsilon}{\partial x_j} \right] = C_{\varepsilon_1} \varepsilon \left[ \mu_t \rho - \frac{2}{3} \left( \mu_t \frac{\partial u_i}{\partial x_i} + \rho k \right) \frac{\partial u_i}{\partial x_i} \right] + C_{\varepsilon_2} \mu_t p_B - C_{\varepsilon_3} \varepsilon \frac{\varepsilon^2}{k} + C_{\varepsilon_4} \rho \varepsilon \frac{\partial u_i}{\partial x_i} \nonumber
\]

\[
- \frac{C_{\mu} \eta^3 \left( 1 - \eta / \eta_0 \right) \rho \varepsilon^2}{1 + \beta \eta^3} \tag{2.3.9}
\]

where \( \eta = S^k_T \) and \( \eta_0 \) and \( \beta \) are empirical constants given in table 2.1 and \( S \) the mean velocity strain rate is given as:

\[
S = \sqrt{2S_{ij}S_{ij}} \quad \text{where} \quad S_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \tag{2.3.10}
\]
where $S_{ij}$ is the mean strain. When comparing the dissipation rate of the standard $k-\varepsilon$ model given in equation 2.3.8 and the RNG model given in equation 2.3.9 the latter has an extra term and the coefficients to the different terms are different from each other. The coefficients are shown in table 2.1.

### Table 2.1: Coefficients and constants for the $k-\varepsilon$ turbulence models

<table>
<thead>
<tr>
<th>Model</th>
<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>$\eta_0$</th>
<th>$\beta$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Standard $k-\varepsilon$</td>
<td>0.09</td>
<td>1.0</td>
<td>1.22</td>
<td>1.44</td>
<td>1.92</td>
<td>1.44</td>
<td>-0.33</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>$k-\varepsilon$ RNG</td>
<td>0.085</td>
<td>0.719</td>
<td>0.719</td>
<td>1.42</td>
<td>1.68</td>
<td>1.42</td>
<td>-0.387</td>
<td>4.38</td>
<td>0.012</td>
</tr>
</tbody>
</table>

In the work of Antila et al. [3] it was shown that modifying the constant $C_{\varepsilon 4}$ from $-0.387$ to $-1.0$ improved the production of turbulent viscosity as it was shown that the RNG model was underestimating the production of the turbulent viscosity. The effect of modifying $C_{\varepsilon 4}$ will also be studied in this work.

$k-\omega$ SST turbulence model

The transport equation of turbulent kinetic energy for the $k-\omega$ SST turbulence model is as follows [6],

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

(2.3.11)

where $\beta^*$ is a function of model constants. Model constants and functions are found in appendix A. The equation for the turbulent dissipation in the $k-\omega$ SST model is as follows [6],

$$\frac{\partial (\rho \omega)}{\partial t} + \frac{\partial}{\partial x_j} \left[ \rho u_j \omega - \left( \mu + \frac{\mu_t}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right] = \alpha \frac{\omega}{k} u_j p - \rho \beta \omega^2 + \rho S_\omega + C_{\varepsilon 3} \mu_t p_b C_\mu \omega$$

(2.3.12)

where the constants $C_{\varepsilon 3}$ and $C_\mu$ are the same as for the standard $k-\varepsilon$ model and are given in table 2.1. The coefficients $\alpha$, $\beta$ and $S_\omega$ are complex functions of the model constants, where both model constants and functions are found in appendix A.

The $k-\varepsilon$ turbulence models were chosen as previous studies on turbulence modeling in internal combustion engines, such as the work of Kaario et al. [16] which compared the standard $k-\varepsilon$ and $k-\varepsilon$ RNG models in terms of length scales. Lendormy et al. [19] compared the models in terms of initial turbulence prior to combustion. Goldsborough
et al. [11] used the $k-\varepsilon$ RNG model for simulations in a two-stroke free piston engine. The $k-\omega$ SST model has been chosen as the third turbulence model for evaluation of the difference between the $k-\varepsilon$ models. Other models such as a Gibson and Launder Reynolds stress model was used by Yang et al. [32] for simulation of in-cylinder engine flow. The Gibson and Launder Reynolds stress model was tested but proved to be too computationally demanding for the CFD model.

### Near Wall Treatment

The distribution of velocity, temperature, turbulence and energy within the boundary layer is in this study treated using standard wall functions. Standard wall functions are used where it is assumed that the cell centroid of the near wall cell lies in the logarithmic layer region of the boundary layer. As the logarithmic layer is assumed the dimensionless wall coordinate $y^+$ has to be in the range of $30 - 100$.

### 2.4 Passive Scalar Modeling

In order to calculate the concentration of fresh air within the cylinder during and after the scavenging process a passive scalar is used. A passive scalar is a scalar which is present and convected in the flow field without affecting the fluid motion or the fluid properties. Using this method the fluid entering the cylinder from the scavenging port can be marked and traced in the engine cylinder and exhaust and the mass fraction of scavenging air at any cell in the domain can be calculated from the instantaneous concentration of the passive scalar. The conservation of a passive scalar is governed by equation 2.2.8. The diffusivity $\Gamma$ of the passive scalar is given as follows [10],

$$\Gamma = \rho \frac{v_t}{Sc_t} \quad (2.4.1)$$

where $v_t$ the turbulent viscosity and $Sc_t$ the turbulent Schmidt number defined as the ratio of turbulent momentum diffusivity (viscosity, $v_t$) and turbulent mass diffusivity, $D_t$.

$$Sc_t = \frac{v_t}{D_t} \quad (2.4.2)$$

The value of the Schmidt number used in the simulations is 0.9 which is the standard value in STAR-CD.

### 2.5 Piston Movement

The motion of the piston connected to a crankshaft with a connecting rod can be described using simple trigonometric relations. Figure 2.2 shows the piston mechanism where $s$ is the distance between the crankshaft axis and the crosshead pin axis, $L$ is the length of the
connecting rod, \( a \) the crankshaft radius and \( \theta \) is the crankshaft angle.

![Diagram of piston and crank mechanism](image)

**Figure 2.2:** Piston and crank mechanism.

The vertical distance from crankshaft axis (\( O \)) to the crankshaft pin (\( N \)) is given as;

\[
s_1 = a \cos \theta \tag{2.5.1}
\]

The vertical distance from the crankshaft pin (\( N \)) to the crosshead pin (\( C \)) is given as;

\[
s_2^2 = L^2 - b^2 \tag{2.5.2}
\]

where \( b \) can be written as \( a \sin \theta \). The equation describing the piston vertical position compared to a reference position, \( s = s_1 + s_2 \) is written as;

\[
s = a \cos \theta + \sqrt{L^2 - a^2 \sin^2 \theta} \tag{2.5.3}
\]

Using the geometrical parameters given in section 1.5 the piston position is calculated using equation (2.5.3) and plotted in figure 2.3.
2.5 Piston Movement

As seen in figure 2.3 the piston moves in positive direction upwards from BDC to TDC at 180° where the piston has the maximum distance from BDC, the stroke $S_t = 2.2\, \text{m}$. After TDC the piston moves downwards until it reaches the BDC and ends the cycle.

2.5.1 Piston Velocity

The motion of the piston affects the flow field and for the flow closest to the piston surface it is assumed that the flow takes the same vertical velocity as the piston due to the no slip condition on the piston surface. The instantaneous piston velocity can be found directly by differentiating equation (2.5.3) with respect to the crank angle.

\[
\frac{ds}{d\theta} = -a\sin \theta - \frac{a^2 \sin \theta \cos \theta}{\sqrt{L^2 - a^2 \sin^2 \theta}} \tag{2.5.4}
\]

The implementation of the piston velocity in the CFD model is as follows where the rate of change in position \( \frac{ds}{dt} \) is given in terms of time instead of crank angle.

\[
\frac{ds}{dt} = \frac{ds}{d\theta} k \quad \text{where} \quad k = \frac{2\pi n}{60} \tag{2.5.5}
\]
The piston velocity is calculated and plotted in figure 2.4 using the same geometrical parameters as in figure 2.3.

![Piston velocity as a function of CAD.](image)

**Figure 2.4:** Piston velocity as a function of CAD.

As seen in figure 2.4 the piston has its maximum velocity at about 90° and takes the value of zero at BDC and TDC when the piston is changing direction.
In this chapter the methods and parameters that need to be set for the simulations are discussed. Boundary conditions for the CFD model are discussed and the method of grid movement due to piston motion is introduced along with the sliding interfaces connecting the individual computational domains. The CFD model is constructed using the commercial CFD program STAR-CD. For grid generation, pre- and post processing the program Prostar version 4.12.033 is used. Simulations are conducted on the high performance Linux cluster, Niflheim at the Technical University of Denmark.

3.1 Solution Algorithm

The governing equations are discretized using the finite volume method (FV), described in [10]. For the iterative solution process of the discretized equation the PISO algorithm is used. The PISO algorithm works with predictor and corrector stages in order to find a converged solution of the discretized equations at each time step. The predictor stage produces a provisional velocity and pressure distribution based on an initial value of the variable fields. The provisional velocity field is then refined in the corrector stage and an approximated solution to the momentum and continuity equation is found by iterations until a specified inner tolerance is reached. The inner tolerances used for evaluating convergence on the governing equations is set to $10^{-3}$. A study of the effect of altering the tolerance to $10^{-5}$ showed no difference in the solution for the test case considered. The governing equations are therefore assumed to reach a reasonable convergence using a tolerance of $10^{-3}$. The momentum and pressure equations are decoupled and solved separately within each corrector step using the segregated solver in STAR.

The solver used for solving the system of decoupled linear equations is of Conjugent Gradient (CG) type. The Algebraic MultiGrid (AMG) solver is also available in STAR and is recommended for simulations including a large number of cells and unstructured grids.
A comparison of the two available solvers showed that the conjugate gradient solver was faster compared to the AMG solver and was therefore chosen.

3.2 Temporal and Spatial Discretization

To account for transient effects the governing equations are discretized in time. When using the PISO algorithm the fully implicit Euler scheme is the only option provided where the fully implicit scheme which is of first order is supplied with correctors resulting in an accuracy lying between first and second order [6].

For the spatial discretization of the convective terms in the governing equations several schemes are available. The simplest scheme available is the Upwind Differencing scheme (UD). The UD scheme is a first order accuracy scheme as the nearest upwind neighbour is the only point used for evaluation of the face value of the control volume. Another scheme based on the UD scheme is the Linear Upwind Differencing scheme (LUD). This is a second order scheme as an additional point upwind the first point is added to the stencil. The third scheme used is the Central Differencing scheme (CD). This scheme is of second order as the LUD scheme but instead of choosing two points upwind the cell face, the control points are chosen on each side of the cell face. The fourth and last scheme used in this study is the MARS scheme, where MARS stands for Monotone Advection and Reconstruction Scheme. The second order MARS scheme is a product only available in STAR-CD [6]. The solution accuracy of the MARS scheme is the least sensitive to the mesh structure and skewness according to STAR-CD methodology [6]. Further informations on the discretization schemes are found in [10] and [2].

3.3 Time Step Size

The time step size is of great importance for both solution accuracy and for stability reasons. All simulation are running with an adaptive time step where the time step size is evaluated at each time step in terms of the Courant \((C_r)\) number or the Courant-Friedrichs-Lewy condition (CFL) which is defined as follows,

\[
C_r = \frac{(u + c)\Delta t}{\Delta x}
\]  

(3.3.1)

where \(u\) is the local cell velocity, \(c\) is the speed of sound, \(\Delta t\) is the time step and \(\Delta x\) is the dimension of a grid cell. The interpretation of the \(C_r\) number is the fraction of a cell length a fluid particle will travel in each time step. When the speed of sound is taken into account the \(C_r\) number gives the fraction of cell that a propagating perturbation will travel in each time step. The \(C_r\) number is calculated for each cell in the domain where the criterion for time step size can either be based on the domain maximum \(C_r\) number or the domain average \(C_r\) number. Due to large variations in cell sizes the maximum \(C_r\) number criterion
As the temporal discretization method is implicit, described in section 3.2, the solution is not as highly dependent of the $C_r$ number as if the temporal discretization was explicit. In this work three $C_r$ numbers have mainly been used; 50, 100 and 200. A study of these $C_r$ numbers in terms of solution accuracy and stability reasons is conducted in section 5.2.

### 3.4 Boundary Conditions

The cell faces which are placed along the physical boundary of the computational domain need to be managed in a different way compared to the internal faces. There are several types of boundary conditions that can be used to describe the flow physically. The boundary conditions used in this work are presented in the following sections.

#### Wall Boundaries

Wall boundary conditions are used to describe the physical boundary of the computational domain. In this work wall boundary conditions are assumed to be thermally insulated (or adiabatic). The rate of heat flux from the fluid to the walls is therefore assumed to be zero. As the fluid is assumed viscous the no-slip condition is used at the walls. The fluid will therefore have a zero velocity relative to the boundary. For a stationary wall the fluid velocity will become zero. For a moving wall e.g. the piston, the velocity of the wall must be taken into account as it could affect the flow field in the cell adjacent to the wall. This is described in further details in section 3.8. For all the walls in the domain there is assumed to be zero mass flux through the walls. The pressure at the walls is treated in the same way as the temperature, where the pressure gradient across the wall is zero.

#### Pressure Boundaries

Pressure boundaries are used in this study for describing the inflow and outflow conditions of the engine. The pressure on the outlet boundary is set as constant and assumed the same as the time averaged pressure measured in the exhaust receiver, $P_{sc}$. The temperature and turbulence quantities on the outlet are treated as zero gradient from the cell adjacent to the boundary. As described in section 1.5, the pressure on the inlet boundary varies with CAD and a constant value is therefore not defined as in the case of the outlet boundary. The time varying pressure on the inlet boundary is controlled by using the user subroutine `bcdefp.f` which is described in section 3.8. The temperature on the inlet boundary takes the measured value of the mean temperature in the scavenging receiver measured in the test described in section 1.5. The turbulence intensity $TI$ of 5% and length scale $l$ of 0.005 m, are given for the inlet boundary for the calculation of the turbulent kinetic energy $k$, the turbulent dissipation $\varepsilon$ and the specific turbulent dissipation $\omega$. The quantities $k$, $\varepsilon$ and $\omega$ are found as follows [6],
3.4 Boundary Conditions

\[ k = \frac{3}{2} T I^2 u^2 \]  
(3.4.1)

\[ \epsilon = C_{\mu}^{3/2} k^{3/2} l \]  
(3.4.2)

\[ \omega = \frac{\sqrt{k}}{l \beta^{1/4}} \]  
(3.4.3)

where \( u \) is the velocity and \( C_{\mu} \) and \( \beta^* \) are turbulence model constants.

Cyclic Boundaries

Cyclic boundary conditions are imposed on the CFD model in order to save computational time. By imposing cyclic boundary conditions on a pair of geometrically identical boundaries it is assumed that repeating flow conditions exist on these boundaries. Repeating flow conditions means that variables are forced to take the same value at these paired boundaries. In this study all flow variables on one member of the cyclic set are matched with the corresponding values of the other member.

Attachment Boundaries

Attachment boundaries are used to define the interface between two blocks of meshes that slide past each other. This type of boundary condition is necessary for defining an arbitrary sliding interface introduced in section 3.7. The attachment boundaries are matched in a local coordinate system which has to be specified by the user. As long as the mesh blocks are aligned they are connected and the fluid is allowed to pass through their boundaries. As the blocks slide past each other and the attachment boundaries are no longer aligned the attachment boundary changes to its alternating boundary which in this study is a wall boundary.

Monitoring Boundaries

Monitoring boundaries are arbitrary surfaces that are defined in the same way as other boundaries, but can be placed at any internal cell face within a domain without affecting the flow field. This type of boundary condition is used in this study for monitoring the mass flow rate through the scavenging ports.
3.5 Initial Conditions

The simulations are started with the fluid at rest, prescribing the pressure in the domain as 3.28 bar, the same pressure as on the outlet boundary and the temperature in the domain is given as 312 K which is the reference temperature $T_{sc}$. The fluid will remain at rest until the pressure from the inlet boundary influences the fluid. As the flow is driven by pressure boundaries a reference pressure is set for the simulations where the absolute value of the pressure in each cell is calculated as the pressure calculated from the governing equations added with the reference pressure. The prescribed values on the pressure boundaries are absolute pressure values and therefore the reference pressure is set to zero.

3.6 Fluid Properties

The properties of the working fluid such as molecular viscosity $\mu$, specific heat $C_p$ and thermal conductivity $k_{th}$ are assumed to follow a polynomial form where the property values are calculated as a function of temperature. As air consists of 78% Nitrogen the thermodynamical properties of Nitrogen are used for the simulations as a polynomial fit for the thermodynamical properties of air is not available. Figures showing variations in these properties as function of temperature can be found in appendix B. The polynomial used for estimating the thermodynamical properties are from STAR-CD methodology [6].

3.7 Arbitrary Sliding Interfaces

The arbitrary sliding interface (ASI) technique is used to connect subdomains that slide past each other in the moving mesh simulation. This technique enables the solver to transfer data from one subdomain to another by merging the boundary faces on each subdomain at every time step. In order to form such a sliding interface, attach boundaries are applied on the surface of each subdomain. The boundary faces that form the sliding interface are distinguished by assigning the former as master faces and the latter as slave faces, where a master face can be fully or partially connected to a number of slave faces. In the ASI there are no restrictions of the relative position of the master and slave faces so the vertices across the interface do not have to be coincident which is very useful especially when one of the subdomains is moving due to compression of the cells in each time step and therefore impossible to ensure a one to one correspondence between the attached boundary faces [6]. The coupling of the master and slave faces is done by projecting the vertices from the slave faces to the master face. The cell faces are then divided into subfaces where the total flux is apportioned accordingly in order to enforce conservation through the interface. As the sliding interface allows for continuous changes in cells connectivity a care has to be taken in addressing the adjacent boundaries. This is done by a indirect addressing methodology in STAR-CD [6].
3.7.1 Events

In order to specify and control the initialization of the ASI, an event file is generated. Several inputs are needed in order to specify an ASI, such as time, master and slave faces and tolerances which the cell faces and the vertices have to satisfy. Time is used to control the sequence of events to occur, where the time specified is the actual flow time that is simulated. An example of a macro used to generate an event file for merging an ASI is shown in appendix C.

3.8 User Subroutines

Controls of various features and operations such as mesh movement, boundary conditions and source terms is handled by adopting the usage of user subroutines. The user subroutines, programmed in Fortran are supplementary programs to the main program generated in Prostar. Several user subroutines have been adopted in this study and they will be described in the following subsections.

Wall Velocity - bcdefw.f

This user subroutine is used to specify the variations in the wall boundary conditions. In the CFD model there are two wall regions that move in time, the piston and the exhaust valve. As the movement of both piston and exhaust valve is parallel to the vertical axis of the coordinate system the movement in time can be described by only the vertical velocity component, \( w \).

The velocity of the piston is given in equation 2.5.4 in section 2.5.1 and the velocity of the exhaust valve is found from the measured valve lift profile where a fourth order polynomial is used for describing the velocity as a function of CAD. The \( bcdefw.f \) subroutine can be found in appendix D.1.

Pressure Boundary Conditions - bcdefp.f

In this user subroutine the values of the scavenging box pressure given in section 1.5 is loaded and the value is applied as an inlet pressure boundary condition at each time step. The program reads in the datafile containing the pressure data and stores them in a vector. As the sampling rate of the pressure data does not match the time step in the simulations, linear interpolation is used to estimate a value of the pressure at each time step. The \( bcdefp.f \) subroutine can be found in appendix D.2.

Mesh Movement - newxyz.f

This user subroutine is used to calculate new positions of the cell vertices for the simulation of piston and exhaust valve movement. The movement of the piston is simulated by
compressing the cells in the cylinder volume. The compression of the cells is controlled by
the position of the cell vertices and for each time step a new position for each vertex in the
cylinder volume is calculated. The position of the piston at each time step is known from
equation (2.5.3) and the piston position from previous time steps can also be calculated
using the same equation, giving the total movement of the piston at each time step. As
the exhaust valve moves into the cylinder, an upper limit for moving the vertices in terms
of piston motion is specified such that the cells that form the lower part of the exhaust
valve mesh are held unmoved in terms of piston motion. The distribution of the vertices
between the upper limit and the piston surface which is the lower limit, is given by linear
interpolation.

The movement of the vertices in the exhaust valve is done in a similar way as the movement
of the vertices in the cylinder mesh, while the exhaust valve position in time is given
from measurements which are read in from a datafile. Similar to the pressure boundary
conditions the data stored in the datafile do not match the time steps in the simulation
and linear interpolation has to be performed in order to find the position of the valve at
the specific time step and for the previous time step to calculate the distance the valve has
travelled since last time step. The newxyz.f subroutine can be found in appendix D.3.

Enthalpy Source Term - sorent.f

As combustion is not simulation in this work the gas inside the cylinder is expanded using
the estimated heat release rate from the measurements given in section 1.5. The energy
released is given in a datafile which is read by the subroutine in the same manner as done
in both newxyz.f and bcdefp.f. The energy is added as a source term to the enthalpy
in equation (2.2.4). The source term specified in the user subroutine is given per unit
volume and the geometrical range of cells that are to be heated is given in the subroutine.
Two volumes for releasing the heat within are studied in this work; a volume which is
constant in time placed at the top of the cylinder close to the cylinder cover and a variable
volume which increases with the piston motion. The sorent.f subroutine can be found in
appendix D.4.

Scalar Source Term - sorsca.f

This subroutine is used to control the passive scalar defined for the fresh air charge in the
scavenging box. The scalar is first initialized as being zero in the whole domain and before
the piston opens for the scavenging ports, the value of the scalar is set equal to one in the
scavenging box and the scavenging port. The air supplied from the scavenging port will
therefore have a mass fraction equal one as long as no blowback occurs into the scavenging
port from the engine cylinder. The sorsca.f subroutine can be found in appendix D.5.
3.9 Parallelization

Parallelization of the numerical code is important in order to improve the CPU time for the simulations. As user subroutines and ASI are present in the simulations, several tasks have to be performed in order to run the simulations. As the total computational domain has been partitioned on several processors the local vertices are by default settings only available to the processor holding that partition. By specifying a special option, `-decompflags=novd` the total vertex set is available to all the processors and the user subroutine `newxyz.f` will only be called by the master processor which then broadcasts the new vertex positions to the remaining partitions.

As the model contains ASI, the boundaries that form the sliding interface have to be decomposed on the same partition. In order to satisfy this demand the computational domain has to be decomposed manually to a setfile which is called by the option, `-decompmeth=s`. The setfile is constructed in Prostar where a part of the mesh is assigned each processor manually using cell sets. An example of a setfile can be found in appendix E.

The datafiles used for reading in data containing scavenging box pressure, energy released in combustion and the valve position have to be copied to all processors using the option `~copy`.

Parallel Performance

Increasing the number of processors by a factor of two does not guaranty a corresponding reduction in the total CPU time. In figure 3.1 the performance of the parallelization is compared to the performance of the single processor run assuming a 100% efficiency when increasing the number of processors. The simulations testing the parallel performance were performed on one, four and eight processors.
The efficiency of the parallelization of the code can be estimated by comparing the total wall clock time for the single processor run to the total wall clock time for parallel run, as it is shown in figure 3.1. The parallel efficiency is given as follows:

$$E_p = \frac{t_s}{n_p t_p}$$

(3.9.1)

where $t_s$ is the total wall clock time for the single processor run, $n_p$ is the number of processors and $t_p$ is the total wall clock time for the parallel run. The parallel efficiency for the four processor parallel run is 89.2% and for the eight processor parallel run the efficiency is 80.9%. One reason for the reduction in parallel efficiency with increasing number of processors is due to the internal communication between the processors.
This chapter describes the generation of the computational grid, the considerations and assumptions that were made in order to construct a mesh that can be compared with the actual geometry of the 4T50MX engine described in section 1.5. The mesh is generated in Prostar using the available mesh module.

4.1 Basic Mesh - 57k Mesh

In this section the basic geometry of the computational grid is presented. The basic mesh is constructed as a very coarse mesh for testing the functionality of the mechanism developed for moving the mesh with sliding interface and the connectivity of the separate domains. In order to save computational time the total computational mesh is generated as a 12° sector out of a full circle. The 12° are chosen as the cylinder liner has 30 scavenging ports and a 12° sector is therefore dedicated to one scavenging port. As the geometry of the engine is presented as a sector, cyclic boundary conditions are used as symmetric flow conditions are assumed and as discussed in section 3.4 the flow variables are equal on each member of the cyclic boundary pair. The total computational domain consists of a scavenging box with scavenging port, cylinder volume, exhaust duct and the exhaust diffuser. An overview of the computational domain is shown in figure 4.1. The individual components of the computational domain are discussed and described later in this section.
Cylinder volume

As a starting point for the mesh generation, the cylinder volume with a diameter of 0.5 m is constructed using a structured cylindrical mesh. The piston surface and the cylinder cover are assumed to be flat as a simplification of the mesh generation. The height of the cylinder volume is chosen in order to satisfy the compression volume given in section 1.5. With a stroke of 2.2 m the height of the sector is 2.33513 m, giving the compression volume of 0.02653 m$^3$. The cylinder volume mesh can be seen in figure 4.2.
Figure 4.2: Bottom of cylinder volume mesh (left) and top of cylinder volume mesh (right) with the piston in BDC.

The numbers in figure 4.2 refer to the boundary conditions on the mesh as they are described in table 4.1.

Table 4.1: Boundary conditions on cylinder volume and cover.

<table>
<thead>
<tr>
<th>Boundary number</th>
<th>Boundary type</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Cyclic boundary</td>
</tr>
<tr>
<td>2</td>
<td>Cyclic boundary</td>
</tr>
<tr>
<td>3</td>
<td>Piston - Wall boundary</td>
</tr>
<tr>
<td>4</td>
<td>Liner - Wall boundary</td>
</tr>
<tr>
<td>5</td>
<td>Cover - Wall boundary</td>
</tr>
<tr>
<td>6</td>
<td>Port - Attach boundary</td>
</tr>
<tr>
<td>7</td>
<td>Valve - Attach boundary</td>
</tr>
</tbody>
</table>

Boundary no. 1 and 2 form the cyclic set of the repeated flow conditions on the walls of the sector. Boundary no. 3 is the moving piston boundary where the displacement and the velocity of that boundary has been described in section 2.5. Boundaries no. 4 and 5 are wall boundaries representing the cylinder liner and cover. Boundary no. 6 represents one half of the sliding interface forming the connection between the scavenging port and the cylinder volume and boundary no. 7 represents the one half of the sliding interface used for the exhaust valve configuration. The missing part of the cylinder mesh shown in figure
4.2 is the exhaust valve region, described in the following section.

**Exhaust valve**

The exhaust valve mesh that was modelled is shown in figure 4.3.

![Figure 4.3](image_url)

**Figure 4.3:** Upper Left: Exhaust valve mesh. Upper Right: Exhaust valve mesh and the top of the cylinder volume mesh (red). Lower: Closed exhaust valve

The numbers in figure 4.3 refer to the boundary conditions as they are described in table 4.2.
Table 4.2: Boundary conditions on the exhaust valve.

<table>
<thead>
<tr>
<th>Boundary number</th>
<th>Boundary type</th>
</tr>
</thead>
<tbody>
<tr>
<td>8</td>
<td>Cyclic boundary</td>
</tr>
<tr>
<td>9</td>
<td>Attach boundary</td>
</tr>
<tr>
<td>10</td>
<td>Valve - Wall boundary</td>
</tr>
<tr>
<td>11</td>
<td>Valve - Wall boundary</td>
</tr>
</tbody>
</table>

As seen in figure 4.3 the geometry of the exhaust valve is very simple as the valve is only modelled with straight lines and sharp edges. The exhaust duct downstream the exhaust valve is modelled as a circular cylinder where as in the test engine the geometry is more complex. Using this configuration of the exhaust valve where a sliding interface is used for the connection of the cylinder volume mesh and the exhaust valve mesh requires that the boundary between the cylinder volume and the exhaust valve is aligned for the vertical movement of the valve. The diameter of the exhaust duct is therefore increased in the CFD model compared to the actual geometry where a contraction follows right after the exhaust valve, this diameter increment is 12%. As seen in figure 4.3, boundaries number 10 and 11 are on each side of a thin plate and the lower part of the mesh is therefore not connected to the upper part and the flow is forced to flow around the thin plate as in the real valve.

The mesh in figure 4.3 shows the valve fully opened and the distance from boundary number 10 to the cylinder cover equals the maximum valve lift given in section 1.5. When the valve starts to close the thin plate moves upwards until it is flush with the cylinder cover as shown in figure 4.3.

Exhaust duct and diffuser

In the real engine the exhaust duct takes a 90° bend after the exhaust valve. For simplification of the mesh generation the bend is straightened out and the exhaust duct is modelled as a straight pipe. The width of the pipe is the same as the mesh in the valve region and in the center of the pipe the grid is removed to compensate for the diameter of the valve shaft.

The last component on the exhaust side of the domain is the exhaust diffuser as it is shown in figure 4.1. The outlet of the computational domain is placed on the diffuser and the exhaust pressure described in section 1.5 is applied at the boundary. The outlet of the diffuser has the same dimension as the diffuser in the real engine while the diameter at the inlet of the diffuser is higher due to the diameter of the exhaust duct and the exhaust valve. The diameter ratio of the numerical diffuser is 15% smaller compared to the actual diameter ratio in the real diffuser.
Scavenging port

As mentioned earlier a sector of $12^{\circ}$ is dedicated to one scavenging port and therefore only one scavenging port will be modelled. From the real geometry it is known that the cross-sectional area of the scavenging port is $8.328 \times 10^{-3} \text{ m}^2$ where the height is 0.21 m and the width is 0.04 m. The real scavenging ports have curved lines and no sharp edges while the port in the numerical model is square with sharp corners. Due to this the same dimensions can not be kept equal between the numerical and the real port geometry while the cross-sectional area and volume are kept equal to the real geometry. Figure 4.4 shows the scavenging ports along with the scavenging box and the alignment to the cylinder liner.

![Figure 4.4: Left: Scavenging port. Right: Alignment of scavenging box, scavenging port and cylinder volume, seen from below.](image)

As seen in figure 4.4 the center line of the scavenging ports is rotated $20^{\circ}$ to the radial direction of the cylinder volume in order to create the swirling motion of fluid inside the cylinder. The scavenging box is modelled as a circular chamber with the same height as the square scavenging box of the real engine. The scavenging box is modelled as as circular chamber in order to apply cyclic boundary conditions and therefore reduce the total number of cells in the mesh.

The total number of cells in the different domains of the basic mesh geometry is given in table 4.3.
4.2 Refinement Step One - 116k Mesh

As the basic mesh was primarily constructed for testing the mechanism of the moving mesh with sliding interfaces, the mesh requires refinement for better resolution of the flow field. In this refinement step the cylinder volume, scavenging port and exhaust valve are refined as the flow field in these parts of the domain are of most interest. The scavenging box is also refined around the scavenging port in order to smoothen the transition from the coarse cells in the scavenging box to more finer cells in the scavenging port.

The scavenging port is refined in tangential and vertical direction by a factor of four and in the streamwise direction the refinement is by a factor of two. This refinement is primarily done in order to resolve the flow field to more detail and then study the $y^+$ values for further refinement as described in section 4.3. The cylinder mesh is refined towards the cylinder liner with a factor of four and the piston surface is refined with a factor of eight for better resolution of the flow coming from the scavenging port. The exhaust valve which was very coarse in the basic mesh is refined to match the cell size of the cylinder mesh on the interface between the cylinder mesh and the exhaust valve. No refinement is undertaken in the exhaust duct region. The scavenging port, before and after refinements conducted in this step can be seen in figure 4.5.

<table>
<thead>
<tr>
<th>Domain</th>
<th>No. of cells</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cylinder volume</td>
<td>28320</td>
</tr>
<tr>
<td>Valve</td>
<td>2592</td>
</tr>
<tr>
<td>Exhaust duct</td>
<td>21712</td>
</tr>
<tr>
<td>Scavenging port</td>
<td>411</td>
</tr>
<tr>
<td>Scavenging box</td>
<td>4340</td>
</tr>
<tr>
<td>Total</td>
<td>57375</td>
</tr>
</tbody>
</table>

Table 4.3: Number of cells in basic mesh.
4.2 Refinement Step One - 116k Mesh

Figure 4.5: Refined scavenging port compared to original scavenging port.

The cylinder volume mesh for the 116k mesh in the top and bottom of the cylinder is shown in figure 4.6

Figure 4.6: Results of cylinder volume refinements performed in refinement step one.
The number of cells in each domain is given in Table 4.4.

**Table 4.4: Number of cells in mesh constructed in refinement step one.**

<table>
<thead>
<tr>
<th>Domain</th>
<th>No. of cells</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cylinder volume</td>
<td>65696</td>
</tr>
<tr>
<td>Valve</td>
<td>6848</td>
</tr>
<tr>
<td>Exhaust duct</td>
<td>21712</td>
</tr>
<tr>
<td>Scavenging port</td>
<td>12640</td>
</tr>
<tr>
<td>Scavenging box</td>
<td>9730</td>
</tr>
<tr>
<td><strong>Total</strong></td>
<td><strong>116626</strong></td>
</tr>
</tbody>
</table>

### 4.3 Refinement Step Two - 174k Mesh

The refinements undertaken in this step are based on results from the mesh constructed in section 4.2. The refinements on the mesh in this step are aimed at the scavenging port and the in-cylinder surfaces in order to satisfy the $y^+$ values for the incoming flow from the scavenging port based on simulations of the 116k mesh. A study of the $y^+$ values for the 116k mesh and the 174k mesh can be found in appendix F. The $y^+$ values on the piston surface are not only varying due to the variations in flow velocity but also due to the change in cell height as the cells are compressed due to the piston motion. The $y^+$ values of the piston surface are evaluated in two positions, under maximum flow through scavenging ports at 7 CAD after port opening and at TDC where the maximum values are $y^+ = 1371$ and $y^+ = 34$, respectively. In order to satisfy the range of $y^+$ values for the wall functions the grid cannot be refined so the values of $y^+$ are below 100 at all time. Refining the cells adjacent to the piston surface with a factor of four brings the maximum value to $y^+ = 342$ and at TDC the value becomes $y^+ = 8.5$. As both these values are the extremes of the engine cycle it is expected that the average $y^+$ values are within a reasonable range through the engine cycle.

The $y^+$ values on the liner wall are also evaluated at same CAD as for the piston, where for the liner wall the $y^+$ has a maximum value when the piston is at TDC with a value of $y^+ = 893$ and at 7 CAD after IPO the value is $y^+ = 299$. Refining the cells adjacent to the liner wall with a factor of four in the wall normal direction. The cyclic average $y^+$ values are assumed to take a reasonable value through the engine cycle in order to satisfy the wall function restriction.

For the cylinder cover the $y^+$ values have their maximum value at TDC where the value is $y^+ = 2940$ while at 7 CAD after IPO the value is $y^+ = 295$. Refining these cells adjacent to
the cylinder cover with a factor of six gives a better resolution for the wall functions, even though at some CAD the value will be considerably large for the wall functions. The cyclic mean value is assumed with the refinement to be within the range specified for the wall functions. Moreover the cells in the sliding interface region of the exhaust valve are refined with a factor of two in the vertical direction in order to get a better flow resolution in the exhaust valve.

The cells in the scavenging port are those who experience the most stable flow velocity over the engine cycle as the port is only connected to the cylinder at a part of an engine cycle where the pressure is relatively constant. The maximum $y^+$ value in the scavenging port is found at the top of the port at 330 CAD, where $y^+ = 4370$. With a refinement factor of 16 in the wall normal direction the maximum $y^+$ values are expected to be in the range of 270. This is a cycle maximum value and at other CAD the $y^+$ values are closer to 100. The side walls of the scavenging port are also refined as the maximum value of $y^+$ has been found to be 710. With a refinement normal to the wall of factor four the maximum value is expected to be around 170 and the average value within the range for the wall functions. Outside the scavenging process the $y^+$ values in the scavenging port are zero due to very low velocities. The scavenging port, before and after refinements conducted in this step can be seen in figure 4.7.

(a) Scavenging port from the 116k mesh. (b) Scavenging port from the 174k mesh.

Figure 4.7: Refinements made on the scavenging port based on $y^+$ values from the 116k mesh.

The cylinder volume mesh in the top and bottom of the cylinder for the 174k mesh can be seen in figure 4.8
4.3 Refinement Step Two - 174k Mesh

Further visualizations of the 174k mesh can be seen in appendix E.1. The total number of cells in the mesh along with the number of cells in each component are shown in table 4.5,

Table 4.5: Number of cells in mesh constructed in refinement step 2.

<table>
<thead>
<tr>
<th>Domain</th>
<th>No. of cells</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cylinder volume</td>
<td>76784</td>
</tr>
<tr>
<td>Valve</td>
<td>10368</td>
</tr>
<tr>
<td>Exhaust duct</td>
<td>21712</td>
</tr>
<tr>
<td>Scavenging port</td>
<td>53920</td>
</tr>
<tr>
<td>Scavenging box</td>
<td>11906</td>
</tr>
<tr>
<td>Total</td>
<td>174690</td>
</tr>
</tbody>
</table>
In this chapter a validation is sought for the CFD model. The validation is made as a sequence of several studies where the first study aims at finding a periodic solution for the model, i.e. how many engine cycles the model has to simulate in order to reach a cycle periodic solution. The different discretization schemes and time step size are then compared and the difference evaluated in order to find the optimal combination for further studies. The different volumes used for releasing the heat from combustion are compared and the difference studied. The different meshes introduced in chapter 4 are compared and a solution convergence is sought when the mesh is refined. Finally the turbulence models are compared and the difference in the solutions obtained is compared and evaluated.

5.1 Engine Cycle Dependency

As the simulations are started from a initial condition where the flow is at rest in the computational domain, the engine has to run several cycles until a periodic solution could be reached. Several parameters are examined in this context for the validation of a periodic solution. The basic model is used for the validation of a periodic solution as it is the most computationally efficient and is still able to produce a relative solution even though the flow field is not fully resolved in terms of grid resolution and solver parameters. The parameters that are examined are the in-cylinder pressure, mass flow rate through scavenging ports and the tangential velocity profile at TDC. For spatial discretization the UD scheme is used, the time step size is controlled by a maximum $C_r$ number of 100. The turbulence model used is the $k-\varepsilon$ RNG model.
5.1 Engine Cycle Dependency

In-cylinder Pressure

The variation in the in-cylinder pressure is examined as the cylinder pressure has a significant impact for the mass flow through the scavenging port and is also one of the results that can be compared to the experimental measurements that exist for the engine. The in-cylinder pressure is sampled at a point in the cylinder liner cover, as is done in the experiment described in section 1.5. The in-cylinder pressure for five consecutive engine cycles is shown in figure 5.1.

![Pressure on cylinder cover for five consecutive cycles.](image)

As figure 5.1 shows the cyclic variations of the in-cylinder pressure for the last four engine cycles are very small. The first cycle is different from the later cycles where a lower pressure occurs both at TDC and at maximum pressure. This is due to that the simulation starts with the piston positioned in BDC with fluid at rest and the scavenging process is not able to fill the cylinder volume with the same amount of mass of fresh air as is done in the later cycles. The result of this is lower pressure at TDC and lower maximum pressure. The pressure at TDC, the cycle maximum pressure and the relative difference to the pressure values in engine cycle two is given in table 5.1.
### Table 5.1: Pressure at TDC and maximum pressure for five consecutive engine cycles.

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Cycle 1</td>
<td>131.6</td>
<td>−1.49</td>
<td>162.9</td>
<td>−0.91</td>
<td>31.3</td>
</tr>
<tr>
<td>Cycle 2</td>
<td>133.6</td>
<td>−</td>
<td>164.4</td>
<td>−</td>
<td>30.8</td>
</tr>
<tr>
<td>Cycle 3</td>
<td>133.4</td>
<td>−0.15</td>
<td>164.2</td>
<td>−0.121</td>
<td>31.0</td>
</tr>
<tr>
<td>Cycle 4</td>
<td>133.4</td>
<td>−0.15</td>
<td>164.1</td>
<td>−0.24</td>
<td>30.7</td>
</tr>
<tr>
<td>Cycle 5</td>
<td>133.4</td>
<td>−0.15</td>
<td>164.1</td>
<td>−0.24</td>
<td>30.7</td>
</tr>
<tr>
<td>Experimental</td>
<td>151.9</td>
<td>12.0</td>
<td>171.9</td>
<td>4.34</td>
<td>20</td>
</tr>
</tbody>
</table>

Considering the last four engine cycles the cyclic variations are within 0.25% for all cycles, thus the values at cycle two are fully representing the later cycles in terms of simulation results.

The comparison with experimental results is performed at this point in order to evaluate if the CFD model is capable of predicting results that are representing the engine conditions. A more thorough comparison will be conducted in chapter 6. A difference of 12% in pressure is accepted at this point for further validation and use of the CFD model.

### Mass Flow Rate

The mass flow rate of each engine cycle and total mass of fresh air charge that flows through the scavenging ports in the scavenging process is also studied in order to validate the cyclic independence of the simulation. The mass flow rate profile through the scavenging port for five consecutive engine cycles is shown in figure 5.2, where the scavenging process labelled as cycle one occurs after the piston has reached TDC in engine cycle one.
Figure 5.2: Mass flow rate through scavenging ports for 5 consecutive cycles.

As shown in figure 5.2 the mass flow rate through the scavenging ports is zero when the piston closes the ports and the flow profiles that are shown, indicate the mass flow when the scavenging ports are open. The cycle-to-cycle variations in flow profiles are very small and the only visual variation is in engine cycle one, where the shape of the profile is slightly different from the other profiles. This difference could be a result of a lower pressure in cycle one compared to the other cycles and also as some fluid transients from the initial flow field still exist within the flow.

The total mass of fresh air charge delivered in each scavenging process along with the relative difference to the total mass of scavenging air in engine cycle two and the deviation from the experimental value is given in table 5.2.
Table 5.2: Total mass flow and relative difference to cycle two in total mass flow through the scavenging port.

<table>
<thead>
<tr>
<th>Engine cycle</th>
<th>Total mass [kg]</th>
<th>% from Cycle 2</th>
<th>% from Exp.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cycle 1</td>
<td>1.8481</td>
<td>0.53</td>
<td>10.6</td>
</tr>
<tr>
<td>Cycle 2</td>
<td>1.8580</td>
<td>–</td>
<td>10.2</td>
</tr>
<tr>
<td>Cycle 3</td>
<td>1.8618</td>
<td>0.20</td>
<td>10.0</td>
</tr>
<tr>
<td>Cycle 4</td>
<td>1.8612</td>
<td>0.17</td>
<td>10.0</td>
</tr>
<tr>
<td>Cycle 5</td>
<td>1.8623</td>
<td>0.23</td>
<td>9.9</td>
</tr>
</tbody>
</table>

The cycle-to-cycle difference in total mass of scavenging air is very small as shown in table 5.2. Variations in engine cycles two-five are showing even smaller difference than when compared to engine cycle one. This support the conclusion made in previous section that engine cycle two can be used to study the flow field and simulating several cycles is not necessary compared to these results. A deviation of 10% from the experimental value is accepted for further simulations and validation of the CFD model.

**Tangential Velocity Profile**

One way of evaluating the cyclic variations in the flow field is to study the tangential velocity profile at a specific position. A tangential velocity profile taken at a height of 20 mm above the piston surface in TDC is shown in figure 5.3.
As seen in figure 5.3 tangential velocity profiles for the four consecutive engine cycles are showing the same tangential velocity at all radial positions and the cycle-to-cycle variations are negligible. Even though the tangential velocity profiles give a good indication on how the flow will develop in the later engine cycles they do not give an overall indication of the flow field. In order to evaluate the cyclic variations in the total flow field at TDC the integrated angular momentum is calculated in order to estimate the total angular momentum of the flow field within the engine cylinder. The total volume integrated angular momentum along the cylinder axis, $L_z$ is given as follows [22],

$$L_z = \int \int (\vec{r} \times m\vec{v}) \cdot \vec{e}_z \, dV$$  \hspace{1cm} (5.1.1)

where $\vec{r}$ is the position vector, $m$ is the mass and $\vec{v}$ is the velocity vector. The total angular momentum along the cylinder axis of the in-cylinder flow at TDC is given in table 5.3.
Table 5.3: Angular momentum at TDC in four consecutive engine cycles.

<table>
<thead>
<tr>
<th>Engine cycle</th>
<th>$L_z$ [kgm$^2$/s]</th>
<th>% from Cycle 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cycle 2</td>
<td>3.392</td>
<td>–</td>
</tr>
<tr>
<td>Cycle 3</td>
<td>3.425</td>
<td>0.96</td>
</tr>
<tr>
<td>Cycle 4</td>
<td>3.430</td>
<td>1.1</td>
</tr>
<tr>
<td>Cycle 5</td>
<td>3.430</td>
<td>1.1</td>
</tr>
</tbody>
</table>

As seen in table 5.3 the cycle variation in the total flow field at TDC is very small. From these results it is concluded that engine cycle no. two is sufficient in order to study the details of the flow field as the changes in the flow field in later cycles are small, saving a significant amount of CPU time.

5.2 Comparing Discretization Schemes and Time Steps

In this section the difference in discretization schemes and time step size is examined. As described in section 3.2 the different discretization schemes were compared and the relative differences in the solution produced by these schemes are compared. The solution is compared to the available experimental data for validation. The time step size is also evaluated and results for different $C_r$ numbers are compared. The comparison is based on results from the computational grid described in section 4.2 with a total number of cells equal $116k$. The discretization schemes that are used are those described in section 3.2 and three different $C_r$ numbers; 50, 100 and 200 are compared. The evaluation of the discretization scheme and $C_r$ number is based on both results and the total CPU time required for the simulation.

All simulations in this section are started with a initial field at 316 CAD ABDC from a previous simulation, based on a second order $CD$ simulation in order to save CPU time for the comparison. The solution based on the $CD$ scheme at 316 CAD ABDC diverged at the opening of the scavenging port at 323 CAD ABDC, while the solution at previous CAD converged without complications and the solution at 316 CAD ABDC is therefore assumed to be sufficient as a restart. The $CD$ scheme is therefore not used for further study. In correspondence with the definition of the $C_r$ number then by increasing the $C_r$ number by a factor of two reduced the total CPU time by a factor of two.

Mass Flow Rate

The mass flow rate of fresh air delivered through the scavenging ports in the scavenging process for engine cycle two is compared for the different discretization schemes and different $C_r$ numbers. The mass flow profiles are shown in figure 5.4.
As seen in figure 5.4 there is no significant change in the mass flow rate by altering the discretization scheme or the \( C_r \) number. The mass flow rate profiles for the basic model shown in figure 5.2 are in good agreement with the mass flow rate profiles shown in figure 5.4. As figure 5.4 (a) shows there is no significant difference in the mass flow profiles for the different discretization schemes and the relative difference in total mass is within 1.0%. Comparing the time step size in figure 5.4 (b) shows no significant variation from \( C_r \) 50 to \( C_r \) 100 and the same is seen in figure 5.4 (c) between \( C_r \) numbers 50 and 100 while for 200 there is a little higher mass flow rate compared to the lower \( C_r \) numbers and the total mass of delivered air is about 2.5% higher compared to the lower \( C_r \) numbers.
Tangential Velocity Profiles

Velocity profiles are also evaluated inside the cylinder for comparing the solution obtained by the different discretization schemes and $C_r$ numbers. At TDC the tangential velocity profile is evaluated at a $z$ position of 20 mm above the piston surface. The tangential velocity ($V_\theta$) profiles are shown in figure 5.5.

![Comparing discretization schemes.](image)

![Courant number comparison for UD.](image)

![Courant number comparison for MARS](image)

**Figure 5.5:** Comparing tangential velocity profiles at TDC for different discretization schemes and Courant numbers.

As figure 5.5 (a) shows there is a difference between the velocity profiles produced by the first and second order discretization schemes. Both second order schemes are showing a maximum tangential velocity of around 28 m/s while the first order scheme is giving a maximum tangential velocity of 18 m/s. The shape of the tangential profiles is very similar for all three schemes and the motion of the fluid has a shape of a solid body rotation. The comparison of the time step size for the UD scheme shown in figure 5.5 (b) shows
that there is no significant change in the tangential velocity by altering the time step. A comparison of the time step size for the MARS scheme is shown in figure 5.5 (c). Altering the time step does not have any significant effect on the tangential velocity profile using the MARS scheme. From this comparison it is concluded that second order discretization scheme will be used for further simulations and the MARS scheme will be used as this is recommended in the user manual for STAR-CD [7].

**Axial Velocity Profiles**

From the conclusion in the previous section a further comparison of the MARS scheme with different Courant numbers is conducted. Instead of considering the same piston position and the tangential velocity profile, the axial velocity profile at BDC will be examined. At $z-$position of one diameter above the piston surface the axial velocity is found and plotted against the radial position. This is shown in figure 5.6

![Axial velocity profile](image)

**Figure 5.6:** Axial velocity profile at a distance of one diameter above piston surface.

As seen in figure 5.6 there is a minor variation in the velocity profile where the maximum velocity for a Courant number of 50 is larger compared to 100 and 200. The point of maximum axial velocity for $Cr = 200$ is shifted towards the center of the cylinder compared to $Cr = 50$ and $Cr = 100$ which have their maximum velocity close to the cylinder liner. Considering the maximum velocity of 47 m/s the relative difference at maximum is only 2.1% and therefore not significant compared to the CPU time saved.
5.3 Heat Release Volume

As mentioned in section 3.8 two different volumes have been used for releasing the heat from the combustion; a constant heat release volume and a variable volume which follows the piston surface during expansion. The relative difference in the solutions obtained using these two different heat release volumes is evaluated on the 116k grid using the second order MARS scheme and a $C_r$ number of 200. The constant volume, fixed in the top of the cylinder volume is defined with a radius of 0.23 m and a fixed height of 0.085 m where the distance to the cylinder cover is 0.03 m and the distance to the piston surface is 0.02 m at TDC. The distance from the heat volume to the piston surface increases due to the fixed position and height of the volume. For the variable volume the radius and the distance to the cylinder cover are the same as for the constant volume. The distance to the piston surface is now fixed at 0.02 m independent of the piston motion, leading to a increasing volume size. The position of the heat volume is illustrated in figure 5.7.

\[ \text{Figure 5.7: Illustration of heat release volume in the top of the cylinder volume, with piston positioned in TDC.} \]

The temperature field inside the cylinder right before opening of the scavenging ports for both constant and variable volume are shown in figure 5.8. The figure is showing the cells in the middle of the cylinder and scavenging port volumes.
As seen in figure 5.8 the temperature field is quite different for the two different volumes. For the constant heat volume the maximum temperature of the gas within the cylinder is right under the exhaust valve, while the maximum temperature for the variable volume is further down the cylinder. The localised maximum temperature under the exhaust valve occurs as the cells in the top of the cylinder are the only cells that are heated during the combustion process, while for the variable volume, a larger amount of cells in the cylinder volume are heated and the volume expands with the piston motion leading to a lower maximum temperature. For the cells adjacent to the piston surface the maximum and minimum local temperature for the constant volume are $800\, \text{K}$ and $318\, \text{K}$, respectively and for the variable volume the maximum and minimum temperatures are $861\, \text{K}$ and $318\, \text{K}$, respectively. The maximum temperatures are reached at approximately 15 CAD ATDC and the minimum temperatures are reached at approximately 30 CAD ABDC where the incoming air has displaced and mixed with the hot burned gas.
The density distribution in the lower half of the engine cylinder volume for both constant and variable volume is shown in figure 5.9 at a position of −30 CAD ABDC which is at the time of scavenging port opening.

\[ \text{Density distribution for constant volume.} \quad \text{(a) Density distribution for variable volume.} \quad \text{(b)} \]

**Figure 5.9:** Comparison of density \( \text{[kg/m}^3\text{]} \) variations in the lower half of the engine cylinder at −30 CAD ABDC for constant and variable heat release volumes.

As seen in figure 5.9 the density of the gas for the variable volume shown in 5.9 (b) is lower compared to the density of the gas with the constant volume shown in figure 5.9 (a). This was also expected from the temperature distribution shown in figure 5.8 where the temperature is higher close to the piston surface with the variable heat release volume. From these temperature and density considerations it is quite forward to look at the pressure within the cylinder through the scavenging process as it is shown in figure 5.10.
5.3 Heat Release Volume

As figure 5.10 shows the cylinder pressure is equal in both cases until opening of the exhaust valve where the pressure for the constant volume decreases faster compared to the variable volume. The pressure difference is again reduced in the scavenging process as the pressure of the incoming air is the same in both cases. The pressure difference could be due to lower density of the air in the exhaust valve region of the constant volume case due to higher temperatures. As the density is lower, the air is lighter and therefore will react faster to the pressure gradient above the exhaust valve. This pressure difference at the beginning of the scavenging process affects the incoming air as can be seen in figure 5.11, which shows the total cylinder mass and the mass of the trapped fresh charge for the two heat release volumes.

Figure 5.10: Comparing the pressure inside the cylinder for the different heat release volumes.
As figure 5.11 shows, at the beginning of the scavenging process a little more fresh air is entering the cylinder from the scavenging port for the constant volume case compared to the variable volume. This is directly related to the pressure inside the cylinder which is higher for the variable volume case and the driving pressure difference is therefore smaller in the case of variable volume.

The difference in the flow field is also considered in terms of influence to the scavenging process. The axial velocity component for the gas in the lower half of the engine cylinder at −30 CAD ABDC is shown in figure 5.12.
5.3 Heat Release Volume

As seen in figure 5.12 the difference in the axial velocity component is quite large at the beginning of the scavenging process where the axial velocity is higher in the case of constant volume with a maximum value of 123 m/s and the maximum value for the variable volume is 95 m/s. The velocity distribution in figure 5.12 shows that in the case of constant volume, negative axial velocity occurs in the middle of the cylinder, while in the case of variable volume the incoming air from the scavenging port pushes the hotter and lighter gas upwards leading to a positive axial velocity in the middle of the cylinder. Figure 5.11 shows this difference in the early stages of the scavenging process, while at the closing of the scavenging port there is no mass difference in the cylinder charge for the two cases.

In order to evaluate if the method of simulating the combustion from the earlier cycle affects the flow field in the upcoming cycle, the tangential velocity profiles at −10 CAD ATDC for both methods are shown in figure 5.13.
As seen in figure 5.13 the difference in the tangential velocity profiles is negligible and the only notable difference is close to the cylinder liner where the difference in the maximum values is 1.34%.

From this analysis of the heat release volume it is concluded that the temperature distribution for the variable volume is more likely to be closer to the real temperature distribution in the engine as the heat from combustion in the real engine is released within a variable volume during the combustion process. Using a constant volume throughout the combustion process is not as realistic as the heat will not be retained in the top of the cylinder in the real combustion process. The velocity distribution is different at the beginning of the scavenging process due to density differences in the engine cylinder. This difference vanishes during the scavenging process and prior to the combustion in next engine cycle the difference in the tangential velocity profiles shown is negligible. In terms of the scavenging process using adiabatic walls there is no significant difference between these two methods as the mass difference evens out at the end of the scavenging process. In terms of heat transfer calculations the temperature distribution in the engine cylinder is important for the local wall temperature and therefore the variable volume is assumed to give more realistic values in terms of heat transfer. For further simulations the variable heat release volume will be used.
5.4 Grid Resolution Study

In the process of this study several grid refinements have been constructed in order to get a higher resolution for the flow field and also to satisfy the $y^+$ values for the wall functions in the turbulence models. Solutions obtained from the different grids are compared and the relative difference between grids is evaluated. The different grids that are compared are those presented in chapter 4. The study of grid independent solution has not been based on known methods, such as the grid convergence index (cgi), as the refinements of the grids are made locally and non uniformly in coordinate directions. In [10] it is stated that it is not absolutely necessary to do such a study as the parameters of interest can be evaluated and the rate of change between grids can be evaluated without calculating the cgi number. In [10] it is specified that the number of grids to be used for such a study are preferably three. For the grid resolution study the MARS scheme with a maximum $C_r$ number of 200 is used for solver settings and for heat release the variable volume is used. A study of the $y^+$ values for the different grid refinements can be found in appendix F. The first parameters studied are the mass flow and the cylinder pressure.

Mass Flow Rate & Pressure

The mass flow rate profiles and the cylinder pressure over the scavenging process for the different grid sizes is shown in figure 5.14.

![Figure 5.14](image)

(a) Mass flow rate through scavenging ports.

(b) Pressure acting on cylinder cover.

**Figure 5.14:** Comparing mass flow rate through scavenging ports with total mass of scavenging air and pressure acting on cylinder cover for different grid sizes during the scavenging process.

As seen in figure 5.14 (a) there is a large difference in the mass flow rate profiles and the total mass of scavenging air. The difference in the first stages of the mass flow rate profiles is explained by the difference in cylinder pressure, shown in figure 5.14 (b). At the intake port opening the pressure in the 57k mesh is quite lower compared to the other
meshes resulting in a higher mass flow rate at $-30$ CAD ABDC. As the pressure in the scavenging box is the same in all meshes due to the boundary condition, the pressure balances and at $-10$ CAD ABDC the pressure is fairly even in all meshes. The mass flow is showing more variations over the scavenging process compared to the pressure, not giving a clear indication on a convergence of the mass flow rate. The total mass of scavenging air is increasing with increasing number of grid points and the relative difference to the experimental value is reduced to 7.2% for the 174k mesh compared to 9.0% for the 57k mesh.

The cylinder pressure over one engine cycle is also compared in terms of convergence. The pressure profiles can be seen in figure 5.15.

![Figure 5.15: Comparing cylinder pressure for the 57k, 116k and the 174k meshes.](image)

As seen in figure 5.15 there is a small difference in the cylinder pressure when comparing the higher resolution meshes to the lower resolution mesh. The pressure at TDC is 130 bar for the higher resolution meshes while the pressure in lower resolution mesh is 131.5 bar at TDC. The pressure increase over the combustion is reduced to 25 bar, while for the 57k mesh the pressure increase was 30 bar. The difference in pressure at TDC rises from difference in both trapped mass and also a difference in the mean temperature. For the 57k mesh, 1.3% more mass is retained inside the cylinder, leading to a higher pressure as the volume and temperature are the same. The difference in pressure rise across the combustion is due to the large cell size in the 57k mesh where more heat is added, as a part of a large cell can be lying outside the prescribed volume where the heat is released. This will result in a larger geometrical volume heated in the 57k mesh compared to the higher resolution meshes.
Tangential Velocity Profiles

The tangential velocity profiles as a function of radial distance at TDC for the different grid sizes can be seen in figure 5.16.

As presented in figure 5.16 the tangential velocity seems to be converging as the difference between the 174k and the 116k meshes is smaller compared to the difference between the 116k and 57k meshes. This evolution of the tangential velocity profile gives a good indication of the convergence of the solution for increased number of grid points.

Angular Momentum

The tangential velocity profiles give a good indication of the development of the flow at one particular position within the cylinder. In order to evaluate the development of the total flow field inside the cylinder for the different grid sizes the total angular momentum for the cylinder volume is evaluated and shown in figure 5.17.
5.5 Comparing Turbulence Models

As described in section 2.3 the turbulence models introduced are compared and the difference is evaluated. The only difference in the CFD model is the turbulence model and other parameters are held the same in order to study the influence of the turbulence model. The simulations are conducted using the MARS scheme and a maximum $C_r$ number of 200.

The angular momentum inside the cylinder shown in figure 5.17 shows a similar convergence as the tangential velocity profiles presented in figure 5.16, where the magnitude is increasing with increased grid points. Angular momentum created with the 174k mesh is higher during the scavenging process compared to the lower resolution meshes. The higher mass flow rate for the 174k mesh seen in figure 5.14 (a) produces a stronger angular momentum during the scavenging process while after the closing of the exhaust valve (EVC) the dissipation in angular momentum is faster for the higher resolution meshes compared to the 57k mesh, which could be due to the boundary layer which is better resolved in the higher resolution meshes. With an higher resolution in the boundary layer the modelling of the shear stress becomes more accurate leading to a more accurate friction in the boundary layer.

The results of this grid resolution study have shown that the approximated solutions obtained seem to be converging towards a final solution with an increased grid resolution. A grid independent value has though not been reached and further grid refinement is needed in order to evaluate the approximations further. The 174k mesh will be used for presenting the final results in this study as it is assumed that the results from the 174k mesh have shown to be converging towards a final solution.
on the 174k model, using the $k-\varepsilon$ RNG turbulence model with standard and modified $C_{\mu 4}$ constant, the standard $k-\varepsilon$ model and the $k-\omega$ SST model. Variable volume is used for releasing the heat from combustion.

**Mass Flow Profiles & Cylinder Pressure**

As shown in previous section the mass flow rate profiles through the scavenging port are not strongly influenced by the variations in discretization schemes or time steps and the flow field is more depending on these parameters. The influence on the mass flow rate from the turbulence models is also studied and the profiles along with cylinder pressure profiles can be seen in figure 5.18.

![Mass flow rate through scavenging port.](image)

![Cylinder pressure over one engine cycle.](image)

**Figure 5.18:** Comparing mass flow rate and pressure profiles for the different turbulence models.

As for the results of the validation studies made in previous sections, there is no significant variation in the mass flow rate through the scavenging ports by varying the turbulence model. The standard $k-\varepsilon$ model is the only model that is giving small variations in the shape of the profile through the opening of the port while the total mass delivered is equal to the $k-\varepsilon$ RNG model. The total mass delivered through the scavenging ports for all models is given in table 5.4.

<table>
<thead>
<tr>
<th>Turbulence Model</th>
<th>Total Mass [kg]</th>
</tr>
</thead>
<tbody>
<tr>
<td>$k-\varepsilon$ RNG Standard</td>
<td>1.922</td>
</tr>
<tr>
<td>$k-\varepsilon$</td>
<td>1.922</td>
</tr>
<tr>
<td>$k-\varepsilon$ SST</td>
<td>1.89</td>
</tr>
<tr>
<td>$k-\varepsilon$ RNG $C_{\mu 4}$</td>
<td>1.922</td>
</tr>
</tbody>
</table>

**Table 5.4:** Total mass of scavenging air delivered for all turbulence models.

The in-cylinder pressure shown in figure 5.18 (b) shows that the $k-\varepsilon$ RNG model is predicting a higher pressure in TDC compared to the standard $k-\varepsilon$ and the $k-\omega$ SST model. The pressure difference is though very small and can be explained in figure 5.19 (a)
which shows that the total mass of cylinder predicted by the RNG model is a little higher compared to the other models, giving a small deviation in the total pressure.

The total mass of cylinder charge, and the trapped fresh air charge are also studied in terms of the turbulence models to see the influence of the turbulence model to the scavenging process and scavenging parameters such as the scavenging efficiency. The total mass of cylinder charge and the fresh mass retained within the cylinder can be seen in figure 5.19.

![Figure 5.19](image)

**Figure 5.19:** Total mass of cylinder charge and retained mass of scavenging air for the different turbulence models.

As seen in figure 5.19 (a) there is a small difference in the total mass of cylinder charge at IPO where the \( k - \omega \) SST model shows a higher cylinder mass. At EVC the RNG models have a higher mass of cylinder charge compared to the standard \( k - \varepsilon \) and the \( k - \omega \) SST model. The fresh mass in cylinder charge shown in figure 5.19 (b) shows very similar behaviour for all models until BDC where the amount of fresh air in the cylinder is smaller for the \( k - \omega \) SST model. The ratio of fresh mass and total mass, giving the scavenging efficiency (\( \eta_{sc} \)) is shown in table 5.5.

<table>
<thead>
<tr>
<th>( k - \varepsilon ) RNG</th>
<th>Standard ( k - \varepsilon )</th>
<th>( k - \omega ) SST</th>
<th>( k - \varepsilon ) RNG Mod. ( C_{\mu4} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \eta_{sc} )</td>
<td>0.983</td>
<td>0.982</td>
<td>0.973</td>
</tr>
</tbody>
</table>

The scavenging efficiency shown in table 5.5 is indicating that the \( k - \omega \) SST model is predicting more mixing of cylinder gasses during scavenging compared to the \( k - \varepsilon \) models.

The comparison of both mass and pressure for the different turbulence models has shown that all models are predicting very similar results. The RNG models are though predicting
a higher pressure in engine cylinder which is also confirmed by the work of Lendormy et al. [19] who showed higher pressures predicted by the RNG model.

### Tangential Velocity Profiles

It has been shown that the variation in turbulence models does not have a significant influence on the scavenging process and the mass flow rate through the engine. Velocity profiles are also studied as it has been shown in section 5.2 that even though the difference in mass flow rate is small, the velocity profiles have shown some variations. The tangential velocity profile at a height of 20 mm above the piston surface with the piston in TDC is shown in figure 5.20.

![Figure 5.20: Comparing tangential velocity profiles at TDC for the different turbulence models.](image)

As figure 5.20 shows there is a difference in the tangential velocity for the different turbulence models. The RNG model is predicting the largest tangential velocity for both values of the model constant $C_4$, while the standard $k-\varepsilon$ is predicting the weakest velocity. Altering the model constant seems to have an effect on the dissipation of momentum resulting in a lower tangential velocity. The work of Lendormy et al. [19] also showed a stronger velocity magnitude for the RNG model compared to the standard $k-\varepsilon$ model. The difference in the velocity profiles is related to the turbulent viscosity and therefore the dissipation which the work of Lendormy et al. [19] showed that is smaller in the RNG model compared to the standard $k-\varepsilon$ model. The tangential velocity predicted by the $k-\omega$ SST model gives a value in between the $k-\varepsilon$ models, indicating that the standard $k-\varepsilon$ model is probably underestimating the tangential velocity.
Angular Momentum

The angular momentum of the gas in the engine cylinder is evaluated for the different turbulence models, and shown in figure 5.21.

As figure 5.21 shows, there is a difference in the angular momentum produced where the RNG model is giving a stronger momentum compared to the standard $k-\varepsilon$ model and the $k-\omega$ SST model. The angular momentum predicted by the $k-\omega$ SST model is though very similar to the RNG model and the difference is much less compared to the standard $k-\varepsilon$ model. The conclusion is again that the standard $k-\varepsilon$ model is underestimating the velocity in the whole cycle compared to the results from the other models which are very similar looking at the angular momentum.

Turbulent Kinetic Energy

A time series has been produced where the volume integrated, mass averaged turbulent kinetic energy is calculated for engine cylinder volume. The turbulent kinetic energy as a function of CAD is shown in figure 5.22.
Comparing Turbulence Models

As shown in figure 5.22(a) there is a quite large difference in the turbulent kinetic energy at the time when the exhaust valve is open. When the exhaust valve is closed the rate of turbulent kinetic energy falls down to similar magnitude for all turbulence models. In Lendormy et al. [19] it is stated that the level of turbulence is very sensitive to the quality of the mesh, and reasonable convergence for the turbulence quantities was reached in that study at a very fine mesh with a maximum $C_r$ number of 9.2, which is significantly lower than the model $C_r$ of 200. It is though quite clear that the standard $k-\varepsilon$ model is predicting a turbulent kinetic energy which is unrealistic compared to the RNG model and the $k-\omega$ SST model. In figure 5.22(b) the turbulent kinetic energy is shown where the exhaust valve is closed. For these piston positions the turbulence models are showing similar values of the kinetic energy and they share a similar trend.

From the study of turbulent kinetic energy it was shown that at valve opening the standard $k-\varepsilon$ model is showing values in the exhaust valve region that are unrealistic compared to the values of the RNG model and the $k-\omega$ SST model. The work of Lendormy et al. [19] who showed very similar values for both of these models through the whole engine cycle also supports that the standard $k-\varepsilon$ model is not giving realistic values for the CFD model. The RNG model and the $k-\omega$ SST model are giving very similar values for all quantities in this comparison but as the RNG model has been recommended for simulations in large two-stroke engines by Lendormy et al. [19] and Antila et al. [3] the conclusion is to use the RNG model for further study.

Figure 5.22: Turbulent kinetic energy for the different turbulence models.
The results presented in this section are based on the $174k$ model, which has been concluded to give reasonable results based on the numerical validation conducted in previous sections. Even though there still exists a discretization error in the model it has been shown that this error is reduced compared to the basic model. The time step is adjusted using a maximum $Re$ number of 200 and the discretization scheme is the MARS scheme. The variable volume is used for releasing the heat from combustion and the turbulence model used is the $k-\varepsilon$ RNG model. The simulation is started with fluid at rest and ran until the TDC in third engine cycle. The total simulation time was 269 hours for 900 CAD.

Previous chapter was mostly based on relative comparison of the results in terms of validation of the numerical model and the physics of the flow were not discussed in any detail. In this chapter the main goal is to study and evaluate the physics into more detail and give a better indication of the flow in the whole engine cycle.

### 6.1 Flow Visualization

An important part of the study is to visualize the flow field in the flow domain. Several flow visualizations are presented; in engine cylinder, scavenging port and in the exhaust valve. Figures on the following pages are showing velocity magnitude of the flow in the whole engine and the individual velocity components. The piston positions shown in the figures are positions characterized with an event in the engine cycle, such as opening or closing of the scavenging port and the exhaust valve. These flow visualizations are mainly shown for the reader to get a picture of the flow conditions in the domain through a whole engine cycle. The velocity magnitudes shown on the contour plots are scaled to give a higher resolution of the flow inside the cylinder and is therefore not fully representing the extreme high or low local velocities.
Figure 6.1: Velocity magnitude in m/s, showing the gas flow in the whole engine, including scavenging port, engine cylinder and exhaust valve.
6.1 Flow Visualization

Figure 6.2: Radial velocity component in m/s, showing the gas flow in the whole engine, including scavenging port, engine cylinder and exhaust valve.
Figure 6.3: Tangential velocity component in m/s, showing the gas flow in the whole engine, including scavenging port, engine cylinder and exhaust valve.
Figure 6.4: Axial velocity component in m/s, showing the gas flow in the whole engine, including scavenging port, engine cylinder and exhaust valve.
6.1 Flow Visualization

Figure 6.1 (a) shows the velocity magnitude with the piston in TDC and scavenging port and exhaust valve are closed leading to a low velocity in the domain. In figure 6.1 (b) the opening of the exhaust valve occurs leading to a high velocity in the exhaust duct, while the velocities in the cylinder are still low. In figure 6.1 (c) the scavenging port has opened and the fresh air charge from the scavenging port begins to flow into the cylinder. The exhaust valve lift has increased further and the burned gas is flowing out of the cylinder through the valve. In figure 6.1 (d) the piston has reached BDC and the scavenging ports are fully opened. In figure 6.1 (e) the inflow from the scavenging ports is significantly reduced as the port is about to be closed. The exhaust valve is still fully open and the remaining burned gas is pushed out the valve by the fresh air charge in the bottom of the cylinder. In figure 6.1 (f) the exhaust valve lift is almost zero and only small amount of air is flowing through the valve. After the closing of the exhaust valve the piston will compress the air and reach the TDC as shown in 6.1 (a).

Figures 6.2 - 6.4 show the velocity components for same piston positions as the velocity magnitude, in order to get an idea of how the distribution is for each component. For more direct comparison of the three velocity components \( v_r \), \( v_\theta \) and \( v_z \) for the scavenging port, cylinder and exhaust valve are shown on figure 6.5.
6.1 Flow Visualization

![Image: Flow visualization of velocity components](image)

Figure 6.5: Velocity components $v_r$, $v_\theta$ and $v_z$ in m/s showing the flow field within the engine cylinder at BDC.

As seen in figure 6.5 (a) the radial component $u$ is large in the scavenging port and exhaust valve while the distribution is fairly uniform within the engine cylinder where the radial component is almost zero. As seen in figure 6.5 (b) the tangential velocity component $v$ is large indicating a strong swirling motion of the gas. The distribution is quite similar throughout the cylinder height with the lowest velocity in the middle of the cylinder and the strongest velocity close to the cylinder liner. The axial velocity component shown in figure 6.5 (c) shows a separation bobble on the liner above the scavenging port and in the middle of the cylinder. At half the cylinder height the separation formed close to the scavenging port has vanished and the axial velocity in the middle of the cylinder is directed upwards. The upstream effect of the exhaust valve is clearly seen in the upper half of the cylinder where the axial velocity is negative in a separation formed upstream the valve which acts as a forward facing step for the flow. The flow outside the upstream exhaust valve effect is accelerated towards the exhaust duct. The flow velocities in the exhaust valve region are much higher then those shown in figure 6.5 in order to resolve the
6.1 Flow Visualization

gradients inside the cylinder for visualization.

The flow in the lower half of the engine cylinder with the entrance effect from the scavenging port is shown in figure 6.6. It should be noted that the vectors have been thinned for visualization and the distance between vector points does not reflect the actual grid resolution.
Figure 6.6: Flow in lower half of the cylinder showing the entrance effect from the scavenging port for several piston positions.
As shown in figure 6.6 (a) the flow is entering the engine cylinder from the scavenging port with high velocity producing a flow separation on the piston surface due to the piston skirt which acts as a forward facing step. At the liner wall above the scavenging port the flow separation discussed in connection with figure 6.5 (c) is also shown. Looking at figure 6.6 (b) shows that the separation bubble at the liner wall above the port has moved downwards closer to the port compared to figure 6.6 (a). The air entering the cylinder is pushed towards the liner wall due to the separation bubble in the middle of the cylinder. In figure 6.6 (c) the vortex core has decreased and the separation bubble at the cylinder liner has moved away from the scavenging port. The entering flow has a wider passage due to the decreasing vortex and from the length of the vectors it can be seen that the flow velocity is reduced compared to figure 6.6 (b). In figure 6.6 (d) the separation bubble at the liner wall has moved down to the scavenging port again and the vortex core is further decreased leading to a wider passage for the incoming flow. The opening of the scavenging port is decreasing due to the upwards piston motion. In figure 6.6 (e) the scavenging port is almost closed and only a small opening is supplying the cylinder with scavenging air. The separation bubble seen at the liner wall and the vortex core have grown, leading to a narrower passage for the flow in the axial direction.

The flow through the exhaust valve along with the part of the cylinder closest to the valve and cylinder cover is shown in figure 6.7.
Figure 6.7: Flow through the exhaust valve for several piston positions.
In figure 6.7 (a) the exhaust valve is lifting and the burned gas leaves the cylinder. Due to high pressure difference through the exhaust valve the velocity in the exhaust valve becomes high compared to the velocities in the cylinder. In figure 6.7 (b) the opening of the exhaust valve is further increased. At the surface of the exhaust valve the flow separates due to sharp edges in the geometry of the valve which are not existing in the real valve geometry. At the wall above the exhaust valve opening a separation bubble is forming, similar to the separation bubble seen above the scavenging port. In figure 6.7 (c) the exhaust valve is fully open and the velocity is increasing in the cylinder. The separation bubble on both the exhaust valve and the wall above the opening has grown in size. In figure 6.7 (d) the upstream effect of the exhaust valve on the flow in the cylinder can be seen where the cylinder flow in the middle of the cylinder is directed downwards. Along the liner wall the flow is directed upwards and out the exhaust valve. In figure 6.7 (e) the exhaust valve is moving upwards starting to close and similar flow conditions are seen in the cylinder as in figure 6.7 (d). In figure 6.7 (f) the exhaust valve is almost closed and the flow into the exhaust duct is directed in radial direction leading to an increased size of the separation bubble at the wall. The velocity in the exhaust duct is decreased both due to smaller pressure difference and also due to decreased opening of the exhaust valve.

Mach number

The velocity vectors shown in figure 6.7 and velocity magnitudes in figure 6.1 showed high velocities in the exhaust valve. As high pressure difference occurs through the exhaust valve at the time of opening, high Mach numbers are expected to occur. Figure 6.8 (a) shows a plot of the Mach number in the exhaust valve, 15 CAD after opening of the exhaust valve.

![Mach number](image1)

![Fluid density](image2)

**Figure 6.8:** Mach number and fluid density in the exhaust valve region, 15 CAD after opening of the exhaust valve.
The maximum Mach number of 2.27 is in good agreement with numerical simulations conducted on the actual engine geometry with a steady geometry conducted at MAN Diesel which are giving a maximum Mach number in the exhaust valve at 2.17. As figure 6.8 (b) shows there is no clear line in the density gradient showing where a shock is produced. Through the opening of the exhaust valve where the flow reaches Mach one, a decrease in density can be observed. Further in the exhaust duct where the flow reaches Mach two a decrease in density is also observed, but no clear line showing a significant decrease is identified.

Scavenging Port

The flow through a horizontal (x-y) plane at a height of half the port height (explained in figure 6.9) is shown in figure 6.10 for six different CAD from IPO to IPC. Velocity vectors are indicating the direction of the flow and the contour is showing the out of plane velocity component. It should be noted that the vectors and magnitude contour shown on figure 6.10 are vertex data.

![Figure 6.9: Horizontal x-y plane at a height of half the port height.](image-url)
As seen in figure 6.10 the flow through the scavenging ports undertakes some variations through the port opening. Flow is entering the scavenging ports on the right hand side and leaving the ports into the engine cylinder on the left hand side. In figure 6.10 (a) the opening area of the scavenging port is small and therefore the velocities at this plane are relatively small. On the left hand side the velocity vectors are directed out of the plane due to the small opening in the top of the scavenging port. In the upper right corner a recirculation zone is forming due to flow separation when the flow enters the scavenging port. The maximum velocity in port direction at this position is 25 m/s. In figure 6.10 (b) the separation zone in the upper right corner has increased and the magnitude of the velocity has also increased. Due to the flow separation the effective flow area of the port is reduced leading to a large flow resistance. The flow vectors are now directed more into
the engine cylinder as the piston has opened a larger area of the scavenging port. The maximum velocity in port direction at this position is 40 m/s.

In figures 6.10(c) and (d) the scavenging port is almost at maximum opening and the flow direction is in the $x-y$ plane into the cylinder. The magnitude of the out of plane velocity component is decreasing as the port opening is at maximum. The maximum velocity in port direction is very similar for both positions, around 40 m/s. In figure 6.10(e) a small separation zone is forming in the lower left corner which is growing in size as can be seen in figure 6.10(f). The separation in the lower left corner could be due to a secondary current resulting from the forward facing step as the piston has partly closed the scavenging port, forming a wall. The work of Wilhelm et al. [9] showed that at a forward facing step secondary vortex currents occur which are directed to the sides and upwards along the step. Similar flow structures are seen in the scavenging port and the partial closure of scavenging port is assumed to cause this secondary current.

Cylinder Temperature

The cylinder temperature is shown in figure 6.11 for several piston positions. The piston positions shown for the temperature are different from those shown for the velocity as more focus is on the heat release period for the temperature visualization.
Figure 6.11: Temperature ([K]) of the fluid for several piston positions.
6.2 Mass Flow Rate

The maximum temperature of the engine cycle is 2400 K, reached at 30 CAD ATDC. The heat release period ends at 60 CAD ATDC and energy is no longer added to the system leading to a drop in cylinder temperature. The mixing of the burned gas and the fresh scavenging gas is shown in more detail in section 6.6.

6.2 Mass Flow Rate

The mass flow rate through the scavenging port, the cylinder pressure and scavenging port pressure are shown in figure 6.12 as a function of CAD from the opening to closing of the scavenging ports.

![Figure 6.12: Mass flow rate profile and pressure profiles from scavenging box and engine cylinder.](image)

As seen in figure 6.12 the shape of the mass flow profile through the scavenging port is in good agreement with the pressure profiles. In the beginning of the scavenging process the cylinder pressure drops rapidly and the pressure in the scavenging box is quite high. As fresh air is entering the cylinder at scavenging box pressure the cylinder pressure is slowly increasing resulting in a drop in the mass flow rate. Due to pressure variations in the scavenging box the mass flow rate is also varying, resulting in an increase in flow rate at $-10$ CAD ABDC. As the pressure difference is decreasing and the area of port opening is decreasing the mass flow rate is also decreasing towards the closure of the scavenging ports. The total mass of scavenging air predicted in the CFD model, 1.92 kg is in good agreement with the experimental value of 2.069 kg, with a relative difference of 7.1%. The experimental value of the cylinder pressure is also shown in figure 6.12 (b). The CFD estimated value of the pressure is in good agreement with the measured pressure during expansion. During opening of the scavenging ports the pressure from the simulations begins to deviate from the measured pressure probably due to pressure fluctuations that exist in the exhaust duct, where the pressure is taken to be a constant value in the simulations.
6.3 Pressure Profile

The cylinder pressure from the CFD calculations along with the experimental pressure is shown in figure 6.13 as a function of CAD.

As discussed in section 2.2 the difference in using the ideal gas law as the equation of state instead of using an equation for a real gas introduces a difference of 6.5% in the pressure. The difference in the pressure at the closure of the exhaust valve is 6.5% which is not related to the difference in real and ideal gas, as for a pressure level of 4 bar the assumption of neglecting the molecular interactions is not as significant as for higher pressure levels as occur in TDC. The difference at the closure of the exhaust valve is more related to the difference in the pressure of the exhaust boundary. The assumption of using constant pressure in the CFD model could lead to a lower pressure during valve opening and therefore the cylinder pressure becomes lower compared to the experimental pressure at EVC. In figure 3.6 (a) the 13% curve is showing the estimated cylinder pressure from the CFD model multiplied with the pressure difference induced by the ideal gas law assumption and the difference at exhaust valve closure.

In figure 6.13 (b) the cylinder pressure after the closing of the exhaust valve is shown. As seen on the pressure curves, pressure waves are travelling in the cylinder due to the closure of the exhaust valve. The amplitude of the pressure waves from the experimental pressure is much higher compared to the amplitude calculated in the CFD model. The frequency of a pressure wave is found from figure 6.13 (b) and for the CFD model the frequency is found to be 255 Hz. The speed of sound can be found as follows [23].
\[ c = \sqrt{\gamma RT} \]  

(6.3.1)

where \( \gamma \) is the specific heat ratio, given as 1.393 for dry air at 400°C. Considering the conditions in the cylinder at \(-60\) CAD ATDC shown in figure 6.13 (b) the mean cylinder temperature is 466 K, giving a speed of sound at \( c = 431.6 \) m/s. The distance from the piston to the cylinder cover is 0.845 m, giving at total distance for the pressure wave to travel between peaks of 1.69 m. Assuming that the pressure waves are travelling in the axial direction the frequency can be estimated as \( \frac{433.6}{1.69} = 256.5 \) Hz. The pressure waves seen in figure 6.13 (b) are therefore travelling in the axial direction in the cylinder.

### 6.4 Tangential Velocity & Swirl Ratio

The distribution of the fuel spray is important in terms of fuel efficiency and the strength of the swirling gas affects the distribution of the fuel. Tangential velocity profiles for 3 z-positions, 20 mm, 70 mm and 115 mm above the piston surface as a function of the radial distance are shown in figure 6.14.

![Tangential velocity profiles](image)

**Figure 6.14:** Tangential velocity profiles for three different positions above the piston surface 10 CAD before TDC.

As presented in figure 6.14 the shape has some variations of the tangential velocity profile with the different z-positions. The velocity profile for the 70 mm position has the shape of a solid body rotation while the other profiles are slightly deviating from the solid body shape. It can though be concluded that the velocity field is considerably uniform with a
solid body rotation as very low axial velocities are observed at this piston position.

Using these velocity profiles the swirl ratio or swirl number can be estimated. The swirl number relates the solid body rotation of the fluid within the engine cylinder to the engine revolutions as follows,

\[ R_s = \frac{V_\theta}{2\pi r n} \]  \hspace{1cm} (6.4.1)

where \( V_\theta \) is the tangential velocity, \( r \) is the radius and \( n \) is the engine speed. Considering the tangential velocity profiles shown in figure 6.14, a tangential velocity of 28.3 m/s at a distance of 0.245 m from the cylinder axis gives a swirl number of 8.96. From the experiment described in section 1.5 the estimated swirl number is found to be 8.32 at -10 CAD ATDC, where the CFD model gives 8.96.

### 6.5 Cylinder Momentum

The transient history of angular and axial momentum gives a good indication of how the flow behaves through one engine cycle. The definition of angular momentum is given in equation (5.1.1), while the definition of total volume integrated linear momentum, \( G \) is given as follows [22].

\[ G = \int \int \int (mv_z) \, dV \]  \hspace{1cm} (6.5.1)

The total angular and axial momentum within the engine cylinder over one engine cycle is shown in figure 6.15. The angular momentum has been presented earlier in figure 5.21 but is reprinted for convenience.
As seen in figure 6.15(a) the angular momentum is showing a steady decay due to wall friction from EVC until EVO. A rapid decrease in angular momentum during EVO as flow is directed upwards into the exhaust duct, until IPO and fresh air charge is blown into the cylinder increasing the angular momentum very rapidly until it reaches a maximum at BDC. As the mass flow rate from the scavenging port is decreasing, the strength of the swirling motion is also decreasing with a small increase between the BDC and IPC due to an increase in the mass flow rate at that time. The angular momentum is rapidly decreasing until the decrease is stabilised as the exhaust valve has began to close. In between IPC and EVC at around 60 CAD ABDC a small bump in the curve is detected. This bump is directly proportional to the cylinder mass shown in figure 6.17 which also shows the same tendency at 60 CAD ABDC. As the exhaust valve is fully closed the swirling motion is decaying due to surface friction at the cylinder wall.

The axial momentum of cylinder charge is shown in figure 6.15(b). With the piston in TDC the axial momentum is zero and the fluid motion is in the tangential direction. As the piston begins to move downwards the fluid slowly gains momentum. The fluid again looses momentum as the piston velocity is reduced. At EVO the cylinder charge is directed outwards through the exhaust valve and the axial momentum is increasing rapidly. At IPO the cylinder charge again gains momentum due to the fresh air charge from the scavenging port. After BDC the axial momentum is decreasing as the mass flow rate through the scavenging port is decreasing, with a small increase at 15 CAD ABDC. After EVC the axial momentum is fluctuating due to pressure waves present in the cylinder. As the axial momentum is not of the same resolution as the angular momentum it is quite hard to analyse it with the very same details.

As combustion is not simulated the momentum contribution from the fuel spray is not
included in the transient momentum plot. Adding fuel spray momentum to the transient history will have a significant effect on the shape and magnitude of the curves from TDC to EVO

6.6 Mixing Process

As described in section 2.4 the mixing of the burned gas and the fresh air charge entering from the scavenging box is evaluated using a passive scalar. The concentration of the passive scalar for several piston positions given with a CAD ABDC is shown in figure 6.16. The figure shows the whole engine cylinder along with the scavenging port in the lower right corner of the engine cylinder.
Figure 6.16: Mass concentration of the passive scalar given with piston position ABDC.
As seen in figure 6.16 (a) the fresh air charge with concentration 1.0 is blown into the engine cylinder and a fraction of the fresh air charge mixes with the burned gas. In figure 6.16 (b) a larger fraction of the fresh gas has mixed with the burned gas and in the middle of the cylinder a burned gas is still at the piston surface while the fresh charge is blown up the liner wall. In figure 6.16 (c) most of the burned gas has been flushed of the piston surface and the front of the fresh air charge is moving upwards in the cylinder. At the liner wall a small region of mixed gas is trapped due to flow separation. In figure 6.16 (d) the piston has started the compression stroke and the front of the fresh air charge is moving upwards with a relative large mixing zone in front. Under the exhaust valve a pocket of burned gas seems to be trapped due to flow symmetry. In figure 6.16 (e) the scavenging port is almost closed and the mixing zone is significantly reduced compared to 6.16 (d). At the closure of both scavenging port and exhaust valve the volume averaged mass concentration is 98.2%. As seen in figure 6.16 no short-circuiting zone is identified in the scavenging process and the process consists of displacement and mixing. At the end of the scavenging the process consists of mainly displacement and the mixing zone is strongly reduced. The sharp gradient of the scalar could be a result of an under prediction of the mixing process using the RANS equations, which has also been detected in other studies such as the work of Ingvorsen [14].

The mass concentration shown in figure 6.16 can be combined with the total mass of cylinder charge and the mass of fresh charge retained in cylinder shown in figure 5.19, but reprinted and shown in figure 6.17 for convenience.

![Graph of mass concentration over time](image)

**Figure 6.17:** Total mass of cylinder charge along with the retained fresh mass and the total delivered fresh mass.
As seen in figure 6.17 the total cylinder mass is decreasing due to exhaust valve opening until IPO where the fresh cylinder charge begins to fill the cylinder. The delivered mass and the retained fresh air mass are in good agreement until BDC where a fraction of the fresh air enters the exhaust duct mixed with burned gas and the delivered mass becomes larger than the retained mass. This can also be seen in figure 6.16 (d) and (e). At IPC the total mass of delivered air is 1.924 kg where 1.33 kg are retained within the engine cylinder at exhaust valve closure. After EVC the ratio of total mass and retained fresh mass gives the scavenging efficiency.

6.7 Scavenging Parameters

As described in section 2.1, several parameters have been defined in order to characterize the scavenging process. The scavenging efficiency is calculated according to equation (2.1.3) and compared to the scavenging efficiency obtained by the models introduced in section 2.1. From the results of the mixing process shown in figure 6.16 some of the model constants have been modified as no short-circuiting has been detected. For the Maekawa model the constant $K$ has been set equal 1 and $q$ equal 0.1. For the Benson and Brandham model the constant $y$ has been set equal to zero. The scavenging efficiency as a function of the delivery ratio is shown in figure 6.18.

![Figure 6.18](image-url)

**Figure 6.18:** Scavenging efficiency obtained from CFD model compared to theoretical scavenging models.

As figure 6.18 shows, the scavenging efficiency is in good agreement with the Perfect displacement model, both in shape in magnitude. This is also confirmed by the contour plots shown in figure 6.16 which shows a small mixing zone and a clear displacement line.
in the scavenging process. The Benson and Brandham model also shows good agreement to the CFD model for the higher values of the delivery ratio, while for the lower values it is underestimating the scavenging efficiency significantly. The scavenging efficiency predicted by the Maekawa model is similar to the Benson and Brandham model for the lower range of delivery ratio while for the higher delivery ratio it is overestimating the efficiency. The Perfect mixing model gives a much lower value which does not represent the scavenging process of the uniflow engine. The scavenging efficiency predicted by the CFD model is of 98.3%, which is higher than the scavenging efficiency found by Litke et al. [20], where for an inlet angle of 20° the scavenging efficiency was found to be 93.0%.

The retaining efficiency, defining the ratio of retained mass and total delivered mass is at closure of the exhaust valve at 69.3%, and the remaining 30.7% are blown out the exhaust valve under the scavenging process.

The charging efficiency which gives the ratio of retained scavenging air to the reference mass is shown in figure 6.19 as a function of the delivery ratio.

![Figure 6.19: Charging efficiency obtained from CFD model compared to theoretical scavenging models.](image)

As seen in figure 6.19 the charging efficiency for the perfect displacement, Benson and Brandham and the CFD models are in good agreement until the delivery ratio exceeds 0.75 which is at the point where fresh air starts to enter the exhaust duct. The charging efficiency predicted by the CFD model has a maximum value of $\eta_{ch} = 1.06$ at the end of the scavenging process.
6.8 Experimental Comparison

In this section the results from the CFD model are been compared to measurements conducted on a static geometry test rig at the fluid mechanics laboratory in DTU. The test rig consists of scavenging ports, cylinder and exhaust duct without an exhaust valve. The geometry of the test rig is shown in figure 6.20. The angle of the scavenging ports is smaller compared to the CFD model and measures $12^\circ$ from radial direction compared to $20^\circ$ in the CFD model. The length of the scavenging ports is $35.5$ mm, compared to $50$ mm in the CFD model. The Reynolds number of the experiment is $2.0 \times 10^4$, which is also lower compared to the Reynolds number of the CFD model at $2.5 \times 10^6$. The measurements where conducted with the scavenging ports $50\%$ closed at positions of one, two and three diameters downstream the piston surface with a reference position at BDC.

![Diagram](image)

(a) Side view showing measurement positions. (b) Alignment of scavenging ports.

**Figure 6.20:** Schematic view of the experimental test rig.

The velocity profiles from the CFD model are compared to the experimental velocity profiles in three different ways. At the specific positions downstream the piston the velocity profiles are extracted with the piston moving downwards at a position where the piston is blocking $50\%$ of the scavenging port. With the piston moving upwards at a position where the piston is blocking $50\%$ of the scavenging port and the last one is with a fixed piston, blocking $50\%$ of the scavenging port. In the simulations running with a fixed piston the pressure on the inlet boundary is set to a constant value in order to reach a quasi-steady state solution. The pressure on the inlet boundary is the average pressure in the scavenging box over the scavenging process, $3.63$ bar. For the fixed piston the simulations are running until a steady state solution is reached and the mass flow rate through the scavenging ports has reached a constant value. The aim of this comparison is not to compare the experiments with the CFD results expecting a full agreement in the velocity profiles on all positions downstream the piston, but to identify similar flow behaviour under steady state conditions and to evaluate the effect of the transient behaviour of the flow.

The tangential and axial velocity profiles for all three positions are shown in figure 6.21.
Figure 6.21: Comparison of experimental results and CFD results for three positions upstream the piston surface with scavenging port 50% closed.
As seen in figures 6.21 (a), (c) and (e) the predicted tangential velocity is higher in the CFD simulations compared to the experimental measurements. The reason for higher tangential velocities in the CFD model occurs from the larger angle of the scavenging port leading to a stronger tangential velocity. Another indication of the port angle can be seen in the point of maximum tangential velocity, where for the CFD predictions the point of maximum velocity is closer to the liner wall compared to the measured velocity profile. Considering the shape of the tangential profiles in figure 6.21 (a) the reason for the behaviour of the profiles close to the wall is a flow separation occurring at this point on the liner wall, disturbing the flow profile. For the fixed piston and compression curves the gradient in the core of the vortex is in good agreement with the experimental value while the expansion profile shows a steeper gradient. The velocity profiles predicted by the CFD model in figure 6.21 (c) also show similar behaviour as the measured velocity profile, where the shape of the profile is looking more like a solid body rotation compared to the profiles shown for position one. In figure 6.21 (e) the tangential profile is showing a clear solid body rotation for all cases and the fact that the expansion profile is almost identical to the experimental profile is just of coincidence.

The axial velocity profiles seen in figures 6.21 (b), (d) and (f) show large differences in the axial velocity profiles, where the expansion profile deviates significantly from the other profiles. Considering figure 6.21 (b) the size of the vortex core is larger in the CFD model as a result of a larger port angle, while the peak values are very similar. The expansion profile shows a strong downwards directed velocity in the core as a result of the piston movement and the strong stream of scavenging air due to larger pressure difference between scavenging port and cylinder compared to the compression profile. Figure 6.21 (f) shows good agreement with the fixed piston and measurements while the expansion profile is showing a more uniform distribution as the fresh air charge from the scavenging port has not reached this position in the expansion stroke.

A general conclusion from this comparison is that the results from the compression stroke are in good agreement with the results from the fixed piston simulations while the expansion profiles are showing a different behaviour where the flow has not had the time to develop due to short opening of the scavenging ports. A comparison of the fixed piston simulation and the experimental measurements shows similar flow behaviour in all velocity profiles despite the difference in port angle and Reynolds number.

### 6.9 Effect of Scavenging Port Depth

In this section the effect of reducing the scavenging port depth on the swirling flow and the scavenging process in the engine cylinder is studied. The depth of the scavenging port has been reduced by 40% compared to the standard depth of the scavenging ports in the test engine. In dimensions the reduction of 40% gives a reduction from 0.05 m to 0.03 m. Reducing the cylinder liner outer diameter would reduce material cost for the cylinder liner. The CFD model is identical to the CFD model used for studying the previous
results in chapter 6, disregarding the depth of the scavenging port and the results from the simulations can therefore be directly compared to the previous results from chapter 6. It should be noted that changes in engine geometry could lead to a change in heat released from combustion as a change in flow field has an influence on the effectiveness of combustion. The changes in scavenging port geometry are though not considered significant in terms of boundary conditions.

Flow Visualization

The flow through a horizontal (x-y) plane at a height of half the port height for the shortened scavenging port is shown in figure 6.10 for 6 different CAD from IPO to IPC. Velocity vectors are indicating the magnitude and the contour is showing the out of plane component. It should be noted that the vectors and magnitude contour shown on figure 6.10 are vertex data.
6.9 Effect of Scavenging Port Depth

Comparing figure 6.22 to figure 6.10 it can be concluded that reducing the port depth has no significant effect on the flow structures in the scavenging port during the scavenging process. The vortex in figure 6.22 (c) has a higher velocity compared to the similar vortex seen in figure 6.10 (c), while the vortex in the lower left corner of the scavenging port which
can clearly be seen in figure 6.10 (e), is not as strong for the scavenging port with reduced depth shown in figure 6.22 (e).

**Mass of Cylinder Charge**

The total mass of the cylinder charge $M_{Cych}$, the retained fresh mass of the cylinder charge $M_{Retscav}$ and the delivered mass through the scavenging port $M_{Totscav}$ along with the scavenging efficiency $\eta_{sc}$ is compared to the results from the model with full scavenging port depth in table 6.1. All values are given for a piston position where the scavenging port has closed.

**Table 6.1:** Comparing cylinder masses and scavenging efficiency for the standard and the reduced scavenging port depth.

<table>
<thead>
<tr>
<th>Model</th>
<th>$M_{Cych}$ [kg]</th>
<th>$M_{Retscav}$ [kg]</th>
<th>$M_{Totscav}$ [kg]</th>
<th>$\eta_{sc}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Standard port depth</td>
<td>1.357</td>
<td>1.334</td>
<td>1.924</td>
<td>0.983</td>
</tr>
<tr>
<td>Reduced port depth</td>
<td>1.377</td>
<td>1.331</td>
<td>1.927</td>
<td>0.966</td>
</tr>
</tbody>
</table>

As seen in table 6.1 the cylinder masses are very similar for both port depths but the scavenging efficiency is reduced from 98.3% to 96.6% by shortening the scavenging port.

**Angular Momentum**

The angular momentum for the standard port depth along with the angular momentum for the reduced port depth is shown in figure 6.23.
Figure 6.23: Angular momentum of the gas in the cylinder volume for standard and reduced scavenging port depths.

The angular momentum showed in figure 6.23 is only from IPO to TDC as the data from the reduced scavenging port depth simulation was defected for the CAD until IPO and as the main focus is on the difference in momentum over the scavenging process this is accepted. The angular momentum is very similar until BDC is reached where the standard port depth is predicting stronger momentum. In between BDC and IPC the reduced port depth shows stronger angular momentum and this difference of 1.1% is maintained throughout the engine cycle. This small difference for the two scavenging port depths can also be confirmed by the tangential velocity profiles at 20 mm above the piston surface in TDC.
6.10 Effect of Wall Temperatures

A preliminary study of the effect of heat transfer between walls and cylinder gas has been performed. The results from the simulations with wall heat transfer are compared with the results obtained from the adiabatic model used previously in this study. For the model with heat transfer calculations, wall temperatures of constant value throughout the engine cycle are specified on the surfaces. The temperature of the piston surface is set to $673 \, \text{K}$, the cylinder cover is set to $523 \, \text{K}$ and the temperature of the exhaust valve is set to $873 \, \text{K}$. The liner temperature is varying linearly from the lowest part of the liner next to the scavenging port with a temperature of $323 \, \text{K}$ to the top of the liner with a temperature of $523 \, \text{K}$. These temperatures are based on measurements performed on the 4T50MX test engine under full load conditions. The simulations are performed on the 174k mesh, using the MARS scheme and a maximum $C_r$ number of 200. Both simulations are running with variable heat release volume.

The temperature distribution in the cylinder volume for both the adiabatic wall model and the model with constant wall temperatures, with piston at TDC, can be seen in figure 6.25.

Figure 6.24: Tangential velocity profiles at TDC for the standard and the reduced scavenging port depths.

As seen in figure 6.24 the difference in the tangential velocity at TDC is negligible, 0.2 m/s at maximum velocity. The conclusion from this study of the effect of reducing the scavenging port depth is that by reducing the depth by 40% has a small effect on the flow within the engine cylinder. The largest difference found in this study is in the scavenging efficiency where the difference is 1.7%.

6.10 Effect of Wall Temperatures
6.10 Effect of Wall Temperatures

(a) Model with adiabatic walls.

(b) Model with constant wall temperatures.

Figure 6.25: Temperature [K] distribution in the cylinder volume for adiabatic model and model with constant wall temperatures, with piston at TDC.

The temperature in the model with constant wall temperatures is predicting higher temperatures in the middle of the cylinder volume. Lower temperatures are predicted close to the piston surface, cylinder liner and cover due to the wall temperatures which are low compared to fluid temperatures. The higher temperatures of the cylinder gas in the middle of the cylinder are expected to occur during early phases of compression where heat is transferred to the cylinder gas from the walls, which does not occur in the adiabatic case. A radial temperature profile showing the temperature in the middle of the cylinder volume at a height of 65 mm above the piston surface is shown in figure 6.26.
6.10 Effect of Wall Temperatures

Figure 6.26: Radial temperature ($T \text{ [K]}$) profile for adiabatic model and model with constant wall temperatures, with piston at TDC.

Considering the temperature in the middle of the cylinder ($r = 0$) the relative difference is only 1.90%. For the near wall region the temperature in the constant wall temperature model decreases significantly in the boundary layer due to the wall temperatures, while the adiabatic model predicts a nearly constant temperature through the whole profile.

The cylinder pressure ($P$) for the adiabatic and constant temperature models is shown in figure 6.27.

Figure 6.27: Temperature [K] distribution in the cylinder volume for adiabatic model and model with constant wall temperatures, with piston at TDC.
The cylinder pressure throughout the whole engine cycle is very similar for both the adiabatic and constant wall temperature models. Considering figure 6.27 (b) the higher pressure in the adiabatic model at TDC is accepted as no heat is lost in that model compared to the constant wall temperature model where heat of the cylinder gas in the cylinder volume is lost through the walls, resulting in a lower pressure.

The velocity distribution is studied at the same position as the temperature profile shown in figure 6.26, at 65 mm above the piston surface. The tangential velocity ($V_\theta$) is shown in figure 6.28.

As seen in figure 6.28 the difference in the tangential velocity is very small, where the relative difference is at 1.3%. Assuming these temperatures on the domain walls does therefore not seem to have any significant influence on the velocity field in TDC.

As the temperatures used are based on cycle average values and kept constant during the engine cycle, the variations in wall temperatures are neglected, which could have a significant effect on the local temperature distribution.
CHAPTER 7

Discussion and Conclusion

7.1 Discussion

In this section the methods and results obtained during this work will be considered and discussed.

Validation of CFD Model

The study of a solution that is independent of the engine cycle showed that after engine cycle no. 2, that is when the piston has reached the TDC the second time the solutions obtained in the later engine cycles are deviating within 1.0% compared to engine cycle no. 2. This finding is very valuable for simulations performed on the models with increased number of grid points where the CPU time is significantly increased compared to the basic model. Considering the heat transfer calculations, a similar study should be performed where a cycle independent solution is found in terms of heat transfer.

The comparison of spatial discretization scheme showed that the second order schemes are predicting similar solutions whereas the solution predicted by the first order scheme is quite different. The difference in the solution is primarily in the velocity field, where the first order scheme is predicting much lower velocities compared to the second order schemes while other parameters such as mass flow rate through the scavenging port is not as sensitive to the choice of discretization scheme.

A study of the grid resolution showed that the solution obtained in flow field values such as tangential velocity and angular momentum is showing a reasonable convergence towards a final solution but a grid independent solution has not yet been reached. Considering the tangential velocity profiles shown in figure 5.16 the relative difference between the 174k mesh and the 116k mesh is 1.5%, while the relative difference between the 116k mesh and the 57k mesh is 3.7%. Further mesh refinements could be performed in order to reduce
the relative difference and come closer to a converged solution. This would result in an increased computational time.

The study of turbulence models showed that the standard $k - \varepsilon$ model is predicting a turbulent kinetic energy in the cylinder volume which is much higher compared to the $k - \varepsilon$ RNG and the $k - \omega$ SST models. The work of Lendormy et al. [19] who also studied large diesel engines showed that the turbulence quantities are very sensitive to both grid resolution and time step size. When closing the exhaust valve the predicted turbulent kinetic energy in the cylinder volume is very similar to the turbulent kinetic energy predicted by the work of Lendormy et al. [19], indicating a too coarse mesh in the exhaust valve region.

Results

The total mass of delivered air during scavenging process predicted by the CFD model is 1.92 kg while the total mass delivered in the test engine predicted from experiment is 2.069 kg. The relative small difference indicates that the flow of gasses during the scavenging process in the CFD model is sufficient to identify a realistic flow field inside the engine. The in-cylinder pressure predicted by the simulations has been shown to give a reasonable agreement with the measured in-cylinder pressure at the time of scavenging port opening, indicating that the resistance through the exhaust valve is within a reasonable value.

The in-cylinder pressure predicted from simulations is underestimated at the time of exhaust valve closing compared to the measured value of the pressure, which results in an underestimated pressure at TDC. The altering of outlet pressure boundary could compensate for this pressure difference and a more precise modeling of the outlet pressure, including a time varying value could lead to a more reasonable in-cylinder pressure. The prediction of the in-cylinder pressure would also be improved by implementing an equation of state for real gas as the main difference in ideal gas and real gas predictions occurs at high pressures such as at TDC.

The angular momentum predicted in the present work shows a large increase in cylinder volume momentum during the opening of the scavenging port and a maximum value is reached at BDC with full port opening. At the time of exhaust valve closing the dissipation of momentum is quite constant until opening of the exhaust valve where a drop in angular momentum is caused by the upwards directed motion of the fluid. The injection of fuel into the combustion chamber at TDC in the real engine changes the angular momentum curve as it has been predicted in the present work. The entering of fuel under high velocity would result in an increase in angular momentum inside the combustion chamber.

The mixing of cylinder gasses predicted by the RANS based turbulence models has been shown by the work of Ingvorsen [14] to be underestimated compared to concentration measurements. The scavenging efficiency predicted by the CFD model is 98.3% which is higher compared to the work of Litke et al. [20] who showed a scavenging efficiency of 93% for a port angle of 20°, indicating that the RANS equations are underestimating the mixing of the cylinder gasses during the scavenging process.
Comparison to experimental results showed that the solution predicted by the CFD model is showing similar behaviour of the flow inside the engine cylinder as the experimental results. The stronger tangential velocity in the CFD model is a result of the difference in port angles. The model can be compared to the experimental measurements into more details by altering the angle of the scavenging ports in the model in order to match the angle of the experimental setup. Performing such a comparison is important for further validation on the model.

The study of scavenging port depth showed that a reduction in the depth of the scavenging port by 40% had no significant effect on neither the mass flow rate through the engine or the velocity field within the engine cylinder. The scavenging efficiency was slightly reduced due to stronger mixing of the cylinder gasses.
7.2 Conclusion

In this work a numerical investigation on the scavenging process in a large two-stroke marine diesel engine has been performed. The main work of this process has been constructing a CFD model in the commercial software STAR-CD where a full engine cycle was simulated. Implementations of user-subroutines was adopted for various controls of the model geometry and the working fluid properties.

A numerical validation of the CFD model showed a significant difference in solutions between first and second order discretization schemes where the second order schemes are assumed to predict the most reasonable results. It was shown that the model is not highly sensitive to Courant numbers and therefore computational time can be saved by performing the simulations with a relative high Courant number limit. A mesh refinement study showed that the solution predicted by the model is converging towards a final solution.

Comparison of turbulence models showed that that there is a reasonable agreement between the \( k-\varepsilon \) RNG model and the \( k-\omega \) SST model, while the standard \( k-\varepsilon \) model overpredicts the turbulent kinetic energy during exhaust valve opening compared to the other turbulence models.

The results from the CFD model showed that the total mass of air delivered through the engine is predicted by a difference of 7.1% compared to measurements, where the total mass predicted by the CFD model is 1.92 kg. The cylinder pressure is predicted by a difference of 14%, where it was concluded that the difference is caused by the assumed outlet boundary condition and the assumption of ideal gas for the cylinder gas. A study of the angular momentum within the cylinder volume showed a rapid increase in angular momentum at the opening of scavenging ports and again a rapid decrease after closing of the ports. After closing of the exhaust valve the angular momentum is decreasing at a constant rate due to wall friction.

The scavenging efficiency predicted by the CFD model is in good agreement with the isothermal perfect displacement model with a value of 98.3%. It was concluded that the mixing of the cylinder gases is underestimated by the turbulence models leading to a overpredicted scavenging efficiency. The estimated retaining efficiency shows that 69% of the delivered scavenging air is retained within the engine cylinder at the closure of the exhaust valve.

Comparison with experimental results showed that the CFD model is predicting similar flow behaviour as the experimental results. The influence of piston motion on the flow through the engine cylinder revealed the transient nature of the flow where a large difference is observed for the same piston position in the expansion and compression stroke.

7.3 Future Work

In this section several suggestions are mentioned regarding further work on the CFD model.

- Refine cylinder for better resolution in both tangential and axial directions and proceed with the study of solution convergence.
• Study transient pressure loss and discharge coefficient for the scavenging port and
the exhaust valve.

• Move port angle to $12^\circ$, equal to the port angle in the experimental test rig and do
more comparison with the experimental setup including a exhaust valve.

• Study effect of port geometry on the scavenging performance by altering port dimen-
sions and shape.

• A more extensive study on the effect of heat transfer on the flow field and also study
the heat loss from the engine through an engine cycle, estimating the total heat loss.

• Study a real gas equation of state and implement in the CFD model, replacing the
ideal gas law. Using a real gas instead of ideal gas results in a more accurate pressure
predictions at TDC.

• Develop a 1–D model to simulate the engine cycle in order to predict boundary
conditions, making the model a predictive study instead of a hindsight study.

• Model the combustion process with fuel injection and study the change in angular
and axial momentum within the cylinder volume.

• Model full $3D$ geometry of the engine and compare with the $12^\circ$ sector symmetry
assumption used in this study.

7.4 Acknowledgments

I would like to thank Simon Matlok and Stefan Mayer from MAN Diesel & Turbo for
extensive informations and data on the test engine. I also want to thank Kristian Mark
Ingvorsen and Michael Vincent Jensen from the section of fluid mechanics at DTU which
have provided with support and feedbacks during the project work. For computational
resources my thanks go to the Department of Physics at DTU through the Danish Center
for Scientific Computing (DCSC).


Appendices
Appendix A

Coefficients and Constants for the $k - \omega$ SST turbulence model

For the SST model, the coefficients are expressed in the following general form [6]:

$$C_\phi = F_1C_{\phi 1} + (1 - F_1)C_{\phi 2}$$  \hspace{1cm} (A.0.1)

where $C_{\phi 1}$ and $C_{\phi 2}$ are the coefficient sets seen in table A.1 and $F_1$ is as follows

$$F_1 = \tanh(\text{arg}^d_1) \hspace{0.5cm} \text{where} \hspace{0.5cm} \text{arg}^d_1 = \min \left[ \max \left( \sqrt{\frac{k}{\nu}} \cdot \frac{500\nu}{\omega^2} \right), \frac{4pk}{\sigma_{\omega 2}^2 CD_{k\omega} \nu^2} \right]$$  \hspace{1cm} (A.0.2)

and

$$CD_{k\omega} = \max \left( \frac{2\rho k}{\omega_2 \sigma_{\omega 2} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}}, 10^{-20} \right)$$  \hspace{1cm} (A.0.3)

$$\alpha_1 = \frac{\beta_1}{\beta_1^*} - \frac{1}{\sigma_{\omega 1}} \frac{\kappa^2}{\sqrt{\beta_1^*}}$$  \hspace{1cm} (A.0.4)

$$\alpha_2 = \frac{\beta_2}{\beta_2^*} - \frac{1}{\sigma_{\omega 2}} \frac{\kappa^2}{\sqrt{\beta_2^*}}$$  \hspace{1cm} (A.0.5)

$$S_\omega = 2(1 - F_1) \frac{1}{\sigma_{\omega 2}^2} \frac{1}{\partial \omega/\partial x_j} \frac{\partial k}{\partial x_j}$$  \hspace{1cm} (A.0.6)
\[ F_2 = \tanh(\text{arg}^2) \quad \text{where} \quad \text{arg}_2 = \max \left( \frac{2 \sqrt{\kappa}}{0.09 \omega y}, \frac{500v}{y^2 \omega} \right) \] (A.0.7)

**Table A.1:** Coefficients and constants for the \( k - \omega \) SST turbulence model

<table>
<thead>
<tr>
<th>Model</th>
<th>( a_1 )</th>
<th>( \sigma_{k1}^\omega )</th>
<th>( \sigma_{k2}^\omega )</th>
<th>( \sigma_{\omega}^1 )</th>
<th>( \sigma_{\omega}^2 )</th>
<th>( \beta_1 )</th>
<th>( \beta_2 )</th>
<th>( \beta_1^* )</th>
<th>( \beta_2^* )</th>
<th>( \kappa )</th>
</tr>
</thead>
<tbody>
<tr>
<td>k-( \omega ) SST</td>
<td>0.31</td>
<td>1.176</td>
<td>1.0</td>
<td>2.0</td>
<td>1.168</td>
<td>0.075</td>
<td>0.0828</td>
<td>0.09</td>
<td>0.09</td>
<td>0.41</td>
</tr>
</tbody>
</table>
Thermodynamical Properties

The polynomials used for variations in the thermodynamical properties are from the methodology section in STAR-CD manual.

Figure B.1: Molecular viscosity as a function of temperature.
Figure B.2: Specific heat as a function of temperature.

Figure B.3: Thermal conductivity as a function of temperature.
Event file

! Defining the Event which controls the sliding interface between scavenging port and cylinder and the interface between the valve and the cylinder.
! Last modified: 12_03_2011
! Matches to 170krefinedpistlinercover.mdl

memory maxeve 200
Evfile,init,170krefinedpistlinercover.evn
evslide 40
emslide add region 6
esslide add region 7
eoslide 2 0 0 0 0 0 0 0.01
epslide on
evsave 40
! Enabling Scavenging port interface
evst 1 time 0.0
easi 40 enable
evsave 1
evslide 50
emslide add region 11
esslide add region 10
eoslide 2 0 0 0 0 0 0 0.01
epslide on
evsave 50
! Enabling exhaust valve interface
evst 2 time 0.000001
easi 50 enable
evsave 2
close 170krefinedpistlinercovervn
D.1 bcdefw.f

User-subroutine used for defining the velocity of the wall boundary conditions in the CFD model, in this case the velocity of piston and exhaust valve.

C*************************************************************************
SUBROUTINE BCDEFW(U,V,W,TORHF,SCALAR,RESWT,RSTSC)
C Boundary conditions definition for walls
C*************************************************************************
C--------------------------------------------------------------------------*
C STAR VERSION 4.12.033 *
C--------------------------------------------------------------------------*
INCLUDE 'comdb.inc'
COMMON/USR001/INTFLG(100)

DIMENSION SCALAR(50),RSTSC(50)
DIMENSION SCALC(50)
INCLUDE 'usrdat.inc'
EQUIVALENCE( UDAT12(001), ICTID )
EQUIVALENCE( UDAT01(006), ELOG )
EQUIVALENCE( UDAT02(070), X )
EQUIVALENCE( UDAT02(071), Y )
EQUIVALENCE( UDAT02(072), Z )
EQUIVALENCE( UDAT04(002), DENC )
EQUIVALENCE( UDAT04(003), EDC )
D.1 bcdew.f

EQUIVALENCE( UDAT04(005), PRC )
EQUIVALENCE( UDAT04(009), SCALC(01) )
EQUIVALENCE( UDAT04(007), TC )
EQUIVALENCE( UDAT04(008), TEC )
EQUIVALENCE( UDAT04(059), UC )
EQUIVALENCE( UDAT04(060), VC )
EQUIVALENCE( UDAT04(061), WC )
EQUIVALENCE( UDAT04(064), UCL )
EQUIVALENCE( UDAT04(065), VCL )
EQUIVALENCE( UDAT04(066), WCL )

C-----------------------------------------------------------------------
C
C This subroutine enables the user to specify WALL boundary
C conditions for U,V,W,TORHF,SCALAR,RESWT and RSTSC(IS).
C
C ** Parameters to be returned to STAR: U,V,W,TORHF,SCALAR,
C RESWT, RSTSC(IS)
C
C NB U,V and W are in the local coordinate-system of the
C wall boundary. The wall boundary may be adjacent to a solid.
C
C-----------------------------------------------------------------------
C
C If the wall velocity (WU, WV, WW) is the same for all boundary
C faces in a region the velocity needs only to be calculated once
C per time-step.
C
C Sample coding: To calculate WU, WV, WW once per time-step
C
C-----------------------------------------------------------------------
C
C L = connecting rod length (m)
C S = stroke (m)
C a = crank radius (m)
C THETA_START = start angle for calculation (rad)
C OMEGA = rotational speed (rpm)
C WU,WV,WW = velocity components at wall (m/s)
C
COMMON /WALLVL/ WU,WV,WW

INTEGER ITLAST
SAVE ITLAST
LOGICAL LCALC
SAVE LCALC

REAL PI,L,S,a,OMEGA,k,THETA_START,DELTHE,THETA,WU,WV,WW,THETA_deg

DATA ITLAST /0/
DATA LCALC /.TRUE./

IF(ITER.NE.ITLAST) THEN
  LCALC=.TRUE.
  ITLAST=ITER
ENDIF

C To specify the calculation at wall region No 5 (IREG=5)

IF(IREG.EQ.5) THEN
  IF(LCALC) THEN
    WU=0
    WV=0

    PI=3.14159265
    L=2.885
    S=2.2
    a=S/2
    OMEGA=123.
    k=PI*OMEGA/30.
    THETA_START=0.*(PI/180.)+PI
    DELTHE=PI*OMEGA/30.*DT
    THETA=((PI*OMEGA/30)*TIME)+THETA_START

    WW=(-a*SIN(THETA)-((a**2*SIN(THETA)*COS(THETA))/
         (SQRT(L**2-a**2*(SIN(THETA))**2))))*k

    THETA_deg=(THETA-THETA_START)/PI*180.

    PRINT *,’THETA_deg’,THETA_deg,’WW’,WW
 ENDIF
ENDIF

LCALC=.FALSE.

IF(IREG.EQ.5) THEN
D.2 bcdefp.f

To specify the calculation of velocity at the exhaust valve (IREG=25)

IF(IREG.EQ.25) THEN
  IF(LCALC) THEN
    WU_valve=0
    WV_valve=0
    WW_valve = 1.516570965176710E6*TIME**4 + -1.066132895328210E6*TIME**3 + 2.770168130174910E5*TIME**2 + -3.160409755588325E4*TIME + 1.335061418061348E3;
  ENDIF
ENDIF

LCALC=.FALSE.

IF(IREG.EQ.5) THEN
  U=WU
  V=WV
  W=WW
ENDIF

IF(IREG.EQ.25) THEN
  U=WU_valve
  V=WV_valve
  W=WW_valve
ENDIF

RETURN
END

C

D.2 bcdefp.f

User-subroutine used for defining a value on a pressure boundary, in this model the pressure in the scavenging box and in the exhaust diffuser.
SUBROUTINE BCDEFP(UB, VB, WB, PR, TE, ED, T, SCALAR, TURINT, RSU)
C Boundary conditions definition for pressure boundaries
C*************************************************************************

C--------------------------------------------------------------------------*
C STAR-CD VERSION 4.12.033
C--------------------------------------------------------------------------*

INCLUDE 'comdb.inc'
COMMON/USR001/INTFLG(100)
DIMENSION SCALAR(*)
DIMENSION RSU(6)
LOGICAL TURINT
INCLUDE 'usrdat.inc'
DIMENSION SCALC(50)
EQUIVALENCE( UDAT12(001), ICTID )
EQUIVALENCE( UDAT02(002), DEN )
EQUIVALENCE( UDAT04(002), DENC )
EQUIVALENCE( UDAT04(003), EDC )
EQUIVALENCE( UDAT04(005), PRC )
EQUIVALENCE( UDAT04(009), SCALC(01) )
EQUIVALENCE( UDAT04(007), TC )
EQUIVALENCE( UDAT04(008), TEC )
EQUIVALENCE( UDAT04(059), UC )
EQUIVALENCE( UDAT04(060), VC )
EQUIVALENCE( UDAT04(061), WC )
EQUIVALENCE( UDAT04(064), UCL )
EQUIVALENCE( UDAT04(065), VCL )
EQUIVALENCE( UDAT04(066), WCL )
EQUIVALENCE( UDAT02(070), X )
EQUIVALENCE( UDAT02(071), Y )
EQUIVALENCE( UDAT02(072), Z )

This subroutine enables the user to specify the conditions at
PRESSURE boundaries for UB, VB, WB, PR, TE, ED, T, SCALAR.

** Parameters to be returned to STAR-CD: UB, VB, WB, PR, TE, ED, T, SCALAR,
TURINT

Sample coding: To specify values of turbulence quantities, tempe-
ture and scalars at pressure boundary region 2
(which will be active only if fluid is flowing into
the solution domain) and temperature and scalars
at region region 5.

If the UV option is selected in pro-STAR then the
subroutine also requires the specification
of the values of tangential components.

IF(IREG.EQ.2) THEN
UB=0.
VB=0.
WB=0.
TURINT=.TRUE.
TE=0.01
ED=0.005
T=312.
SCALAR(2)=0.1
SCALAR(5)=0.3
ELSE IF(IREG.EQ.5) THEN
TURINT=.FALSE.
T=473.
SCALAR(2)=0.1
SCALAR(5)=0.3
ENDIF

mpress is a fixed max size of the arrays
rpress is the inlet pressure
npress is the actual number of data in rpress

INTEGER mpress, c
PARAMETER(mpress=2049)
DIMENSION rpress(mpress),apress(mpress)
INTEGER npress
COMMON /Xrc_pressX/ rpress , apress
COMMON /Xic_pressX/ npress

REAL spangle, cpress
LOGICAL first
SAVE first

(other declarations
DATA first /.TRUE./

PRINT*,’Pressure !’
IF (first) THEN
    first = .FALSE.
    C PRINT*, 'Pressure !'
    OPEN(97, FILE='ScavReceiverPres.dat')
    i = 0
    DO
        i = i + 1
        C PRINT*, ' i = ', i
        C IF (i.GT.mpress) STOP 'increase mpress!'
        READ(97, *, END=100) apress(i), rpress(i)
    ENDDO

100 CONTINUE
    npress = i - 1
    CLOSE(97)
ENDIF

C VARIABLES FOR ENGINE CYCLE

PI=3.14159265D0
L=2.885D0
S=2.2D0
a=S/2.D0
OMEGA=123.D0
k=PI*OMEGA/30.
THETA_START=PI
DELTHE=PI*OMEGA/30.0D0*dt_old
THETA=((PI*OMEGA/30.0D0)*TIME)+THETA_START
TIMEPCYCLE = 1.0D0/(OMEGA/60.0D0)
ANGLETIMEF = 360.0D0/TIMEPCYCLE

C USING CONSTANT c TO REGISTER WHICH CYCLE THE PISTON IS IN IN ORDER TO CALCULATE
C THE PRESSURE INDEPENDENT OF THE CYLINDER CYCLE

c = INT(TIME/TIMEPCYCLE)
spangle = ((Time-(cvalve*(TIMEPCYCLE)))*ANGLETIMEF)-180.0
j = -1
DO i=1, npress-1
    IF (apress(i).LT.spangle.AND.apress(i+1).GE.spangle) THEN
j = i
GOTO 200

ENDIF
ENDDO

200 CONTINUE

IF(IREG.EQ.20) THEN

C LINEAR INTERPOLATION IN THE DATAFILE

PR = rpress(i) + (spangle-apress(i))*(rpress(i+1)-rpress(i))/
&(apress(i+1)-apress(i))

C PRINT*, rpress(i) = ',rpress(i)
C PRINT*, rpress(i+1) = ',rpress(i+1)
C PRINT*, PR = ',PR

ENDIF

C-----------------------------------------------------------------------
RETURN
END
C

D.3 newxyz.f

User-subroutine used for moving the mesh in the cylinder volume and exhaust valve by looping over the vertices and move them by a specific distance such as piston motion or exhaust valve closure. The piston motion is simulated by an analytical expression while the exhaust valve motion is an experimentally estimated motion.

C*************************************************************************
SUBROUTINE NEWXYZ(VCORN)
C New X, Y and Z coordinates
C*************************************************************************
C--------------------------------------------------------------------------*
C STAR VERSION 4.12.033 *
C--------------------------------------------------------------------------*
INCLUDE 'comdb.inc'
COMMON/USR001/INTFLG(100)
DIMENSION VCORN(3,NDMAXU)
INCLUDE 'usrdat.inc'
C INCLUDED 'dtu_common.inc'

C-------------------------------------------------------------------------
C
C This subroutine enables the user to specify new Cartesian vertex
coordinates of the mesh x=VCORN(1,IV), y=VCORN(2,IV), z=VCORN(3,IV),
in pro-STAR units, where IV is the vertex number.
C
** Parameters to be returned to STAR: VCORN
C
C-------------------------------------------------------------------------
C
C VERSION: 09_04_2011 MATCHING 174k model
C
C Vertex movement calculation for a reciprocating
piston in a fixed cylinder, where the piston is
activated by a rotating crank mechanism. Also vertex movement
calculation for the valve geometry. Parameters to be set are:
C
C L = connecting rod length                    (m)
C S = stroke                                    (m)
C a = crank radius                              (m)
C THETA_START = start angle for calculation     (rad)
C OMEGA = rotational speed                      (rpm)
C DELTHE = crank angle covered per time step    (rad)
C ZREF = z coordinate above which the mesh remains unchanged (m)
C NDTULOWER = Number of the first vertex to be moved in the piston motion (no)
C NDTUUPPER = Number of the last vertex to be moved in the piston motion (no)
C LOCALMAX = Vertex Z coordinate at the lower edge of the valve (no)
C LOCALMIN = Vertex Z coordinate at the upper edge of the valve (no)
C NDTUVALVELOWER = Number of the first vertex that is moved in the valve mesh (no)
C NDTUVALVEUPPER = Number of the last vertex that is moved in the valve mesh (no)
C VALVEMAX = Gives the higher z coordinate for linear interpolation of mesh points in the valve mesh (m)
C VALVEMIN = Gives the lower z coordinate for linear interpolation of mesh points in the valve mesh (m)
C VALVEMAXLIFT = Maximum displacement of the valve (m)
C LINERHEIGHT = Height of the cylinder liner (m)
C VALVEFULLOPEN = z position of the valve at full opening (m)
C
C DECLARING VARIABLES

INTEGER I,NDTUUPPER, NDTULOWER,NDTUVALVELOWER,NDTUVALVEUPPER, OFFSET
DOUBLE PRECISION PI,L,S,a,OMEGA,ZREF,THETA_START,DELTHE,THETA,x,x_1,DELTA_x
DOUBLE PRECISION TIMEPCYCLE, ANGLETIMEF, VALVEMAXLIFT, LINERHEIGHT
DOUBLE PRECISION ZMIN,DZ,VALVEMAX,VALVEMIN,LOCALMIN,LOCALMAX, VALVEFULLOPEN
DOUBLE PRECISION dt_old

C IMPORTING DATAFILE TO READ THE VALVE POSITION
C mvalve is a fixed max size of the arrays
C rvalve is the valve displacement
C nvalve is the actual number of data in rvalve

INTEGER mvalve, cvalve
PARAMETER(mvalve=2049)
DIMENSION rvalve(mvalve),avalve(mvalve)
INTEGER nvalve
COMMON /mydtsave/ dt_old
COMMON /Xrc_valveX/ rvalve, avalve
COMMON /Xic_valveX/ nvalve

DOUBLE PRECISION sanglevalve1, sanglevalve2, cv
LOGICAL first, Closing
SAVE first, Closing

C (other declarations
DATA first /.TRUE./
DATA Closing /.FALSE./

IF (first) THEN
  first = .FALSE.
  OPEN(76,FILE='ValveLift.dat')
  OPEN(79,FILE='ValvePROFILE.dat')
  i = 0
  DO
    i = i + 1
    IF (i.GT.mvalve) STOP 'increase mvalve'!
    READ(76,*END=100) avalve(i),rvalve(i)
  ENDDO
100      CONTINUE
        nvalve = i - 1
        CLOSE(76)
ENDIF

        PRINT*, 'TIME = ', TIME
        PRINT*, 'DT = ', DT
        PRINT*, 'dt_old = ', dt_old

C VARIABLES FOR THE PISTON MOTION
PI = 3.14159265D0
L = 2.885D0
S = 2.2D0
a = S/2.0D0
OMEGA = 123.0D0
ZREF = 2.248D0
THETA_START = PI
DELTHE = PI*OMEGA/30.0D0*dt_old
THETA = ((PI*OMEGA/30.0D0)*TIME)+THETA_START
TIMEPCYCLE = 1.0D0/(OMEGA/60.0D0)
ANGLETIMEF = 360.0D0/TIMEPCYCLE
NDTULOWER = 200000
NDTUUPPER = 269184
ZMIN = 2312321.0

C VARIABLES FOR THE VALVE MOTION

NDTUVALVEUPPER = 312075.0
NDTUVALVELOWER = 300000.0
VALVEMAX = 2.40513
VALVEMIN = 2.248
LOCALMIN = VCORN(3, 303760)
LOCALMAX = VCORN(3, 302829)
VALVEMAXLIFT = 0.066226D0
LINERHEIGHT = 2.33513D0
VALVEFULLOPEN = 2.2689D0

C CALCULATING THE PISTON POSITION AT CURRENT AND PREVIOUS TIME STEP

x = a*COS(THETA-DELTHE)+SQRT(L**2.0D0-a**2.0D0*(SIN(THETA-DELTHE))**2.0D0)

x_1 = a*COS(THETA)+SQRT(L**2.0D0-a**2.0D0*(SIN(THETA))**2.0D0)
DELTA_x = x_1 - x  
PRINT*, 'DELTA_x = ', DELTA_x  

CC---- LOCATE MIN Z  

C ZMIN = VCORN(3, 266033)  
DO 20 I = NDTULOWER, NDTUUWER  
   IF(VCORN(3, I) .LT. ZMIN) ZMIN = VCORN(3, I)  
   IF(Delta_x .GE. 0 .AND. ZMIN .GE. S) THEN  
      DELTA_x = 0.0D0  
      PRINT*, 'DELTA_x_corrected = ', DELTA_x  
   ENDIF  
20 CONTINUE  

CC---- CALCULATE NEW Z VALUES FOR THE PISTON MOTION  

DO 10 I = NDTULOWER, NDTUUWER  
   C IF(VCORN(3, I) .GE. ZREF) GO TO 10  
   VCORN(3, I) = VCORN(3, I) + DELTA_x * ((ZREF - VCORN(3, I)) / (ZREF - ZMIN))  
10 CONTINUE  

CC---- CALCULATE NEW Z VALUES FOR THE VALVE  

cvalve = INT(TIME / TIMEPCYCLE)  

C THE ANGLE INDEPENDENT OF THE NUMBER OF CYLINDER CYCLES  

sangle = (Time - (cvalve * (TIMEPCYCLE)))) * ANGLETIMEF  
IF(sangle .GT. 0.0D0 .AND. sangle .LT. 180.0DO) THEN  
   sanglevalve1 = sangle  
   sanglevalve2 = sanglevalve1 - (DT * ANGLETIMEF)
ELSEIF(sangle .GT. 180.0D0 .AND. sangle .LT. 360.0D0) THEN
    sanglevalve1 = sangle-360.0D0
    sanglevalve2 = sanglevalve1-(DT*ANGLETIMEF)
ENDIF

CC --------
C INTERPOLATING IN THE DATAFILE FOR THE VALVE CLOSING
j = -1
DO i=1,nvalve-1
   IF (avalve(i).LT.sanglevalve1.AND.avalve(i+1).GE.sanglevalve1) THEN
      j = i
      GOTO 200
   ENDIF
ENDDO
200 CONTINUE

C COMPUTE THE CURRENT VALVE POSITION USING LINEAR INTERPOLATION
   IF(j.GE.1) THEN
      C CALCULATING THE CURRENT POSITION OF THE VALVE FOR THE VALVE CLOSING
      zvalve1 = rvalve(j) + (sanglevalve1-avalve(j))*(rvalve(j+1)
      &-rvalve(j))/(avalve(j+1)-avalve(j))
      PRINT*, 'zvalve1 = ',zvalve1
   ENDIF

k = -1
DO s=1,nvalve-1
   IF (avalve(s).LT.sanglevalve2.AND.avalve(s+1).GE.sanglevalve2) THEN
      k = s
      GOTO 300
   ENDIF
ENDDO
300 CONTINUE
   IF(k.GE.1) THEN
C CALCULATING THE PREVIOUS POSITION OF THE VALVE FOR THE VALVE CLOSING

\[
z_{\text{valve2}} = r_{\text{valve}}(k) + (s_{\text{anglevalve2}}-a_{\text{valve}}(k)) \times (r_{\text{valve}}(k+1) - r_{\text{valve}}(k)) / (a_{\text{valve}}(k+1) - a_{\text{valve}}(k))
\]

PRINT*, 'z_{\text{valve2}} = ', z_{\text{valve2}}

ENDIF

IF(k.EQ.-1) THEN
  DELTAZ=0
  PRINT*, 'DELTAZ1 = ', DELTAZ
ELSEIF (J.LT.1 .AND. s_{\text{anglevalve1}} \geq 70.0D0 .AND. s_{\text{anglevalve1}} < 100.0D0) THEN
  DELTAZ= LINERHEIGHT-LOCALMIN
  PRINT*, 'DELTAZ2 = ', DELTAZ
ELSEIF (z_{\text{valve1}} \equiv z_{\text{valve2}} \equiv VALVEMAXLIFT) THEN
  DELTAZ = VALVEFULLOPEN-LOCALMIN
  PRINT*, 'DELTAZ3 = ', DELTAZ
ELSE
  DELTAZ = z_{\text{valve2}}-z_{\text{valve1}}
  PRINT*, 'DELTAZ4 = ', DELTAZ
ENDIF

PRINT*, 'DELTAZ = ', DELTAZ

C-----

C FOR THE VALVE CLOSING

IF(s_{\text{anglevalve1}} \geq 38.0D0 .AND. s_{\text{anglevalve1}} \lt 90.0D0 .AND. LOCALMIN \lt LINERHEIGHT) DO 30 J=NDTUVALVELOWER,NDTUVALVEUPPER
  PRINT*, 'J = ', J
  IF(VCORN(3,J).LE.LOCALMAX) THEN
    VCORN(3,J)=VCORN(3,J)+DELTAZ*(VCORN(3,J)-VALVEMIN)/(LOCALMAX-VALVEMIN)
  ELSE
    VCORN(3,J)=VCORN(3,J)+DELTAZ*(VALVEMAX-VCORN(3,J))/(VALVEMAX-(LOCALMIN))
  ENDIF
30 CONTINUE

C INTERPOLATING IN THE LOWER PART OF THE VALVE FOR THE VALVE CLOSING

VCORN(3,J)=VCORN(3,J)+DELTAZ*(VCORN(3,J)-VALVEMIN)/(LOCALMAX-VALVEMIN)
ELSE
C INTERPOLATING IN THE UPPER PART OF THE VALVE FOR THE VALVE CLOSING

VCORN(3,J)=VCORN(3,J)+DELTAZ*(VALVEMAX-VCORN(3,J))/(VALVEMAX-(LOCALMIN))
ENDIF
30 CONTINUE
D.4 sorent.f

User-subroutine used for adding energy to the engine to simulate the pressure increase due to combustion. The energy is added to a user defined volume, which in this case is the combustion chamber of the engine.

C*************************************************************************
SUBROUTINE SORENT(S1P,S2P)
This subroutine enables the user to specify a source term (per unit volume) for enthalpy in linearized form:

\[ \text{Source} = S1P - S2P \times T, \quad (W/m^3) \]

in an arbitrary manner.

If temperature is to be fixed to a given value \( T \), then the following may be used:

\[ S1P = \text{GREAT} \times T \]
\[ S2P = \text{GREAT}, \]
where $T$ can be a constant or an arbitrary function of the parameters in the parameter list.

** Parameters to be returned to STAR-CD: **S1P, S2P

---

Modified > 9.3.2011

Moving volume for heat release

$mheat$ is a fixed max size of the arrays

$rheat$ is the heat release

$nheat$ is the actual number of data in $rheat$

Parameters to be set are:

$L$ = connecting rod length (m)

$S$ = stroke (m)

$a$ = crank radius (m)

$THETA\_START$ = start angle for calculation (rad)

$OMEGA$ = rotational speed (rpm)

$DELTHE$ = crank angle covered per time step (rad)

$ZREF$ = z coordinate above which the mesh remains unchanged (m)

$UPPER$ = Upper limit for heat volume (m)

$ABOVEPIST$ = Lower limit above the piston surface for heat volume (m)

$HEATRADIUS$ = RADIUS OF HEAT VOLUME (m)

$VERTEXSTART$ = VALUE OF PISTON POS. AT START (m)

$VERTEXHEAT$ = PISTON POS. AT EACH TIME STEP (m)

Declaring Variables

```fortran
DOUBLE PRECISION :: stime, cheatt, omega, L, S, a, PI, z_calc, VERTEXSTART, VOL
DOUBLE PRECISION :: THETA\_START, DELTHE, THETA, HEATRADIUS, ABOVEPIST
DOUBLE PRECISION :: UPPER, VERTEXSTART
INTEGER mheatt, c
PARAMETER(mheatt=1000000)
DIMENSION rheatt(mheatt), aheatt(mheatt)
INTEGER mheatt
COMMON /Xrc\_heattX/ rheatt, aheatt
COMMON /Xic\_heattX/ mheatt
LOGICAL first
SAVE first
DATA first /.TRUE./
```
IF (first) THEN
  first = .FALSE.
  OPEN(77,FILE='hrrT50fullload.dat')
  i = 0
  DO
    i = i + 1
    IF (i.GT.mheatt) STOP 'increase mheat!'
    READ(77,*,END=100) aheatt(i),rheatt(i)
  ENDDO
  100 CONTINUE
  nheatt = i - 1
  CLOSE(77)
ENDIF

PI=3.14159265D0
L=2.885D0
S=2.2D0
a=S/2.D0
OMEGA=123.D0
THETA_START=PI
DELTHE=PI*OMEGA/30.0D0*dt_old
THETA=((PI*OMEGA/30.0D0)*TIME)+THETA_START
TIMEPCYCLE = 1.0D0/(OMEGA/60.0D0)
ANGLETIMEF = 360.0D0/TIMEPCYCLE
UPPER = 2.305D0
ABOVEPIST = 0.02D0
HEATRADIUS = 0.23D0

C CALCULATING THE PISTON POSITION

VERTEXSTART = 1.785D0
z_calc=a*COS(THETA)+SQRT(L**2.0D0-a**2.0D0*(SIN(THETA))**2.0D0)
VERTEXHEAT = z_calc-VERTEXSTART

C Using constant c to register which cycle the piston is in in order to calculate the heat release independent.
C c=1 indicates that you are in cycle nr.2

c = INT(TIME/TIMEPCYCLE)
spangle = ((Time-(c*(TIMEPCYCLE)))*ANGLETIMEF)-180.0

C CALCULATING THE VOLUME FOR HEAT RELEASE

VOL = (PI/4)*(2*HEATRADIUS)**2*(UPPER-(VERTEXHEAT+ABOVEPIST))

j = -1

DO i=1,nheatt-1
  IF (aheatt(i).LT.spangle.AND.aheatt(i+1).GE.spangle) THEN
    j = i
    GOTO 200
  ENDFI
ENDDO

200 CONTINUE

IF (j.LT.1) THEN
  S1P=0.0
  S2P=0.0
ELSE
  IF (Z.LT.UPPER.AND.Z.GT.(VERTEXHEAT+ABOVEPIST)) THEN
    IF (sqrt((X*X)+(Y*Y)).LT.HEATRADIUS) THEN
      C compute the current heat release using linear interpolation;
      cheatt = rheatt(j) + (spangle-aheatt(j))*(rheatt(j+1)-rheatt(j))/
      & (aheatt(j+1)-aheatt(j))
      S1P=(cheatt/VOL)
      S2P=0.0
    ELSE
      S1P = 0.0
      S2P = 0.0
    ENDIF
  ELSE
    S1P = 0.0
    S2P = 0.0
  ENDIF
ENDIF
D.5 sorsca.f

User-subroutine used for defining a value of a scalar quantity, in this case the passive scalar used for modelling the mass fraction of a cell.

C*************************************************************************
SUBROUTINE SORSCA(S1P,S2P)
C Source-term for scalar species
C*************************************************************************
C--------------------------------------------------------------------------*
C STAR-CD VERSION 4.12.033
C--------------------------------------------------------------------------*
INCLUDE 'comdb.inc'
COMMON/USR001/INTFLG(100)
INCLUDE 'usrdat.inc'
DIMENSION SCALAR(50)
EQUIVALENCE( UDAT12(001), ICTID )
EQUIVALENCE( UDAT03(001), CON )
EQUIVALENCE( UDAT03(002), TAU )
EQUIVALENCE( UDAT03(009), DUDX )
EQUIVALENCE( UDAT03(010), DVDX )
EQUIVALENCE( UDAT03(011), DWDX )
EQUIVALENCE( UDAT03(012), DUDY )
EQUIVALENCE( UDAT03(013), DVDY )
EQUIVALENCE( UDAT03(014), DWDY )
EQUIVALENCE( UDAT03(015), DUDZ )
EQUIVALENCE( UDAT03(016), DVDZ )
EQUIVALENCE( UDAT03(017), DWDZ )
EQUIVALENCE( UDAT03(019), VOLP )
EQUIVALENCE( UDAT04(001), CP )
EQUIVALENCE( UDAT04(002), DEN )
EQUIVALENCE( UDAT04(003), ED )
EQUIVALENCE( UDAT04(004), HP )
EQUIVALENCE( UDAT04(006), P )
EQUIVALENCE( UDAT04(008), TE )
EQUIVALENCE( UDAT04(009), SCALAR(01) )
EQUIVALENCE( UDAT04(059), U )
EQUIVALENCE( UDAT04(060), V )
EQUIVALENCE( UDAT04(061), W )
EQUIVALENCE( UDAT04(062), VISM )
EQUIVALENCE( UDAT04(063), VIST )
EQUIVALENCE( UDAT04(007), T )
EQUIVALENCE( UDAT04(067), X )
EQUIVALENCE( UDAT04(068), Y )
EQUIVALENCE( UDAT04(069), Z )
EQUIVALENCE( UDAT09(001), IS )
common/cdsudp/dsudp

This subroutine enables the user to specify source terms (per unit volume) for species in linearized form:

Source = S1P-S2P*SCALAR(IS), (kg/sm3)

in an arbitrary manner.

If the species is to be fixed to a given value SCI, then the following may be used:

S1P=GREAT*SCI
S2P=GREAT,

where SCI can be a constant or an arbitrary function of the parameters in parameter list.

** Parameters to be returned to STAR-CD: S1P,S2P
dsudp is the partial derivative of mass transfer rate with respect to absolute pressure. It is only used in user cavitation model.

Parameters to be set are:

OMEGA = rotational speed (rpm)
INIT1 = Start CAD for initialization of scalar (CAD)
INIT2 = End CAD for initialization of scalar (CAD)
INIT3 = Start CAD for giving value to scalar (CAD)
INIT4 = END CAD for giving value to scalar (CAD)
DOUBLE PRECISION PI, OMEGA, sangle
INT INIT1, INIT2, INIT3, INIT4, c

C VARIABLES THAT NEED TO BE SET

PI = 3.14159265D0
OMEGA = 123.D0
THETA_START = PI
DELTHE = PI * OMEGA / 30.0D0 * dt_old
THETA = ((PI * OMEGA / 30.0D0) * TIME) + THETA_START
TIMEPCYCLE = 1.0D0 / (OMEGA / 60.0D0)
ANGLETIMEF = 360.0D0 / TIMEPCYCLE
INIT1 = 280.0D0
INIT2 = 282.0D0
INIT1 = 317.0D0
INIT2 = 322.0D0

C Using constant c to register which cycle the piston is in in order to calculate the
C c = 1 indicates that you are in cycle nr. 2

C = INT(TIME / TIMEPCYCLE)

sangle = ((Time - (c * (TIMEPCYCLE))) * ANGLETIMEF)

C -- INITIALIZING THE SCALAR IN THE WHOLE DOMAIN

IF (IS.EQ.1 .AND. sangle .GE. INIT1)
& .AND. sangle .LT. INIT2) THEN
S1P = GREAT * 0.0
S2P = GREAT
C PRINT*, 'TidSOR = ', TIME
C PRINT*, 'S1P = ', S1P
ENDIF

IF (IS.EQ.1 .AND. ICTID.EQ.2 .AND. sangle .GE. INIT3)
& .AND. sangle .LT. INIT4) THEN
S1P = GREAT * 1.0
S2P = GREAT
C PRINT*, 'TidSOR = ', TIME
C PRINT*, 'S1P = ', S1P
ENDIF
D.6 posdat.f

This subroutine gives access to all flow variables at each cell at each time step, giving the opportunity to store the variables in a datafile. The angular momentum and the total mass of cylinder charge and retained fresh mass are calculated and stored in datafiles. Mass averaged volume integrated turbulent kinetic energy is also calculated for the cylinder volume and the result stored in a datafile.

```
USE allmod

C This subroutine enables the user to output data and is called
C at the beginning and at the end of each iteration/time step,
C i.e.
C
C if (level.eq.1) then
C called at the beginning of iteration/time step
C else if (level.eq.2) then
C called at the end of iteration/time step
C end if
C Any user code which is not enclosed in the IF condition will
C be executed for both calls
C
C*************************************************************************
C subroutine posdat(level)
C Post-process data
C*************************************************************************
C--------------------------------------------------------------------------*
C STAR-CD VERSION 4.12.000
C--------------------------------------------------------------------------*
C
C INCLUDE 'dtu_common.inc'
C-------------------------------------------------------------------------
```
Note: 1. File units available to the users for opening their own files are from 84 to 89. Users may write to unit 6 or 60 if they want to see their output on the terminal or the run file.

2. All variables passed to this routine use STAR cell numbering, which is different from pro-STAR cell numbers. pro-STAR cell number can be obtained from a STAR cell number ICSTAR by ICPROSTAR=ICLMAP(ICSTAR)

Sample coding: (a) To write values of U-velocity component, absolute pressure and temperature at 5 specified points to a file at each time step (for plotting).

(b) To calculate average temperature at all walls and in cell next to walls in domain 1.

(c) To calculate and print mass averaged concentration of SC1 at the end of the run into the run file.

Local variables

REAL(ra) :: Angmomcell,mass,masstot,massfresh,massfreshtot
REAL(ra) :: ANGMOM,dummy,total,TurKEtot,TurTDtot,TurKE, TurTD
REAL(ra) :: TurVItot,TurVI
INTEGER :: nd,nset,i,isc,j
LOGICAL first
SAVE first
DATA first /.TRUE./

IF (first) THEN
  first = .FALSE.
  OPEN(91,FILE='Angularmom.dat')
  OPEN(92,FILE='Masscalc.dat')
  OPEN(93,FILE='Turbquant.dat')
  OPEN(94,FILE='Turbvisc.dat')
ENDIF

ANGMOM = 0.
masstot = 0.
massfreshtot = 0.
TurKEtot = 0.
TurTDtot = 0.
TurVItot =0.
isc = 1

C.... SUM ANGULAR MOMENTUM OVER ALL DOMAINS

    do nd=1,doma_no
        if (doma(nd)%mattyp.eq.FLUID) then
            call cset(cs,0,nd,NSD_ALL,INTERNAL)
            do nset=1,cs%no
                do i=cs%ns(nset),cs%ne(nset)
                    IF (sqrt(cx(1,i)**2+cx(2,i)**2).LE.0.25
                        &.AND. cx(3,i).LE.2.33613) THEN
                        mass = dens(i)*vol(i)
                        masstot = masstot + mass
                        massfreshtot = massfreshot + massfresh
                        Angmomcell = cx(1,i)*mass*u(2,i) - cx(2,i)*mass*u(1,i)
                        ANGMOM=ANGMOM+Angmomcell
                        ENDIF
                end do
                do j=cs%ns(nset),cs%ne(nset)
                    IF (sqrt(cx(1,j)**2+cx(2,j)**2).LE.0.25
                        &.AND. cx(3,j).LE.2.33613) THEN
                        mass = dens(j)*vol(j)
                        TurKE = (TE(j)*mass)/masstot
                        TurTD = (ED(j)*mass)/masstot
                        TurKEtot = TurKEtot + TurKE
                        TurTDtot = TurTDtot + TurTD
                        ENDIF
                end do
            end do
        end if
    end do

C print*,'ANGMOM1: ',ANGMOM
C print*,'masstot1: ',masstot
C print*,'massfreshtot1: ',massfreshtot

C SUM UP ACROSS PROCESSORS

    if (parrun) then
        ANGMOM=gsum(ANGMOM)
        masstot=gsum(masstot)
        massfreshtot = gsum(massfreshtot)
        TurKEtot=gsum(TurKEtot)
TurTDtot = gsum(TurTDtot)
TurVItot = gsum(TurVItot)
endif

C print*, 'ANGMOM2: ', ANGMOM
C print*, 'masstot2: ', masstot
C print*, 'massfresh2: ', massfresh

C WRITING TO DATAFILES

WRITE(91,*) TIME, ANGMOM
WRITE(92,*) TIME, masstot, massfreshtot
WRITE(93,*) TIME, TurKEtot, TurTDtot
end
!A script used to decompose the mesh manually.
!Script Matches 57k model

!Choosing cells for the first set

cset news group 2
bset news region 6 7
vset news bset
cset add vset face
bset news region 20
cset add bset

!Writing the first set to the file

setwrite,T50_diffuser_lowerport.set,1

!Choosing cells for the second set

cset news gran 0 0.3 0 13 0 0.68
setread,T50_diffuser_lowerport.set,1,all,dele

!Writing the second set

setwrite,T50_diffuser_lowerport.set,2
!Choosing cells for the third set

cset news gran 0 0.3 0 13 0.6 1.33
setread,T50_diffuser_lowerport.set,1,all,dele
setread,T50_diffuser_lowerport.set,2,all,dele

!Writing the third set

setwrite,T50_diffuser_lowerport.set,3

!Choosing cells for the forth set

cset news gran 0 0.3 0 13 1.25 2.0
setread,T50_diffuser_lowerport.set,1,all,dele
setread,T50_diffuser_lowerport.set,2,all,dele
setread,T50_diffuser_lowerport.set,3,all,dele

!Writing the fourth set

setwrite,T50_diffuser_lowerport.set,4

!Choosing cells for the fifth set

cset news gran 0 0.3 0 13 1.8 2.41
setread,T50_diffuser_lowerport.set,1,all,dele
setread,T50_diffuser_lowerport.set,2,all,dele
setread,T50_diffuser_lowerport.set,3,all,dele
setread,T50_diffuser_lowerport.set,4,all,dele

!Writing the fifth set

setwrite,T50_diffuser_lowerport.set,5

!Choosing cells for the sixth set

cset news gran 0 0.3 0 13 2.2 3.1
bset news region 11 12
setread,T50_diffuser_lowerport.set,1,all,dele
setread,T50_diffuser_lowerport.set,2,all,dele
setread,T50_diffuser_lowerport.set,3,all,dele
setread,T50_diffuser_lowerport.set,4,all,dele
setread,T50_diffuser_lowerport.set,5,all,dele

!Writing the sixth set
setwrite,T50_diffuser_lowerport.set,6

!Choosing cells for the seventh set

cset news gran 0 0.3 0 13 3.0 3.68
setread,T50_diffuser_lowerport.set,3,all,dele
setread,T50_diffuser_lowerport.set,1,all,dele
setread,T50_diffuser_lowerport.set,2,all,dele
setread,T50_diffuser_lowerport.set,4,all,dele
setread,T50_diffuser_lowerport.set,5,all,dele
setread,T50_diffuser_lowerport.set,6,all,dele

!Writing the seventh set
setwrite,T50_diffuser_lowerport.set,7

close T50_diffuser_lowerport.set

!Choosing cells for the eighth set

cset all
setread,T50_diffuser_lowerport.set,3,all,dele
setread,T50_diffuser_lowerport.set,1,all,dele
setread,T50_diffuser_lowerport.set,2,all,dele
setread,T50_diffuser_lowerport.set,4,all,dele
setread,T50_diffuser_lowerport.set,5,all,dele
setread,T50_diffuser_lowerport.set,6,all,dele
setread,T50_diffuser_lowerport.set,7,all,dele

!Writing the eighth set
setwrite,T50_diffuser_lowerport.set,8

close T50_diffuser_lowerport.set
E.1 Mesh Visualizations

Figure E.1: Scavenging box and scavenging port for the 174k mesh.
Figure E.2: Cylinder volume and scavenging port for the 174k mesh.
Figure E.3: Cylinder volume and scavenging port for the 174k mesh.
Figure E.4: Exhaust valve and exhaust diffuser for the 174k mesh.
The first parameter examined for the grid resolution study is the $y^+$ value on several locations in the engine. The piston surface, scavenging port, liner wall and cylinder cover are the locations that are examined as the main focus is on the in-cylinder flow and not on the exhaust valve. The values that are presented in figure F.1 are at a CAD where maximum flow is obtained through the scavenging ports, at 7 CAD after port opening. The $y^+$ values for the in-cylinder and scavenging port locations are shown in figure F.1.
Figure F.1: Dimensionless wall coordinate $y^+$ shown for comparison of the 116k grid and the 174k grid.

As shown in figures F.1 (a)-(d) the $y^+$ values are in good agreement with the refinement considerations made in section 4. The $y^+$ values on the piston wall shown in figure F.1 (a) are the cycle maximum values while the cycle minimum values are at TDC where the values are at maximum $y^+ = 8$ and $y^+ = 34$ for the 174k and the 116k grids, respectively.

Figure F.1 (b) shows the $y^+$ values on the top of the scavenging port where the maximum velocity occurs in the scavenging port. These are maximum values and in other positions in the scavenging port the $y^+$ values are lower. For the 116k grid the $y^+$ values are to high at all regions in the scavenging port while for the 174k grid the $y^+$ values are in the vicinity of the required range for the wall functions.

Figure F.1 (c) shows the $y^+$ values on the cylinder liner. The maximum $y^+$ values on the cylinder liner occur at TDC where the total momentum of the flow is directed in the
tangential direction with the highest velocity close to the cylinder liner. The maximum values at TDC are 229 and 893 for the 174$k$ and the 116$k$ grids, respectively.

In figure F.1 (d) the $y^+$ values on the cylinder cover are shown. At TDC the $y^+$ values have their maximum of 193 and 2940 for the 174$k$ and the 116$k$ grids, respectively. At exhaust valve opening the $y^+$ values are quite lower compared to TDC as a recirculation zone forms due to the geometry of the cover and the velocity in the cells close to the wall is very low.