Numerical Simulation of the Flow in Fuel Nozzles for Two-Stroke diesel Engines

Fredrik Herland Andersen
MEK - FM - EP - 2011- 05
July 2011
Numerical Simulation of the Flow in Fuel Nozzles for Two-Stroke Diesel Engines

MEK-FM-EP-2011-05


This report was prepared by
Fredrik H Andersen

Supervisors
Jens Honore Walther (DTU-MEK)
Knud Erik Meyer (DTU-MEK)
Kristian Mark Ingvorsen (DTU-MEK)
Simon Matlok (MAN)
Stefan Meyer (MAN)

Department of Mechanical Engineering
Section of Fluid Mechanics (FM)
Technical University of Denmark
Nils Koppels Allé
Building 403
DK-2800 Kgs. Lyngby
Denmark
www.mek.dtu.dk
Tel: (+45) 45 25 19 60
Fax: (+45) 45 93 14 75
E-mail: info@mek.dtu.dk

Release date: Date published
Category: 1 (public)
Edition: First
Comments: This report is part of the requirements to achieve the Master of Science in Engineering (M.Sc.Eng.) at the Technical University of Denmark. This report represents 30 ECTS points.
Rights: ©Fredrik Herland Andersen, 2011
Preface

This master thesis is written at the fluid mechanics section at MEK-DTU and in collaboration with MAN Diesel & Turbo in the period of February to July 2011. This aim for this report is to develop a CFD model that is able to model cavitation and then apply this model to a real life fuel injector provided by MAN Diesel & Turbo. As far I know this is the first project where internal nozzle flow and cavitation are investigated numerically both at MEK-DTU and MAN Diesel & Turbo. A second master project that focused building up a cavitation test rig and collecting experimental data was performed at MAN Diesel & Turbo parallel to this study. Initially the idea was that the experimental data obtained for that test rig would be used as benchmark for developing the CFD code used in this study. It was fast discovered that the time lines in the two projects did not coincide since it would take a long time before the test rig would yield data that could be used in this project. The objective was therefore changed to develop a code based on experimental data for cavitation found in the literature and then apply this model to a real life fuel injector provided by MAN Diesel & Turbo.

I would like to thank my supervisor Dr. Jens Honore Walther for great guidance, support and inspiration throughout the project.
Abstract

The fuel injector is an integral component of large two-stroke marine diesel engines as it is responsible for the injection of fuel into the combustion chamber and subsequently influences the combustion process. The fuel injector is responsible for the atomization process for the fuel and the presence of cavitation in the injector nozzles will influence the atomization process. Over the last decades there have been a increasing focus on emission from marine diesel engines and strong regulations have forced the industry to invest in research on how to optimize the fuel consumption and power output. As of today there have not been performed numerical investigation on the internal nozzle flow and cavitation in the fuel atomizers at MAN Diesel & Turbo and this project will serve as a starting point for investigating the presence of cavitating flow in fuel injectors.

A CFD model have been developed for cavitation modeling in this study. The cavitation model in this study is based on a homogenous distribution of bubble seeds present in the fluid that grow and collapse by use of the Rayleigh-Plesset equation. This is the first project performed at both DTU-MEK and MAN Diesel & Turbo, so the project consists of two phases, a development phase where a numerical model is developed and tuned against experimental data from the literature. And a second phase where the numerical model is implemented into the real life F0002 fuel injector under operating conditions. Simulations are performed both for simulations using constant pressure boundaries for inlet and outlet and by use of a transient pressure signal for the inlet. A vortex structure is identified inside the SAC volume and by comparing mass flow rates through the nozzle holes and a vortex shedding frequency is identified. The flow conditions applied for the F0002 injector showed a supercavitating flow regime.
Contents

List of Figures viii
List of Tables xi
Nomenclature 1

1 Introduction 5
  1.1 Background ................................. 5
  1.2 Thesis statement .............................. 6
  1.3 Non-dimensional numbers and flow coefficients .......................... 6

2 Theory 9
  2.1 Diesel fuel injector ............................ 9
  2.2 Cavitation .................................... 10
    2.2.1 Hydrodynamic Cavitation regimes .................. 12
    2.2.2 Vortex Cavitation ............................ 13
  2.3 Bubble dynamics .............................. 14
    2.3.1 Rayleigh Plesset equation ..................... 14
    2.3.2 Bubble equilibrium, growth and collapse ............ 18

3 Numerical model 21
  3.1 Introduction ................................. 21
  3.2 Finite Volume Method ........................... 21
    3.2.1 Governing equations ......................... 22
    3.2.2 Transient term ............................. 23
    3.2.3 Source term ............................... 23
    3.2.4 Convective term ............................. 23
    3.2.5 The Diffusive term ........................... 24
  3.3 Segregated Flow Approach ...................... 25
    3.3.1 SIMPLE Algorithm ............................ 27
  3.4 VOF Multi-Phase Model .......................... 28
3.5 Cavitation model ........................................... 29
3.6 Turbulence modeling ........................................ 32
  3.6.1 k-Epsilon turbulence model ......................... 33
3.7 Boundary Conditions ...................................... 34
3.8 Subroutines added to the CCM+ Solver .................. 36
3.9 Alternative numerical models .............................. 37

4 Development and tuning of the numerical model 39
  4.1 Winklhofer model ........................................ 39
  4.2 Mesh Generation .......................................... 42
  4.3 Model parameters ........................................ 44
    4.3.1 Boundary conditions, discretization schemes, turbu-
    lence specification and initial conditions .................. 44
    4.3.2 Fluid properties .................................... 45
    4.3.3 Cavitation model parameters ........................ 46
    4.3.4 Turbulence .......................................... 52
  4.4 Solution Procedure ...................................... 53
  4.5 Results from the tuned cavitation model ................. 56
  4.6 Investigation of grid dependence ................. 61
  4.7 Discussion for the model development ............... 63

5 MAN Diesel F0002 Fuel Injector 67
  5.1 Real life operating conditions ......................... 67
  5.2 Initial conditions, time step and boundary conditions . 69
  5.3 Mesh ................................................... 71
    5.3.1 Grid independent solution .......................... 74
  5.4 Convergence problems for continuity equation .......... 75

6 Results 81
  6.1 Introduction ............................................. 81
  6.2 Results from the full F0002 geometry without activating the
  Cavitation model ............................................ 82
    6.2.1 Mass flow and flow parameters ...................... 82
    6.2.2 Flow field Visualization ........................... 86
  6.3 Results from the full F0002 geometry including the Cavitation
  model .................................................... 89
  6.4 Results from the Full F0002 geometry without activating the
cavitation model using a transient pressure input boundary
condition .................................................... 94
    6.4.1 Estimation of Cavitation inception .................. 97

7 Discussion and Conclusion 99
  7.1 Discussion ............................................... 99
  7.2 Conclusion and future studies .......................... 101
## List of Figures

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.1</td>
<td>SAC-type Diesel fuel injector (Dam, 2007)</td>
<td>9</td>
</tr>
<tr>
<td>2.2</td>
<td>Principle sketch of sac volume and needle (Martynov, 2005)</td>
<td>10</td>
</tr>
<tr>
<td>2.3</td>
<td>Schematic diagram of phase change for water (Franc, 2006)</td>
<td>10</td>
</tr>
<tr>
<td>2.4</td>
<td>The venturi principle</td>
<td>11</td>
</tr>
<tr>
<td>2.5</td>
<td>Relation between the cavitation number CN and the length of the cavitation region. (Martynov, 2005)</td>
<td>12</td>
</tr>
<tr>
<td>2.6</td>
<td>Sketch of nozzle entrance that show cavitation inception (Martynov, 2005)</td>
<td>13</td>
</tr>
<tr>
<td>2.7</td>
<td>Examples of vortex cavitation</td>
<td>14</td>
</tr>
<tr>
<td>2.8</td>
<td>Spherical bubble in an infinite liquid (Brennen, 1995)</td>
<td>15</td>
</tr>
<tr>
<td>2.9</td>
<td>Part of bubble surface to show force balance (Brennen, 1995)</td>
<td>17</td>
</tr>
<tr>
<td>2.10</td>
<td>Radius of equilibrium of a microbubble as a function of external pressure (Franc, 2006)</td>
<td>19</td>
</tr>
<tr>
<td>3.1</td>
<td>Control volume associated with the node P (Martynov, 2005)</td>
<td>22</td>
</tr>
<tr>
<td>3.2</td>
<td>Example of spatial distribution of bubble seeds in a liquid (User guide, 2010)</td>
<td>29</td>
</tr>
<tr>
<td>3.3</td>
<td>bubble growth rate calculated for an arbitrary pressure series $p_\infty$</td>
<td>31</td>
</tr>
<tr>
<td>3.4</td>
<td>Inlet and outlet shown on the mesh of the MAN Diesel Fuel Nozzle</td>
<td>35</td>
</tr>
<tr>
<td>4.1</td>
<td>View of the transparent “two dimensional” nozzle used for the experiments (Kärrholm, 2007)</td>
<td>40</td>
</tr>
<tr>
<td>4.2</td>
<td>Dimensions of the geometry used in this study</td>
<td>40</td>
</tr>
<tr>
<td>4.3</td>
<td>Massflow and corresponding cavitation regimes (Winklhofer et al. (2001))</td>
<td>42</td>
</tr>
<tr>
<td>4.4</td>
<td>The computational grid used in the validation process</td>
<td>43</td>
</tr>
<tr>
<td>4.5</td>
<td>The $dp = 85$ bar supercavitation regimes of Winklhofer and Kärrholm (Kärrholm, 2007)</td>
<td>49</td>
</tr>
</tbody>
</table>
4.6 Distribution of cavitation using hydroscaled values for seed distribution and radius, $dp = 85$ bar .......................... 49
4.7 Distribution of cavitation and velocity using $R_0 = 0.5\mu m, n_0 = 1.9 \cdot 10^{13} \, 1/m^3$ and $\alpha_0 = 1 \cdot 10^{-5}$, $CN = 5.68$ ................................. 50
4.8 Distribution of cavitation using $R_0 = 0.01061 \mu m, n_0 = 2 \cdot 10^{18} \, 1/m^3$ and $\alpha_0 = 1 \cdot 10^{-5}$, $CN = 5.68$ ................................. 51
4.9 Distribution of cavitation when incorporating the turbulent pressure fluctuations ........................................... 53
4.10 Residual monitors for different solution strategies ................. 54
4.11 Final distribution of cavitation for $dp = 85$ bar, $CN = 5.68$, Supercavitation ................................................. 55
4.12 Mass flow versus pressure difference ................................ 56
4.13 Discharge coefficient and mass flow compared to the Cavitation number ..................................................... 58
4.14 Cavitation regimes for different $CN$ ........................................ 59
4.15 Pressure distribution inside the nozzle ................................. 60
4.16 Cavitation production rates for $dp=85$bar $CN = 5.68$ .......... 61
4.17 The refined Winklhofer grid ............................................ 61
4.18 Cavitation field for refined grid, $dp = 85$ bar, $CN = 5.68$ .... 62
4.19 vector plot of the entire computational domain $dp = 85$, $CN = 5.68$ ................................................................. 63
4.20 Simulation using second order upwind discretization for the segregated flow solver $dp = 85$, $CN = 5.68$ ............ 64
4.21 Simulation using $p_v = 5400 \, Pa$, $dp = 85$, $CN = 5.68$ .... 65

5.1 SAC-type Diesel fuel injector (Dam,2007) ............................ 68
5.2 Placement of boundaries and planes for data collection ........... 70
5.3 Internal volume and preliminary grid for the F0002 geometry 72
5.4 Modifications to the F0002 geometry ................................... 73
5.5 Close up of the SAC volume and the nozzle outlets ............... 73
5.6 Local refinement zones shown by the volumetric control application ................................................................. 74
5.7 Mass flow for several preliminary computational grids ............ 75
5.8 Residuals for the 700K model ............................................ 76
5.9 Spatial distribution of the continuity residual where $r_{\text{continuity}} > 1 \cdot 10^{-4}$ ......................................................... 77
5.10 Residual monitor and mass imbalance scalar view for simulation for 500K cells fitted with expansion tubes ............ 78
5.11 The 700K grid ................................................................. 79

6.1 Mass flow ......................................................................... 83
6.2 Close up of mass flow fluctuations ....................................... 84
6.3 Normalized mass flow for individual nozzles ......................... 84
6.4 Flow coefficients .............................................................. 85
6.5 Spatial pressure distribution in F0002 .................................. 86
6.6 Streamlines showing fluid path through the SAC volume and the nozzles ................................................................. 86
6.7 Streamline and vector field showing fluid path through the SAC volume and the nozzles .................................................. 87
6.8 Placement of plane used to collect vector field ......................... 87
6.9 Transient behavior of the vortex ......................................... 88
6.10 Mass flow through injector .............................................. 90
6.11 Change in mass flow \(\frac{\dot{m}_{\text{cav}}}{\dot{m}_{\text{nocav}}} \cdot 100\) when activating the cavitation model ................................................................. 91
6.12 Volume of Fraction (VOF) .............................................. 91
6.13 Cross section view of nozzle 3 .......................................... 92
6.14 Cavitation production rates \(\frac{m^3}{s}\) ........................................... 93
6.15 Pressure distribution in the SAC volume .............................. 93
6.16 The transient pressure signal used in the simulations ............ 94
6.17 Mass flow through the nozzles .................................. 95
6.18 Flow coefficients for the transient simulation ..................... 96
6.19 Local pressure and placement of probes .......................... 97
## List of Tables

3.1 Model coefficients for the standard k-epsilon model . . . . . . 34
4.1 Dimensions for the geometry used in this study . . . . . . . 40
4.2 Physical properties of Diesel Fuel . . . . . . . . . . . . . 46
4.3 Typical values for the cavitation model (Giannadakis,2005) . 47
4.4 The final parameters for the cavitation model . . . . . . . 51
4.5 Parameters for the cavitation model . . . . . . . . . . . . 62
5.1 Model pressure boundaries for the constant boundary simu-
lations . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 69
5.2 Maximum mass imbalance values for the 700K and 1160K grids 77
6.1 Frequencies and Strouhal numbers for the nozzles . . . . . . . 83
6.2 Frequency and Strouhal number for the vortex structure . . 89
6.3 Frequencies and Strouhal numbers for the nozzles for the sim-
ulation using a transient pressure input boundary . . . . . . . 96
Nomenclature

Abbreviations
CFD  Computational Fluid Dynamics
CFD  Semi Implicit Method for Pressure Linked Equations
VOF  Volume of fluid

Non-Dimensional numbers
CN   Cavitation Number
CN   Cavitation number \( \text{unitless} \)
\( \alpha_v \)  Volume fraction vapor

Greek symbols
\( \rho_v \)  Vapor density \( \frac{kg}{m^3} \)
\( \rho_l \)  Liquid density \( \frac{kg}{m^3} \)
\( \mu \)  Dynamic viscosity \( Pa \cdot s \)
\( \rho \)  Density \( \frac{kg}{m^3} \)
\( \nu \)  Kinematic viscosity \( \frac{m^2}{s} \)
\( \Gamma \)  Diffusivity
\( \nabla \)  Gradient Operator
\( \omega \)  Under relaxation factor
\( \phi \)  Scalar quantity
\( \phi \)  Turbulent kinetic energy
\( \tau \)  Non-dimensional time

**Latin symbols**

\( \vec{a} \)  Area vector

\( f \)  face

\( n_0 \)  Initial seed density \( \frac{1}{m} \)

\( S_{cav} \)  cavitation source term

\( K \)  Constant

\( m' \)  Mass flow correction

\( p_B \)  Bubble Pressure \( Pa \)

\( p_g \)  Gas Pressure \( Pa \)

\( p_v \)  Critical Pressure \( Pa \)

\( p_{inlet} \)  Inlet Pressure \( Pa \)

\( p_{inlet} \)  Pressure at inlet \( Pa \)

\( p_{outlet} \)  Outlet Pressure \( Pa \)

\( p_{outlet} \)  Pressure at outlet \( Pa \)

\( p \)  Pressure \( Pa \)

\( p_v \)  Vapor Pressure \( Pa \)

\( p_\infty \)  Ambient Pressure \( Pa \)

\( p_v \)  Vapor Pressure \( Pa \)

\( R \)  Bubble radius \( m \)

\( R_0 \)  Initial bubble radius \( m \)

\( R_c \)  Critical bubble radius \( m \)

\( \dot{m} \)  Mass flow rate \( \frac{Kg}{s} \)

\( S \)  Surface tension \( \frac{N}{m^2} \)

\( u \)  Velocity \( \frac{m}{s} \)

\( \vec{p}^* \)  Uncorrected pressure \( pa \)

\( \vec{p}' \)  Pressure correction \( Pa \)
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\vec{v}^*$</td>
<td>Uncorrected velocity</td>
<td>$\frac{m}{s}$</td>
</tr>
<tr>
<td>$\vec{v}$</td>
<td>Velocity correction</td>
<td>$\frac{m}{s}$</td>
</tr>
<tr>
<td>$v$</td>
<td>Velocity</td>
<td>$\frac{m}{s}$</td>
</tr>
<tr>
<td>$r$</td>
<td>radius</td>
<td>$m$</td>
</tr>
<tr>
<td>$t$</td>
<td>time</td>
<td>$s$</td>
</tr>
</tbody>
</table>
Chapter 1

Introduction

In the current study a numerical model is developed to model cavitation in fuel nozzles. In this chapter the background for the project is presented along with a declaration of some of the non-dimensional parameters and coefficient used later in the report. Chapter 2 is at theory chapter where some fundamentals of cavitation is presented along with a derivation of the Rayleigh-Plesset equation, which is the equation that the cavitation model in this study is based on. The numerical model used in this study is presented in chapter 3. Chapter 4 is dedicated to the model development where the model parameters and the solution strategy is presented. When the numerical model was developed it was applied to a real life fuel injector, namely the F0002 injector provided by MAN Diesel & Turbo, the development of this model is presented in chapter 5. The results are presented in chapter 6 and a discussion and conclusion is presented in chapter 7.

1.1 Background

Ever since the inception of the internal combustion engine, scientists and engine builders have tried to optimize the combustion process to maximize power outlet and minimize fuel consumption. Over the years as the environmental consciousness have grown there have been a lot of focus on reducing the emission of hazardous gases from the internal combustion engine. The automotive industry have been subjected to emission regulations for several decades and new agreements have already been made for even tighter regulations in the future. This have in turn forced the industry to invest in research towards new technology to comply with the future regulations. Due to high costs and a high level of practical complications, the research
on large two stroke diesel engines lack behind the automotive engines. The shear size of a large two stroke engine makes experimental work difficult, not to mention the cost involved with building, planning and operating full scale test facilities. The market for large two stroke engines is also significantly smaller than for automotive engines so the financial resources available for research is smaller.

There are numerous ways of optimizing the combustion process in large two stroke diesel engines e.g timing of the exhaust valve, massflow through the scavenging ports, various after treatment, optimizing fuel valve and nozzle configurations and injection timing. The scope of this study is to investigate the internal flow in the diesel fuel injector upstream the spray especially investigating the flow at cavitating conditions as this is believed to have great influence on the spray and subsequently the atomization process.

1.2 Thesis statement

The main purpose of this project is to develop and tune a CFD code capable of modeling cavitation and then apply this model to a real life operating condition in a full scale fuel injector called F0002 provided by MAN Diesel & Turbo. This is the first project conducted at MEK-DTU concerning numerical modeling of cavitation so subsequently a lot of time was spent on developing and tuning the model parameters for the cavitation model and obtaining a solution strategy. When the cavitation model was obtained it was implemented in a full scale injector to give a preliminary estimation of the cavitation regimes present at operating conditions. The purpose of this study is to gain knowledge of the internal flow conditions in a fuel injector and to investigate the presence of cavitation as this is expected to have downstream effects on the fuel spray. Providing a tool that provide better understanding of the flow conditions inside the fuel injectors opens up for possibilities in nozzle design and a better fuel consumption.

This study only concerns with the flow upstream of the nozzle outlet and there will be no coupling of the flow fields observed in this study and the subsequent spray.

1.3 Non-dimensional numbers and flow coefficients

Non-dimensional numbers and discharge coefficients are used in this report to characterize flow regimes at different operating conditions and to standardize the output when post processing results from simulations.

The Reynolds number gives a measure of the ratio of inertial forces to viscous forces and is a key parameter when characterizing flow regimes as
1.3 Non-dimensional numbers and flow coefficients

the ratio of inertial to viscous forces denotes the degree of turbulence in the flow. The Reynolds number is written as

\[ Re = \frac{\rho U_b D}{\mu} \]  

(1.1)

Where \( \rho \) is the density of the fluid, \( U_b \) is the mean flow velocity, \( \mu \) is the dynamic viscosity of the fluid and \( D \) is the characteristic length for the flow equal to the diameter for duct flows. If nothing else is stated the bulk velocity is estimated by a theoretical Bernoulli type velocity scale \( U_b \) defined like

\[ U_b = \sqrt{\frac{\rho}{2} \Delta p} \]

(1.2)

where \( \Delta p \) is the driving pressure difference for the flow. All simulation in this study is pressure driven so \( \Delta p = p_{\text{inlet}} - p_{\text{outlet}} \).

Vortex structures are likely to occur in internal flow for fuel injectors and subsequently a shedding frequency is likely to be detected. To normalize the frequencies the Strouhal number is applied. The Strouhal number is a dimensionless number used to describe the oscillating mechanism of vortex shedding and is given by the following expression

\[ St = \frac{fL}{U_b} \]

(1.3)

Where \( f \) is the vortex shedding frequency and \( U_b \) and \( L \) is the velocity and length scale as for the Reynolds number.

Due to the high pressures and subsequent velocities inside fuel injectors the compressibility must be taken into consideration. For this report the Mach number is used to determine if compressible effects should be included in the simulations. The Mach number is the ratio between the free stream velocity \( U_\infty \), and the speed of sound \( c \), for the fluid.

\[ M = \frac{U_\infty}{c} \]

(1.4)

The Mach number is given by equation (1.4) and the rule of thumb is that compressible effects should be included if \( M > 0.3 \).

To evaluate the mass and momentum though the ducts the discharge coefficient and momentum coefficient are applied. The discharge coefficient is the ratio of the actual mass flow though to the theoretical mass flow through a orifice

\[ C_d = \frac{\dot{m}_{\text{actual}}}{\dot{m}_{\text{theoretical}}} = \frac{\dot{m}}{A_0 \rho U_b} \]

(1.5)
where $A_0$ is the original cross sectional area of the orifice. The momentum coefficient is the ratio of momentum flow to the theoretical momentum flow

$$C_m = \frac{\dot{M}_{\text{actual}}}{\dot{M}_{\text{theoretical}}}$$  \hspace{1cm} (1.6)

The actual momentum flow rate is found by extracting an area average velocity $U_{\text{avg}}$ from a plane in the simulation domain and calculating the actual momentum flow rate manually, equation 1.6 is then written

$$C_m = \frac{A_0 \rho U_{\text{avg}}^2}{A_0 \rho U_b^2} = \frac{U_{\text{avg}}^2}{U_b^2}$$  \hspace{1cm} (1.7)

$U_{\text{avg}}$ is calculated from the mass flow like

$$U_{\text{avg}} = \frac{\dot{m}}{A_0 \rho}$$
2.1 Diesel fuel injector

The diesel fuel injector is an integral part of the diesel engine as it injects the fuel into the compressed air in the combustion chamber. It is also responsible for the fuel atomization, which for engines running by the diesel principle has a major influence on the combustion process and directly effects the power outlet, fuel consumption and emissions.

Figure 2.1: SAC-type Diesel fuel injector (Dam, 2007)

Figure 2.1 show a SAC type fuel injector atomizer as the one mounted on MAN Diesel & Turbo's engines. Fuel is delivered to the nozzle from a supply pump with supply pressure at approximately 800 bars in the point marked "Head" on the left hand side of figure 2.1. Further downstream there is a valve that opens at 350 bars giving the fuel a clear path through the sac volume and out the nozzle holes.

Figure 2.2 is a close up to the nozzle and shows the needle, sac volume and nozzle hole. The aim for this project is to investigate the flow field in
this region of the atomizer as it is here cavitation is most likely to occur.

2.2 Cavitation

Cavitation is commonly known as the process of formation of vapor droplets in a liquid created by a sudden drop in the local pressure below the saturation pressure for the liquid. When the local tension $p_v - p$ exceeds the tensile strength of the liquid $p_v - p_{cr}$ the fluid surface rupture and yields a small void which serves as a nuclei for the phase transition process [4]. The liquid will then vaporize in these cavities and bubbles containing vaporized gas will form in the liquid. Other sources of nuclei is non condensable gas, typically air, who to some extent is present in fluids in most practical applications.

Figure 2.3: Schematic diagram of phase change for water (Franc,2006)
2.2 Cavitation

Since the density of vapor phase is assumed much smaller than the liquid phase the amount of heat consumed locally for the evaporation is negligible the process can be assumed isothermal. This is illustrated in figure 2.3 which shows a schematic diagram of the phase change of water. It shows that although cavitation and boiling share the same phase change the physical phenomenon is completely different. For boiling the driving phenomenon is a barotropic change in temperature while cavitation is caused by a isothermal change in pressure.

Cavitation is known to occur in many industrial applications like hydrodynamical systems, turbopumps, on the trailing edge of a propeller and in diesel injection nozzles. Cavitation in all these examples are the result of a sudden change in the velocity due to changes in geometry. This is understood by use of the Bernoulli equation:

\[ P + \frac{1}{2} \rho V^2 = \text{constant} \]  

(2.1)

The Bernoulli equation gives the relation between the static and dynamic pressure. Figure 2.4 shows the a Venturi where the fluid is accelerated through the contraction subsequently yielding a pressure drop as indicated. If the local pressure in the contraction falls below the vapor pressure of the liquid cavitation occurs.

Figure 2.4: The venturi principle
2.2.1 Hydrodynamic Cavitation regimes

The flow conditions described in section 2.2.1 corresponds to hydrodynamic cavitation. Hydrodynamic cavitation is generated when the local pressure decrease is caused by the hydrodynamic motion of the fluid. To describe the nature of the cavitating flow the cavitation number is applied. The cavitation number relates the pressure drop to the local static pressures. There are several definitions of the cavitation number, but for this thesis the following version is found appropriate [7]

\[ CN = \frac{p_{\text{inlet}} - p_{\text{outlet}}}{p_{\text{outlet}} - p_v} \] (2.2)

Where \( p_{\text{inlet}} \) and \( p_{\text{outlet}} \) is the system pressure at the inlet and outlet respectively and \( p_v \) is the vapor pressure of the fluid, usually \( p_v \) is the same as the saturation pressure for the given temperature. The cavitation number is a dimensionless scalar for cavitation used to indicate the cavitating nature of the flow. The cavitation number is not a independent scalar for cavitation as it is geometry dependent but several experimental works have shown a relationship between the cavitation number and the extent of the cavitation region [7],[17].

![Figure 2.5: Relation between the cavitation number CN and the length of the cavitation region. (Martynov,2005)](image)

Figure 2.5 shows a relation between the cavitation number \( CN \) and the average length of the cavitating region \( L_{cav} \). The figure includes the names proposed by Saito and Sato (2001) [15], namely

- Cavitation inception
2.2 Cavitation

- Sub-cavitation
- Transitional cavitation
- Supercavitation

Cavitation inception is when cavitation first occurs in the system. This is typically at the nozzle entrance or in the region immediately adjacent to the vena contracta. Sub-cavitation stage is when the cavitation regions fill the recirculation region located at the entrance of the nozzle. Transitional cavitation is when the cavitation region stretches further downstream the nozzle. When the length of the cavitation region stretches throughout the entire nozzle region the regime is called supercavitation. When the flow experiences supercavitation the flow "choke” and the mass flow becomes independent from any increase in pressure difference this point is called "critical cavitation”.

![Figure 2.6: Sketch of nozzle entrance that show cavitation inception (Martynov,2005)](image)

2.2.2 Vortex Cavitation

Cavitation can also occur in vortex structures in a flow. Due to centrifugal forces the pressure in the core of the vortex is lower than the pressure far away from the core and if \( p_c > p \), cavitation is expected to occur in the center of the vortex flow.

Figure 2.7a shows a known configuration of a three dimensional hydrofoil where the pressure difference between the pressure side and the suction side generates a secondary flow which goes around the tip of the hydrofoil and yields a vortex string attached to the top. If the local pressure drops below the vapor pressure for the liquid cavitation bubbles form. This phenomenon is very common on the trailing edge of ship propellers as shown in figure 2.7b.

Vortex string cavitation is also likely to occur in diesel fuel injectors since vortices is likely to form in the SAC volume of the nozzles.
2.3 Bubble dynamics

2.3.1 Rayleigh Plesset equation

Bubble dynamics is an important aspect of any cavitating flow since the phase change from liquid to gas turns the flow from being single phase the multiphase. Cavitation is usually considered to consist of microscopic nuclei carried by the flow. These nuclei is considered point of weakness for the liquid from which macroscopic cavities are generated and grow in the low pressure regions of the flow. As mentioned in section 2.2 these small voids in the liquid is induced when the local tension exceeds the tensile strength of the liquid, or as impurities of the liquid such as air bubbles. There are several ways to assess the bubble dynamics in a cavitating flow the most prominent being the Rayleigh-Plesset equation, derived by Lord Rayleigh (1842-1919) and further developed by Milton Plesset (1908-1991).

The Rayleigh-Plesset equation assumes that the nuclei’s start out as spherical micro-bubbles of typically a few microns in diameter that contains a gaseous mixture of of vapor for the liquid and possibly some non condensable gas e.g air. Some air is usually present in most liquid, specially if the liquid has been subjected to degassing [9]. Pressure is the main driving parameter for bubble dynamics and growth and collapse is controlled by the pressure difference between the pressure inside the bubble, usually set equal to the vapor pressure, and the ambient pressure. The bubble nuclei is transported with the moving fluid holding the same velocity as it so the pressure difference between the bubble and the local pressure is time dependent for hydrodynamic applications.

Consider a spherical bubble of radius $R(t)$ in an infinite domain of liquid as shown in figure 2.9. $T_\infty$ and $P_\infty(t)$ is the temperature and pressure far away from the bubble. The temperature is assumed to be constant and the
ambient pressure $P_\infty(t)$ is a known input that controls the growth and collapse of the bubble. The collapse of a bubble is known to happen really fast and could produce high local Mach numbers and even shock waves, however for simplicity this derivation assumes the process to be incompressible, which is valid for most of the process except the final stages of collapse. Further it is assumed that the temperature $T_B(t)$ and pressure $P_B(t)$ inside the bubble is constant at all time. The radius of the bubble $R(t)$ is the primary result of the analysis so parameters will be functions of position from the center of the droplet and time. Conservation of mass requires that

$$ u(r,t) = \frac{F(t)}{r^2} \quad (2.3) $$

where $F(t)$ is related to $R(t)$ by a kinematic boundary condition at the bubble surface. For the idealized case of zero mass transport across the bubble boundary, it is clear that $u(R,t) = \frac{dR}{dt}$ and hence

$$ F(t) = R^2 \frac{dR}{dt} \quad (2.4) $$

This is a good approximation even if there is massflow over the boundary [4]. Volume rate of production of vapor is equal to the rate of increase of bubble volume

$$ 4\pi R^2 \frac{dR}{dt} \left[ \frac{m^3}{s} \right] $$

and therefore the mass rate of evaporation.

$$ \rho_v(T_B)4\pi R^2 \frac{dR}{dt} \left[ \frac{kg}{s} \right] $$
This must equal the massflow of liquid inward relative to the interface. Inward velocity is given by
\[
\rho_v(T_B) \frac{dR}{dt} \tag{2.5}
\]
and therefore the velocity can be written
\[
u(R, t) = \frac{dR}{dt} - \frac{\rho_v(T_B) dR}{\rho_L dt} = \left[1 - \frac{\rho_v(T_B)}{\rho_L}\right] \frac{dR}{dt}
\]
and by the relation (2.3) and since \( r \approx R \) (2.5) can be written
\[
F(t) = \left[1 - \frac{\rho_v(T_B)}{\rho_L}\right] R^2 \frac{dR}{dt} \tag{2.6}
\]
In most practical cases the density for the vapor phase is much smaller than the density for the liquid phase and the approximate form of equation (2.3) is adequate. For Newtonian liquids, the Navier-Stokes equations for motion in the \( r \) direction.
\[
-\frac{1}{r} \frac{\partial p}{\partial r} = \frac{\partial u}{\partial r} + u \frac{\partial u}{\partial r} - v_L \left[ \frac{1}{r} \frac{\partial}{\partial r} \left( r^2 \frac{\partial u}{\partial r} \right) - \frac{2u}{r^2} \right] \tag{2.7}
\]
substituting \( u \) according to equation (2.4)
\[
-\frac{1}{r} \frac{\partial p}{\partial r} = \frac{1}{r^2} \frac{dF}{dt} - \frac{2F^2}{r^3} \tag{2.8}
\]
Note that the viscous term the Navier-Stokes equation vanishes. In fact the only viscous contribution in the full Rayleigh-Plesset equation comes from the dynamic boundary condition at the bubble surface. Applying the condition \( p \to p_\infty \) as \( r \to \infty \), equation (2.8) can be integrated to give
\[
p - p_\infty = \frac{1}{r} \frac{dF}{dt} - \frac{1}{2} \frac{F^2}{r^4} \tag{2.9}
\]
A dynamic boundary condition on the bubble surface must be constructed. Considering a small, infinitely thin lamina containing a segment of the interface, see figure 2.9. The net force on this lamina in the radially outward direction per unit area is
\[
(\sigma_{rr})_{r=R} + p_B - \frac{2S}{R} \tag{2.10}
\]
Where \( S \) is the surface tension of at the bubble interface. since
\[
(\sigma_{rr})_{r=R} = -p + 2\mu_L \frac{\partial u}{\partial r}
\]
the net force per unit area is
\[
p_B - (p)_{r=R} = \frac{4\mu_L}{R} \frac{dR}{dt} - \frac{2S}{R} \tag{2.11}
\]
In the absence of mass transport across the boundary this force must be zero, and substituting the value for \((p)_{r=R}\) from equation (2.9) with

\[ F = R^2 \frac{dR}{dt} \]

yields the generalized Rayleigh-Plesset equation first derived by lord Rayleigh (1917), without the surface tension and viscous contribution, and applied to traveling cavitation bubbles by Plesset (1949) [4]

\[ \frac{p_B(t) - p_\infty(t)}{\rho_L} = R \frac{d^2R}{dt^2} + \frac{3}{2} \left( \frac{dR}{dt} \right)^2 + \frac{4\nu_L dR}{R \frac{dt}{dt}} + \frac{2S}{\rho_L R} \]  

(2.12)

Where \(p_B(t)\) in most applications are equal to \(p_{\text{vapor}}\). This equation can then be solved for bubble radius as a function of the local pressure \(p_\infty\). As one can see, the main driving force to this equation is the pressure difference between the local pressure and the liquid vapor pressure. The equation includes effects from surface tension and liquid viscosity, but as these contributions are inversely proportional to the bubble radius they only have significantly influence when the bubble is very small just after growth start and at final stages of collapse. Notice that this model does not include the presence of any non condensable gas in the vapor bubble and the thermal effects are neglected.

When the effects of non condensable gas, surface tension and viscosity is negligible which is the case for large enough bubbles the Rayleigh-Plesset equation reduces to the simple Rayleigh equation

\[ R \frac{d^2R}{dt^2} + \frac{3}{2} \frac{dR}{dt} = p_{\text{vapor}} - p_\infty \]  

(2.13)
which can be integrated once to give the bubble interface velocity

\[
\left( \frac{dR}{dt} \right)^2 = \frac{2}{3} \frac{p_{\text{vapor}} - p_\infty}{\rho_L} \left[ 1 - \left( \frac{R_0}{R} \right)^3 \right] \tag{2.14}
\]

Equation (2.14) is the simplified Rayleigh-Plesset equation and yields the asymptotic growth rate for bubble also known as the \textit{inertia controlled growth model} [1]

\[
\frac{dR}{dt} \approx \sqrt{\frac{2}{3} \frac{p_{\text{vapor}} - p_\infty}{\rho_L}} \tag{2.15}
\]

The \textit{inertia controlled growth model} is the equation used in the cavitation model in STAR-CCM+ and will be the basis equation for the cavitation modeling in this report.

### 2.3.2 Bubble equilibrium, growth and collapse

After establishing a model for the growth of bubble nuclei it is time to evaluate the equilibrium radius \(R\) of a bubble as a function of the liquid pressure \(p_\infty\). A given nucleus is characterized by a mass of non-condensable gas which is assumed constant at any time during the evolution of the bubble. The following equilibrium can be established

\[
p_{\text{g}} + p_{\text{vapor}} = p_\infty + \frac{2S}{R} \tag{2.16}
\]

The evolution of the bubble nuclei is supposed to be isothermal so by use of the ideal gas law the pressure of the non-condensable gas is inverse proportional to the volume of the bubble

\[
p_{\text{g}} = \frac{K}{R^3} \tag{2.17}
\]

where the constant \(K\) is a characteristic of the considered nucleus. equation (2.16) can be rewritten to

\[
\frac{K}{R^3} + p_v = p_\infty + \frac{2S}{R} \tag{2.18}
\]

For a given value of \(K\) equation (2.18) allows the computation of the equilibrium radius of the nuclei as a function of the external pressure \(p_\infty\).

Figure 2.10 show the equilibrium curve for a vapor bubble. The solid part of the line corresponds to stable equilibrium whereas the dotted line corresponds to when the pressure level is lower than the critical pressure and the bubble grows indefinitely without reaching equilibrium. This means that when the local pressure drops below the critical pressure who is not exactly the same, but slightly lower than the vapor pressure, the nucleus becomes unstable and grow into a macroscopic bubble that travels with the
2.3 Bubble dynamics

Figure 2.10: Radius of equilibrium of a microbubble as a function of external pressure (Franco, 2006)

liquid. As mentioned the critical pressure is slightly different to the vapor pressure, this is due to surface tension and the following correlation can be applied

\[ p_c = p_{vapor} - \frac{4S}{3R_c} \]

\[ R_c = \sqrt{\frac{3p_{vapor} R_0^3}{2S}} \]

where \( R_c \) is the critical radius where the bubble no longer can obtain equilibrium.

If the local pressure \( p_{\infty} \) is lower than the vapor pressure \( p_{vapor} \) the bubble radius decreases (\( R < R_0 \)). When this happens the bubble collapses. For simplicity the effect of surface tension, viscosity and presence of non-condensable gas are neglected. The Rayleigh-Plesset equation (2.14) is then written

\[ \frac{dR}{dt} \approx -\sqrt{\frac{2}{3} \frac{p_{\infty} - p_{vapor}}{\rho_L} \left[ \left( \frac{R_0}{R} \right)^3 - 1 \right]} \] (2.19)

Equation (2.19) allows the computation of the collapse time for a bubble i.e. the time necessary for a bubble of initial radius \( R_0 \) to completely
disappear \((R = 0)\). This time is called the rayleigh time and is given by \([9]\)

\[
\tau_{\text{collapse}} \approx 0.915 R_0 \sqrt{\frac{\rho \Delta}{p_{\infty} - p_{\text{vapor}}}}
\]

\((2.20)\)
Numerical model in STAR-CCM+

3.1 Introduction

This study have been carried out by using the commercial CFD software STAR-CCM+ by CD-Adapco. So the numerical approach by default is the Eulerian multi-phase flow with the volume of fluid (VOF) model and the Rayleigh-Plesset equation for modeling the phase change. The equations are solved using the segregated flow solver which solves the flow equations, one for each component of velocity and one for the pressure in a segregated way. The solution algorithm is the SIMPLE algorithm proposed by Spalding and Patankar in 1972 [14]. The simulations where turbulence is included is modeled by use of the standard k-epsilon model. The discretization schemes used in this study is first and second order upwind for the convective terms and the first order temporal discretization scheme for the transient terms.

The mesh was generated by use of the STAR-CCM+ mesh generating capabilities. Where neutral IGES files, prepared in Pro-Engineer, is imported to the STAR-CCM+ 3D-CAD feature where its surfaces is defined and a volume mesh is generated.

3.2 Finite Volume Method

The finite volume method is a method for representing and evaluating the conservations laws [11] where the solution domain is divided up in a finite number of cells on a computational grid. Discrete versions of the integral form of the continuum transport equations are applied to each of the con-
control volumes (cells). The objective is to obtain a linear set of equations corresponding to the number of cells in the computational domain [1]. The grid used in this study is generated using the meshing capabilities in STAR-CCM+. Polyhedral cells with prismatic orthogonal cells in the regions close to wall boundaries make out the body fitted grid for all simulations in this study. All variables are allocated at the centers of the control volumes.

Figure 3.1: Control volume associated with the node P (Martynov,2005)

Figure 3.1 shows the principle layout of a computational cell in the finite volume framework where $P$ is the cell node with neighboring nodes $E$, $W$, $S$, $N$, $L$ and $H$ and cell faces $e$, $w$, $s$, $n$, $l$ and $h$.

### 3.2.1 Governing equations

The transport of a scalar quantity $\phi$ in a continuum is represented by the integral equation

$$\frac{d}{dt} \int_V \rho \chi \phi dV + \oint_A \rho \phi (v - v_g) \cdot da = \int_A \Gamma \nabla \phi \cdot da + \int_V S_\phi dV \quad (3.1)$$

The terms in equation (3.1) is, from left to right, the transient term, the convective flux, the diffusive flux and the volumetric source term. Applying equation (3.1) to a cell centered control volume for cell 0, the following
3.2 Finite Volume Method

The discrete term is obtained

\[
\frac{d}{dt} \left( \rho \chi \phi V \right)_0 + \sum_f [\rho \phi (\mathbf{v} \cdot \mathbf{a} - G)]_f = \sum_f (\Gamma \nabla \phi \cdot \mathbf{a})_f + (S \phi V)_0 \tag{3.2}
\]

Where \( G \) is the grid flux computed from the mesh motion. This study does not make use of a moving mesh, so this term is negligible.

3.2.2 Transient term

Running a simulation transient means that the solution is evolving with time and the transient term must be included in the governing equation. For steady simulations this term is simply neglected. STAR-CCM+ offers several transient solvers, however the transient simulations in this study has been performed by use of the *implicit unsteady solver* with a first order temporal discretization scheme. The fist order temporal scheme is known as Euler implicit and discretizes the unsteady term using the solution at the current time level, \( n + 1 \), as well as the the previous time level \( n \) as follows

\[
\frac{d}{dt} \left( \rho \chi \phi V \right)_0 = \frac{(\rho_0 \phi_0)^{n+1} - (\rho_0 \phi_0)^n}{\Delta t} V_0 \tag{3.3}
\]

3.2.3 Source term

The source term has already been evaluated by equation (3.1) and (3.2), but is repeated here for consistency. By the product of the value of the integrand \( S \phi \), evaluated at the center of a computational cell with a cell volume \( V \) the source term is written

\[
\int_V S \phi dV = (S \phi)_0 \tag{3.4}
\]

3.2.4 Convective term

The convective term at a cell face is discretized as the following expression

\[
[\phi \rho (\mathbf{v} \cdot \mathbf{a} - G)]_f = (\dot{m} \phi)_f = \dot{m}_f \phi_f \tag{3.5}
\]

where \( \phi_f \) and \( \dot{m}_f \) are the scalar value and the mass flow rate at the cell. \( G \) is the grid flux. The discretization scheme for the convective term has a large influence on the numerical stability and accuracy. STAR-CCM+ offers a wide range of schemes, but when using the Raynolds Average Navier-Stokes (RANS) turbulence modeling only the first and second order Upwind differencing schemes are available. Upwind schemes use the flow direction to choose which points should be involved in approximating the convective
Numerical model

term. The idea is that since information is only convected in the direction of the flow, the interpolation scheme should favor the points that are upstream over points that are downstream [11]. For the first order scheme, the convective flux is computed as

$$
(\dot{m}\phi)_f = \begin{cases} 
\dot{m}_f \phi_0 & \text{for } \dot{m}_f \geq 0 \\
\dot{m}_f \phi_1 & \text{for } \dot{m}_f < 0 
\end{cases} \quad (3.6)
$$

This scheme introduces a dissipative error that is stabilizing and helps the solver achieve robust convergence [1]. This scheme has a tendency to smear discontinuities, especially if the discontinuities are not aligned with the grid.

For the second order upwind scheme, the convective flux is computed as

$$
(\dot{m}\phi)_f = \begin{cases} 
\dot{m}_f \phi_{f,0} & \text{for } \dot{m}_f \geq 0 \\
\dot{m}_f \phi_{f,1} & \text{for } \dot{m}_f < 0 
\end{cases} \quad (3.7)
$$

where the face values \(\phi_{f,0}\) and \(\phi_{f,1}\) are linearly interpolated from the cell values on either side of the face as follows

$$
\phi_{f,0} = \phi_0 + s_0 \cdot (\nabla \phi)_{r,0} \quad (3.8)
$$

and

$$
\phi_{f,1} = \phi_1 + s_0 \cdot (\nabla \phi)_{r,1} \quad (3.9)
$$

where

$$
s_0 = x - x_0
$$

$$
s_0 = x - x_1
$$

and \((\nabla \phi)_{r,0}\) and \((\nabla \phi)_{r,1}\) are the reconstruction gradients in cells 0 and 1 respectively. As stated earlier the second order scheme yields a more accurate solution than the first order scheme.

### 3.2.5 The Diffusive term

The diffusive term from equation (3.2) is written on the discrete form as

$$
D_f = \sum_f (\Gamma \nabla \phi \cdot \vec{a})_f \quad (3.10)
$$

where \(\Gamma\) is the face diffusivity, \(\nabla \phi\) is the gradient vector and \(\vec{a}\) is the area vector. the subscript \(f\) refers to a given face.
3.3 Segregated Flow Approach

The VOF multi-phase model with phase change due to cavitation is only available in STAR-CCM+ when selecting the segregated flow model. The segregated flow model solves the flow equations, one for each component of velocity and one for pressure, in a segregated or uncoupled manner. The linkage between the momentum and continuity equations is achieved by a predictor-corrector approach. The model has its roots in constant density flows, but is able to handle mildly compressible flows [1]. However it is not suited to capture shocks and therefore possible shock waves created by bubble collapse can not be resolved by use of the segregated approach. The velocity and pressure fields in this solver is based upon a guessed value and a correction term

\[ \mathbf{v} = \mathbf{v}^* + \mathbf{v}' \]  
\[ p = p^* + p' \]  

Where \( \mathbf{v}^* \) and \( p^* \) are the guessed values for velocity and pressure and \( \mathbf{v}' \) and \( p' \) are the corrections. To obtain convergence, this procedure must result in velocity and pressure fields consistent with the continuity and momentum equation.

\[ \frac{d}{dt} \int_V \rho \chi dV + \oint_A \rho (\mathbf{v} - \mathbf{v}_g) dA = \int_S S_u dV \]  

(3.13)

\[ \frac{d}{dt} \int_V \rho \chi dV + \oint_A \rho \mathbf{v} \otimes (\mathbf{v} - \mathbf{v}_g) dA = -\oint_A p \mathbf{I} dA + \oint_A \mathbf{T} \cdot dA + \oint_A (f_r + f_g + f_p + f_u) dV \]  

(3.14)

The terms on the left hand side of the momentum equation (3.14) are the transient term and the convective flux. The terms on the right hand side are the pressure gradient term, the viscous flux and the body force terms

<table>
<thead>
<tr>
<th>( f_r )</th>
<th>( f_g )</th>
<th>( f_p )</th>
<th>( f_u )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rotational forces</td>
<td>Buoyancy forces</td>
<td>Porous media</td>
<td>User defined</td>
</tr>
</tbody>
</table>

\( \mathbf{T} \) is the viscous stress tensor, for most turbulence models the viscous stress tensor is given by invoking the Bussinesq approximation [1] like

\[ \mathbf{T} = \mathbf{T}_{laminar} + \mathbf{T}_{turbulent} = \mu_{eff} \left[ \nabla \mathbf{v} + \nabla \mathbf{v}^T - \frac{2}{3} (\nabla \cdot \mathbf{v}) \mathbf{I} \right] \]  

(3.15)
where $\mu_{e f f} = \mu_{l a m i n a r} + \mu_{t u r b u l e n t}$ and $I$ is the identity matrix. The momentum equation (3.14), when applied to a cell-centered control volume for cell 0, is written on discrete form

$$
\frac{d}{dt}(\rho \chi v V)_0 + \sum_f [\rho (v - v_g) \cdot a]_f = - \sum_f (p I \cdot a)_f + \sum_f T \cdot a \quad (3.16)
$$

The continuity equation (3.13) is written on discrete form

$$
\sum_f \dot{m}_f = \sum_f (\dot{m}_f^* + \dot{m}_f^\prime) = 0 \quad (3.17)
$$

The uncorrected face mass flow rate $m_f^*$ is computed after the discrete momentum equation (3.16) have been solved. The mass flow correction $m_f^\prime$ is required to satisfy continuity. The uncorrected mass flow rate at an interior face may be written in terms of the cell variables as

$$
m_f^* = \rho \left[ \mathbf{a} \cdot \left( \frac{v_0^* + v_1^*}{2} \right) - G_f \right] - \Upsilon_f \quad (3.18)
$$

where $v_0^*$ and $v_1^*$ are the cell velocities after the discrete momentum equation (3.16) have been solved. $G_f$ is the grid flux and $\Upsilon_f$ is the Rhie-and-Chow type dissipation at the face

$$
\Upsilon_f = Q_f (p_1^* - p_0^* - \nabla \bar{p}_f^* \cdot \mathbf{ds}) \quad (3.19)
$$

where

$$
Q_f = \rho_f \left( \frac{V_0}{\bar{\alpha}_0} + \frac{V_1}{\bar{\alpha}_1} \right) \mathbf{\bar{\alpha}} \cdot \mathbf{a}
$$

is the coefficient of dissipation and $V_0$ and $V_1$ is the volume of cell-1 and cell-2 respectively. $\bar{\alpha}_0$ and $\bar{\alpha}_1$ are the average of the momentum coefficients for all components of momentum. $p_0^*$ and $p_1^*$ are the cell pressures from the previous iteration and $\nabla \bar{p}_f^*$ is the volume weighted average of the cell gradients of pressure $\nabla p_0^*$ and $\nabla p_1^*$. $\mathbf{\bar{\alpha}}$ is the face metric quantity given by equation (??).

If the flow is compressible, the density must also be corrected. Mass flow is then expressed by the following

$$
\dot{m}_f = (\rho + \rho^\prime)_f (v_{f n}^* + v_{f n}^\prime)|\mathbf{a}| = (\rho_f v_{f n}^* + \rho_f v_{f n}^\prime + \rho_f v_{f n}^\prime + \rho_f v_{f n}^\prime)|\mathbf{a}| \quad (3.20)
$$

where $f_n$ denotes the face normal component, further

$$
\rho_f v_{f n}^\prime |\mathbf{a}| \equiv -Q_f (p_1^* - p_0^*) \quad (3.21)
$$
3.3 Segregated Flow Approach

where $p_0'$ and $p_1'$ are the cell pressure correction and

$$\rho_f' \textbf{v}_f^* |_{\text{a}} = \frac{\dot{m}_f^*}{\rho_f} \left( \frac{\partial \rho}{\partial p} \right)_T p_{\text{upwind}}' \tag{3.22}$$

where $p_{\text{upwind}}'$ is found using regular first order upwind interpolation.

$$p_{\text{upwind}}' = \begin{cases} p_0' & \text{for } \dot{m}_f^* > 0 \\ p_1' & \text{for } \dot{m}_f^* < 0 \end{cases} \tag{3.23}$$

The mass flow correction is then found by combining the above equations to the following term

$$\dot{m}_f' = Q_f(p_0' - p_1') + \frac{\dot{m}_f^*}{\rho_f} \left( \frac{\partial \rho}{\partial p} \right)_T p_{\text{upwind}}' \tag{3.24}$$

The discrete pressure correction equation is obtained from equations (3.17) and (3.24). Written in coefficient form it is

$$a_p p_p' + \sum_n a_n p_n' = r \tag{3.25}$$

the residual $r$ is the net mass flow into the cell:

$$r = \sum_f \dot{m}_f^* \tag{3.26}$$

equation (3.26) is the equivalent to the continuity residual and when observing the continuity residual monitor in STAR-CCM+ this is the value that is presented.

3.3.1 SIMPLE Algorithm

The segregated flow model is solved by use of the Semi Implicit Method for Pressure Linked Equations (SIMPLE) algorithm. The SIMPLE algorithm was first introduced by Patankar and Spalding in 1972 [14] and is a widely used procedure to solve the Navier-Stokes Equations. A simplified flowchart for the SIMPLE algorithm, as presented in the user guide [1], is as the following

1. Guess a Pressure field $p^*$
2. Solve the momentum equation (3.16) using the guessed pressure field $p^*$ to obtain a intermediate velocity field $\textbf{v}^*$
3. Calculate the mass imbalances and solve the pressure correction equation (3.25)
4. Update the pressure field:

\[ p_{m+1} = \omega p' \]

where \( \omega \) is the under-relaxation factor for pressure and \( m \) is the iteration step.

5. Correct the face mass fluxes:

\[ \dot{m}_{f}^{m+1} = \dot{m}_{f}^{s} + \dot{m}_{f}' \]

6. Correct the cell velocities with the velocity correction equation

\[ \mathbf{v}_{m+1} = \mathbf{v}^{s} - \frac{V \nabla p'}{a_{p}^{v}} \]

where \( \nabla p' \) is the gradient of the pressure correction and \( a_{p}^{v} \) is the vector of central coefficients for the discretized linear system representing the velocity equation and \( V \) is the cell volume

7. Repeat step 2-6 until convergence.

3.4 VOF Multi-Phase Model

As stated earlier the numerical approach is to use the volume Of Fluid (VOF) multi-phase model. The VOF model is a simple multi-phase model that is well suited to simulate flows that consist of two or more immiscible fluids. The model assumes that all immiscible fluids present in each control volume share the same velocity, pressure and temperature fields. As a result of this assumption, the same set of basic governing equations describing momentum, mass and energy transport in a single-phase flow is solved for an equivalent fluid whose physical properties are calculated as function of its respective phases volume of fraction. The volume of fraction

\[ \alpha_{i} = \frac{V_{i}}{V} \]

is the fraction of the \( i \) th phase compared to the total cell volume. Physical properties in each cell are then calculated as a function of the volume of fraction e.g for density and viscosity

\[ \rho = \sum_{i} \rho_{i} \alpha_{i} \]  \hfill (3.27)

\[ \mu = \sum_{i} \mu_{i} \alpha_{i} \]  \hfill (3.28)
The transport of volume fractions $\alpha_i$ is described by the following conservation equation

$$
\frac{d}{dt} \int_V \alpha_i dV + \int_S \alpha_i (\mathbf{v} - \mathbf{v}_g) \cdot d\mathbf{a} = \int_V S_{\alpha_i} dV
$$

(3.29)

where $S_{\alpha_i}$ is the source of the $i$th phase and $\mathbf{v}_g$ is the grid velocity.[1]

### 3.5 Cavitation model

The cavitation model calculates the phase change between the liquid and its vapor. In STAR-CCM+ the cavitation model is based on a homogenous distribution of bubble seeds throughout the liquid. The spectral distribution of these seeds are approximated by an initial average seed radius $R_0$ and an average seed density $n_0$ [1]. The seed density is a liquid dependent constant and does not change during the simulation. The seed radius $R$ is a time dependent variable of the Rayleigh-Plesset equation (2.12), derived in section 2.3.1.

![Figure 3.2: Example of spatial distribution of bubble seeds in a liquid (User guide, 2010)](image-url)

It is assumed that all vapor bubbles in the computational domain have the same radius $R$ at all time producing a homogenous distribution as displayed in figure 3.2. This assumption yields that the bubble distribution in the liquid can be described by a scalar function, the vapor volume fraction $\alpha_v$. The volume fraction is calculated as a function of the number of bubble seeds and the bubble radius by the following [1]

$$
\alpha_v = \frac{V_v}{V_{tot}} = \frac{V_v}{V_l + V_v} = \frac{n_0 V_l \frac{4}{3} \pi R^3}{V_l + n_0 V_l \frac{4}{3} \pi R^3} = \frac{n_0 \frac{4}{3} \pi R^3}{1 + n_0 \frac{4}{3} \pi R^3}
$$

(3.30)

where $V_{tot}$ is the total control volume, $V_l$ is the volume occupied by the liquid, $V_v$ is the volume occupied by the vapor and $n_0$ is the seed density.
Numerical model

Under the assumption that $\rho_v << \rho_l$ the transport of $\alpha_v$ is calculated by the following transport equation

$$\frac{d}{dt} \int_V \alpha_v dV + \int_S \alpha_v (v - v_g) \cdot da = \int_V S_{cav} dV$$  \hspace{1cm} (3.31)

Where the right hand side of equation (3.31) is the source term for cavitation as in equation (3.29), the source term is given by the following expression

$$S_{cav} = \frac{n_0}{1 + n_0 \frac{4}{3} \pi R^3} \frac{d}{dt} \left( \frac{4}{3} \pi R^3 \right)$$ \hspace{1cm} (3.32)

The source term for cavitation (3.32) includes a time derivative with respect to the bubble radius $\frac{dR}{dt}$, this term is then replaced with the *Inertia controlled growth model* equation 2.15 from section 2.3.1 to include the bubble growth in the transport equation for the volume of fraction (3.31). Replacing the time derivative of $R$ with the Rayleigh equation the source term for cavitation is written

$$S_{cav} = \frac{n_0}{1 + n_0 \frac{4}{3} \pi R^3} \frac{d}{dt} \left( \frac{4}{3} \pi R^3 \right)$$

$$= \frac{n_0}{1 + n_0 \frac{4}{3} \pi R^3} \cdot 4\pi R^2 \cdot (\text{sign}) \sqrt{\frac{2 |P_v - P_\infty|}{\rho_l}}$$ \hspace{1cm} (3.33)

Where the *sign* is used to distinguish between bubble growth and bubble collapse, *sign* is given by the following expression

$$\text{sign} = \frac{p_v - p_\infty}{|p_v - p_\infty| + \text{small}}$$ \hspace{1cm} (3.34)

and *small* is just a very small number (*e.g. small* = $10^{-20}$) to avoid division by zero, in the event of cell pressure being equal to the vapor pressure. To show how the *sign* allows the source term in equation (3.33) to handle both bubble growth and collapse, the growth rate has been calculated with the *inertia controlled growth model* (2.15) for an arbitrary pressure series ranging from $p_\infty = -0.3$ bar to $p_\infty = 0.5$ bar setting $p_v = 5400$ Pa and using $\rho_l = 832$ $[kg/m^3]$. Figure 3.3 shows the growth rate $\frac{dR}{dt}$ for the pressure series. The figure shows that when $p_\infty < p_v$ the bubble will grow and when $p_\infty > p_v$ the bubble collapses.
The cavitation source term, $S_{cav}$, from equation (3.33) is an average value of the current time step and the previous time step. This is because the bubble radius $R$ used in (3.33) is from the previous time step while the time derivative $dR/dt$ corresponds to the current time step. To update the bubble radius to the current time step value the volume fraction $\alpha_v$ in equation (3.30) is solved with respect to bubble radius $R$ for each cell like

$$R = \frac{3}{4} \sqrt[3]{\frac{\alpha_v}{(1 - \alpha_v)n_0 \pi}}$$

(3.35)

Since the source term $S_{cav}$ is an average value of two time steps, the time step used when modeling cavitation have to be very small to minimize modeling errors. This will be shown in section 4.1 where a time steps of $\Delta t = 1 \cdot 10^{-8}$ s was found necessary when modeling cavitation. The transport equation for $\alpha_v$ from equation 3.31 is solved and the density and viscosity are updated to in each cell by the following relation

$$\rho = \alpha_v \rho_v + (1 - \alpha_v) \rho_l$$

(3.36)

and

$$\mu = \alpha_v \mu_v + (1 - \alpha_v) \mu_l$$

(3.37)

The cavitation model presented in this section correspond to the explanation provided by the STAR-CCM+ user guide [1]. When requesting the CD-Adapco support service for a more detailed description of the implementation of the cavitation model they refer to the PhD thesis of Jürgen
Sauer [16] stating that their implementation of the cavitation model is based on his work. Jürgen Sauer describes the cavitation model accordingly to the model presented in this section, but he also states that a modification to the cavitation source term 3.33 must be made to avoid divergence problems in the SIMPLE algorithm. A brief presentation of this modification has been included in appendix A.2.

### 3.6 Turbulence modeling

To account for the effect of turbulence in nozzle flow a Reynolds Averaged Navier-Stokes (RANS) equation based model have been applied. To obtain the RANS equations the instantaneous velocity and pressure components are divided into a mean value and a fluctuating component like

\[
\mathbf{v} = \mathbf{\bar{v}} + \mathbf{v}'
\]

(3.38)

\[
p = \bar{p} + p'
\]

(3.39)

where the averaging process can be considered as time averaging for steady state situations like

\[
\bar{v} = \frac{1}{T} \int_{0}^{T} u \, dt
\]

and the fluctuating component is the root mean squared value. The resulting equations for the mean quantities are essentially the same as the original equations, but an additional term, called the Reynolds stress tensor, is added to the momentum transport equation. The Reynolds stress tensor has the following definition

\[
\mathbf{T}_t = -\rho \mathbf{\bar{v}} \mathbf{\bar{v}}' = -\rho \begin{bmatrix}
 u' u' & u' v' & u' w' \\
 u' v' & v' v' & v' w' \\
 u' w' & v' w' & w' w'
\end{bmatrix}
\]

(3.40)

The challenge is then to model the Reynolds stress tensor \( \mathbf{T}_t \) in terms of the mean flow quantities to yield closure to the governing equations. In this study, a modified version of the standard k-Epsilon model called the standard k-epsilon Two-Layer model is applied. This model falls in the genre of Eddy viscosity models, which makes use of a turbulent viscosity \( \mu_{turbulent} \) to model the Reynolds stress tensor. The most common model, and the one employed by the standard k-epsilon Two-Layer model, is known as the Boussinesq approximation:

\[
\mathbf{T}_t = 2\mu_t \mathbf{S} - \frac{2}{3} (\mu_t \nabla \cdot \mathbf{v} + \rho k) \mathbf{I}
\]

(3.41)
3.6 Turbulence modeling

where $\mathbf{I}$ is the identity matrix, $k$ is the turbulent kinetic energy and $\mathbf{S}$ is the strain tensor given by

$$\mathbf{S} = \frac{1}{2}(\nabla \mathbf{v} + \nabla \mathbf{v}^T) \quad (3.42)$$

3.6.1 k-Epsilon turbulence model

As stated earlier, the standard k-Epsilon model with a Two-layer approach have been selected for this study. The standard k-epsilon model is the most common model for turbulence and it has proven to be a good compromise between accuracy and computational costs. The k-Epsilon model is a two-equation model in which transport equations are solved for the turbulent kinetic energy $k$ and its dissipation rate $\varepsilon$ [1]. The two-layer approach refers to how the viscous sublayer is resolved. The two-layer approach is an alternative to the low- Reynolds number approach that allows the k-epsilon model to be applied in the viscous sublayer. In this approach, the computation is divided into two layers. In the layer close to the wall, the turbulent dissipation rate $\varepsilon$ and the turbulent viscosity $\mu_t$ are specified as functions of wall distance. The values of $\varepsilon$ specified in the near-wall region are blended smoothly with the values computed from solving the transport equation far from the wall. The equation for the turbulent kinetic energy $k$ is solved in the entire flow [1]. A formulation on the two layer approach and wall treatment is found in appendix A

The transport equations for turbulent kinetic energy and the turbulent dissipation rate for the standard k-Epsilon model are:

$$\frac{d}{dt} \int_V \rho k dV + \int_A \rho k (\mathbf{v} - \mathbf{v}_g) \cdot \mathbf{a} = \int_A \left( \mu + \frac{\mu_t}{\sigma_k} \right) \nabla k \cdot \mathbf{a} + \int_V \left[ G_k + G_b - \rho ((\varepsilon - \varepsilon_0) + \Upsilon_M) + S_k \right] dV \quad (3.43)$$

$$\frac{d}{dt} \int_V \rho \varepsilon dV + \int_A \rho \varepsilon (\mathbf{v} - \mathbf{v}_g) \cdot \mathbf{a} = \int_A \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \nabla \varepsilon \cdot \mathbf{a} + \int_V \frac{1}{T} \left[ C_{\varepsilon 1} (G_k + G_{nl} + C_{\varepsilon 3} G_b) - C_{\varepsilon 2} \rho (\varepsilon - \varepsilon_0) \rho \Upsilon_y + S_\varepsilon \right] dV \quad (3.44)$$

Where $S_k$ and $S_\varepsilon$ are the user specified source terms. $\varepsilon_0$ is the ambient turbulence value in source terms that counteract turbulence decay. The
turbulent production $G_k$ is evaluated as

$$G_k = \mu_t S^2 - \frac{2}{3} \rho k \nabla \cdot \mathbf{v} - \frac{2}{3} \mu \left( \nabla \cdot \mathbf{v} \right)^2$$  \hspace{1cm} (3.45)$$

where $\nabla \cdot \mathbf{v}$ is the velocity divergence and $S$ is the modulus of mean strain rate tensor

$$S = |S| = \sqrt{2S : S^T} = \sqrt{2S : S}$$

which is a scalar by the double dot product of two tensors $S : S = S_{ij}S_{ji}$ where the strain rate tensor $S$ is given in equation (3.42). The remaining scalars in equation (3.43) and (3.44) are: $G_{nl}$ that is the non linear turbulent production not applied in this study, turbulent production due to buoyancy $G_b$ neither applied here, nor the Yap correction $\Upsilon_y$. The dilatation dissipation $\Upsilon_M$ is modeled by the following expression

$$\Upsilon_M = \frac{C_{\kappa\varepsilon}}{c^2}$$ \hspace{1cm} (3.46)$$

Where $c$ is the speed of sound and the constant $C_M = 2$. The turbulent viscosity is calculated as

$$\mu_t = \rho C_\mu kT$$ \hspace{1cm} (3.47)$$

where $T$ is the turbulent time scale calculated, without realizible scale option, like

$$T = max \left( \frac{k}{\varepsilon}, C_t \sqrt{\frac{\nu}{\varepsilon}} \right)$$ \hspace{1cm} (3.48)$$

The remaining model coefficients are given in table 3.1

<table>
<thead>
<tr>
<th>$C_{\varepsilon 1}$</th>
<th>$C_{\varepsilon 2}$</th>
<th>$C_\mu$</th>
<th>$\sigma_k$</th>
<th>$\sigma_\varepsilon$</th>
<th>$C_t$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.44</td>
<td>1.92</td>
<td>0.09</td>
<td>1.0</td>
<td>1.3</td>
<td>1.0</td>
</tr>
</tbody>
</table>

The $C_{\varepsilon 3}$ constant is a constant that is set to 1 by default, it is activated if the turbulence model is set to be Buoyancy driven, this is never the case in this study.

3.7 Boundary Conditions

All simulations in this study is based on a flow driven by a pressure difference between a inlet and an outlet. So the boundary conditions set by the user in STAR-CCM+ is the following four:
3.7 Boundary Conditions

- Stagnation Inlet
- Pressure outlet
- Wall, no slip condition
- Symmetry plane

Figure 3.4 shows the grid and geometry of MAN Diesel fuel nozzle and is shown here for illustration purposes. As one can see of the figure the inlet boundary is placed at the left end side of the geometry and the outlets boundaries are placed at the right hand side, one for each of the four outlet nozzles. The rest of the geometry is treated as a wall with no slip condition.

The boundary condition used for the inlet is the stagnation inlet. In short, the stagnation inlet boundary represent the inlet of a duct flow where the stagnation values are known. The boundary face total pressure $p_{tf}$ is specified where the total pressure according to $[1]$ is defined as the static pressure obtained by isotropically bringing the flow to rest

$$p_{tot} = p_{static} \left[ 1 + \frac{(\gamma - 1)}{2} M^2 \right]^\frac{\gamma}{\gamma - 1}$$  \hspace{1cm} (3.49)

Where $M$ is the mach number and $\gamma$ is the specific heat ratio.

The outlet boundary condition used is the Pressure outlet. The static pressure $p_{static}$ are specified by the user. The pressure outlet boundary is primary used for outlet, but can also be used for inlets. When outflow occurs the boundary pressure $p_f$ is assumed to be given by

$$p_f = p_{specified} - \frac{1}{2} \rho_f |v_n|^2$$  \hspace{1cm} (3.50)

where $v_n$ is the normal component of the boundary inflow velocity. This is to discourage backflow from happening so the dynamic head is added to the faces experiencing backflow.
The boundary face velocity is extrapolated from the interior using reconstruction gradients calculated in a similar fashion as equation (3.8) and (3.8).

The wall boundary is as mentioned earlier no-slip condition, meaning that the tangential velocity is explicitly set to zero.

For the validation case Winkhofer nozzle, only a quarter of the geometry is used to save computational time and symmetry planes are used. The shear stress at the boundary is set to zero and the velocity and pressures are extrapolated from the adjacent cell using reconstruction gradients.

3.8 Subroutines added to the CCM+ Solver

In STAR-CCM+ it is possible to include user defined field functions, they may be scalars, vectors, arrays or position types. They are defined in terms of other field functions. For this study two user defined field functions have been incorporated into the code, one to include compressible effects and another one to include turbulent effects on the vapor pressure. When using the Multiphase Mixture material group in CCM+ the phases are defined as Eulerian Multiphase where one can define the phases that are present in the flow, or expected to be present at any time in the flow. Within this framework the equation of state is chosen, in this study the user defined density equation of state was chosen. As mentioned earlier the segregated flow model is only able to handle mild compressible flows so to include compressibility the density for each phase was made a function of the local pressure and the speed of sound in the respective phase.

\[
\rho_{\text{phase}} = \rho_0 + \frac{P_{\infty}}{c^2}
\]  

(3.51)

Where \( c \) is the speed of sound for the given phase under the operating conditions. Equation 3.51 was included as a scalar field function to each cell with the following syntax

\[
832 + \frac{\text{Pressure}}{(1400 \times 1400)}
\]

(3.52)

for the liquid phase and

\[
0.1361 + \frac{\text{Pressure}}{(632 \times 632)}
\]

(3.53)

for the vapor phase.

The other user defined field function used in this study is a modification to the \( p_{\text{vapor}} \) node in the Saturation pressure node in the Eulerian liquid phase. It is believed that the stresses induced by turbulence enhances cavitation and to incorporate these effects these effects should be incorporated
into the liquids vapor pressure. Sergey Martynov suggests the following expression in his Ph.D thesis [14]

\[ P_{\text{critical}} = P_v + 2 \cdot (\mu + \mu_t) \cdot S_{ij}^{\text{max}} \]  

(3.54)

where \( S_{ij}^{\text{max}} \) is the maximum value of the strain rate tensor. As it was not clear how to extract this value as a user field function in CCM+ another model for the critical pressure was applied. The following model is proposed by Sinhal, et al. (2002)[3] and incorporates the turbulent pressure fluctuations in the following way

\[ P_{\text{critical}} = P_v + 0.39 \cdot \rho k \]  

(3.55)

Where \( k \) is the turbulent kinetic energy. Equation 3.55 was implemented as a field function to each cell with the following syntax

\[ 5400 + 0.39 \cdot 0.5 \cdot \rho \cdot \text{Density} \cdot \text{TurbulentKineticEnergy} \]  

(3.56)

### 3.9 Alternative numerical models

The cavitation model in this study is based on a homogenous distribution of vapor bubbles and impurities in the liquid that grow and collapse by use of the Rayleigh-Plesset equation. There are several other cavitation models who will not be elaborated on in this study. However, the development and validation process for the model applied in this project is based on the experimental work of Winklhofer [7] and the numerical work of Kärholm [8]. The numerical work of Kärholm is also based on the work of Winklhofer and was therefore chosen to be used as a benchmark for this project. The numerical model of Kärholm is based on a barotropic equation of state, unlike the model used by STAR-CCM+ that is based on transport of nuclei and the growth and collapse of them. His model determines the degree of cavitation by examining the local density to the saturation values of the liquid and vapor phase.

A very short introduction of Kärholms model will be presented here, but only the bare fundamentals will be included. To see the complete formulation, please refer to [8]. The barotropic equation of state is the non-equilibrium differential equation of state

\[ \frac{D\rho}{Dt} = \psi \frac{Dp}{Dt} \]  

(3.57)

where \( \psi \) is the compressibility, equal to the inverse of the speed of sound squared. The two states is described linearly as

\[ \rho_c = \psi_c p \]
and
\[ \rho_l = \rho_l^0 + \psi_l p \]
The parameter that describes the degree of cavitation, or how the presence of each phase in the mixture is \( \gamma \).
\[ \gamma = \frac{\rho_\infty - \rho_{l, sat}}{\rho_{v, sat} - \rho_{l, sat}} \] (3.58)

Where \( \gamma = 0 \) corresponds to no cavitation and \( \gamma = 1 \) correspond to fully cavitated. The saturation values are calculated from
\[ \rho_{v, sat} = \psi_v p_{sat} \]
These properties together form the mixture's equilibrium equation of state
\[ \rho = (1 - \gamma)\rho_l^0 + (\gamma \psi_v + (1 - \gamma)\psi_l)p_{sat} + \psi(\gamma)(p - p_{sat}) \] (3.59)

The compressibility for the mixture is found as a function of the degree of cavitation as
\[ \psi = \gamma \psi_v + (1 - \gamma)\psi_l \]
These physical properties are included in the continuity and momentum equation and solved using a PISO algorithm. As stated early in this section, the full model will not be described only the fundamental differences to the model applied in this study. It is also assumed that despite the modeling differences, the numerical work of Kärrholm is a viable reference for validation purposes.
Chapter 4

Development and tuning of the numerical model

4.1 Winklhofer model

Consistent and comprehensive flow data for fuel nozzles are difficult to obtain both experimentally and numerically as the physics involved with the cavitation phenomenon still is not fully understood. Experimental data is difficult to obtain since traveling cavitation bubbles hinders the optical access and makes it difficult to investigate the flow inside [7]. As a result of this, many numerical models have only been validated by use of global parameters such as total mass flow, empirical discharge coefficients, cavitation numbers and average velocities [8]. The model development and parameter tuning process of the cavitation model in this study is based on the experimental work of Winklhofer et al. [7] and the numerical cavitation model used by Karrholm et al. [8]. The numerical model used by Kärrholm is briefly described in section 3.9. Winklhofer's nozzle setup was made by a transparent rectangular throttle limiting cavitation to the upper and lower walls thus allowing optical access. Winklhofer, in his paper, refers to this setup as a two dimensional nozzle, this is a modified version of the truth since secondary flow must be present in the flow due the side walls. Since the geometrical change is limited to the upper and lower wall the cavitation formes only here, the cavitation visualization is created by back illumination and image capturing of the flow during stationary flow conditions. The cavitation visualization presented in [7] is based on the average of 20 images. The flow is pressure driven where the inlet pressure is kept at a constant 100 bar and the back pressure at the outlet is varied to obtain the desired pressure drop. A more detailed description of the experimental setup can
be found in [7] and [6].

**Figure 4.1:** View of the transparent "two dimensional" nozzle used for the experiments (Kärrholm, 2007)

<table>
<thead>
<tr>
<th>Dimensions</th>
<th>$R_1[\mu m]$</th>
<th>$H_1[\mu m]$</th>
<th>$H_2[\mu m]$</th>
<th>$W[\mu m]$</th>
<th>$L[\mu m]$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>20</td>
<td>301</td>
<td>284</td>
<td>300</td>
<td>1000</td>
</tr>
</tbody>
</table>

Figure 4.1 shows the transparent nozzle used by winklhofer in his experiments. The geometry was eroded into $300\mu m$ thick steel sheets that is sandwiched between a pair of sapphire windows making it possible to observe the flow through the side windows. The dimensions for the nozzle are displayed in table 4.1.

**Figure 4.2:** Dimensions of the geometry used in this study

Figure 4.2 shows the dimensions for the geometry used in this study. The
width of the geometry, denoted $W$ in table 4.1, is not shown in figure 4.2 since it is a two dimensional plane, but can be understood as the dimension going inwards when looking at the figure. To save computational time only a quarter of the geometry shown in figure 4.2, in this case $90^\circ$ of the geometry, is used in the simulations. Symmetry-plane boundary conditions are used for the side and bottom planes. Winklhofer states that the pressure levels are measured $35\text{mm}$ upstream and downstream of the nozzle, but including the total geometry would yield a large computational domain and subsequently many cells would have to be generated far from the nozzle where the physics of interest is. The inlet pressure boundary is placed $1.5 \cdot L$ upstream the nozzle entrance the outlet pressure boundary is placed $2 \cdot L$ downstream the nozzle outlet. The total height of the geometry is $2 \cdot L$. No slip wall boundary conditions are applied to the walls. The inspiration for the dimensions comes from the PhD thesis of Sergey Martinov (2005) [14] who have performed similar CFD work on the Winklhofer nozzle and on the work by Yuan, et al (2001) [20]. Yuan et al. have done numerical work on a fuel nozzle very similar in geometry to the one of Winklhofer. The dimensions applied in both their work is very similar to the one applied in this study except that neither of them includes the back chamber after the nozzle outlet but enforces the outlet pressure boundary directly onto the nozzle outlet. In this study the outlet pressure boundary is placed further away from the nozzle outlet to avoid that a strong pressure boundary at the nozzle outlet would influence the physics inside the nozzle. Winklhofer’s study includes data on mass flow rates versus pressure drop $\Delta p = p_{\text{inlet}} - p_{\text{outlet}}$ and the corresponding cavitation fields. Mass flow rates and pressure drops corresponding to cavitation inception, transitional cavitation and critical cavitation are presented and will be used for developing the model in this study. Critical cavitation is when the mass flow rate becomes independent of increase of the pressure drop and choked flow occurs.

Figure 4.3 shows the mass flow rate and corresponding pressure drop for the three nozzles Winklhofer investigates in his report. Note that the geometry used in this study correspond to the nozzle called ”Throttle U” in figure 4.3. The cavitation images shown in the figure correspond to critical cavitation, the point where the mass flow chokes and becomes independent of increase in pressure drop. The parameter $CCN$ is the critical cavitation number, a cavitation number calculated for the precise pressure where critical cavitation occurs. This point is very difficult to obtain accurately using a CFD model and will therefore not be used in this report.
4.2 Mesh Generation

The first step was to generate an adequate computational mesh. The mesh was generated from a CAD-file made in the commercial CAD-package Pro-Engineer and then imported as a neutral IGES file into the STAR-CCM+ 3D-CAD environment. The following mesh models were used for the continuum:

- Polyhedral cells
- Prism layer cells
- Surface remesh model

Several meshes of various size and cell numbers were used in the preliminary simulations when developing the model. The final mesh consisted of 140,641 cells with 10 prism layers adjacent to the wall. The prism layer is a set of orthogonal prismatic cells that reside next to wall boundaries in the domain. They are helpful when using turbulence models where one wishes to resolve the viscous boundary layer since they make it easy to put a lot of very small cells close to the wall.

Figure 4.4 shows the computational grid as it was used in the process of developing and tuning the model parameters. As one might notice, there are two regions where grid refinement was applied to the grid. This is
done to insure that there is enough cells in the nozzle region of the geometry without spending a lot of cells to simulate the flow far from the nozzle. In figure 4.4a the grid refinement is shown as the slightly darker areas of the grid. The main refinement zone covers the entire nozzle and the area adjacent to the entrance and exit to avoid interpolation errors between large and small cell surfaces. The cells in this area are scaled to be $\sim 20\%$ of the cell base size of the model. The second refinement area is limited to the nozzle entrance. This refinement is added to make sure the $20^\circ$ rounding of the inlet is properly resolved. The cell size in this region is set to be $\sim 10\%$ of the base size. The cell sizes in the three different regions are approximately $\frac{1}{5} \cdot W$ in the non refined area, $\frac{1}{20} \cdot W$ in the nozzle refinement zone and $\frac{1}{60} \cdot W$ in the nozzle entrance refinement. The cell sizes are measured at the side symmetry plane, front plane in figure 4.4 and does not account for the much smaller prism layer cells close to the wall. A short study of grid dependence have been performed where the number of cells have been doubled and difference in mass flow and the cavitation field is compared to the original grid. Judging from this study it was decided that the cell sizes presented here are adequate to resolve the two phase cavitating flow. Small cells are important especially in the nozzle entry where nucleation and inception of cavitation takes place. Since the information regarding the nozzle dimensions where scarce, development of the mesh have been a
iterative process. The mesh presented in this section is the one used for the validation process and is assumed to contain sufficiently small cells to resolve the bubble dynamics in the flow. Keep in mind that only a quarter of the full geometry is meshed for the simulation so the 140 641 cells correspond to 562 564 cells for the full geometry.

4.3 Model parameters

4.3.1 Boundary conditions, discretization schemes, turbulence specification and initial conditions

During the development and tuning of the model several model parameters, initial conditions and flow specifications where varied before obtaining the final model and solution strategy. A detailed description of the process will be presented in this section, but to make the following sections easier to read an overview of some integral parameters will be given here.

The inlet pressure boundary condition $p_{\text{inlet}}$ is kept constant at 100 bar for every simulation. To obtain the desired pressure drop the outlet pressure boundary $p_{\text{outlet}}$ is adjusted to match the pressure difference. The pressure drop used in the simulations span from $\Delta p = 60 \text{ bar}$ to $\Delta p = 90 \text{ bar}$ in accordance with the pressure drop from figure 4.3. The bottom region and the region facing the reader in figure 4.4 have symmetry plane boundary conditions. The back wall and the top wall have no slip boundary conditions.

The implicit unsteady solver is used for all simulations in this report and the temporal discretization scheme used is the first order Euler implicit scheme shown in equation (4.3.1). Several time steps are used for the simulations, the first time step is $\Delta t = 3 \cdot 10^{-8} \text{ s}$ this is the same time step as Yuan et al [20] used in their simulations. When trying to converge the simulations, as will be discussed later, time steps as low as $\Delta t = 5 \cdot 10^{-9} \text{ s}$ was used and finally the time step of $\Delta t = 1 \cdot 10^{-8} \text{ s}$ was decided to be the time step for cavitation modelling. This time step yielded courant numbers in the order of 0.2 and was considered adequate for the cavitation simulations. As stated in section 3.5 a very low time step is needed since the cavitation source term is an average value of the current and previous time step. the

The spatial discretization scheme for the convective terms used in this study is the second order upwind scheme and the first order upwind scheme. The second order upwind scheme is the default discretization scheme in STAR-CCM+ and the one used when developing the model. However, there where problems converging the simulations and the spatial discretization scheme was changed to a first order upwind for the segregated flow solver. The first and second order upwind convective schemes is described in section 3.2.4.
The convective scheme for the turbulence models and the VOF solver was kept second order for all simulations.

During the development stages for the cavitation model the initial flow field was set stagnant for all simulations, but the initial pressure in the domain was set to $p_{\text{inlet}}$ for all simulations. This was done to prevent back flow in the nozzle during the initial time steps of the simulations and was proven to be a good initial condition for obtaining a steady state solution. It was later discovered that initializing the cavitation simulation from a fully developed flow field obtained by a steady simulation was the best approach as this was time saving and led to better convergence of the equations.

During the initial stages of the model development and tuning turbulence models was not included in the simulations. When tuning of parameters started to yield consistent and good results, the standard k-epsilon Two-Layer model was applied. A formulation of the two layer model and the wall treatment can be found in appendix A.1. The turbulence specification used at both the inlet boundary and the outlet boundary was turbulence intensity $I = 0.01$ and turbulence length scale $l_{\text{turb}} = 0.1 \cdot H_{\text{avg}}$. Where $H_{\text{avg}} = \frac{H_1 + H_2}{2}$ is the average value of the height of the nozzle inlet and the height of the nozzle outlet. The Reynolds numbers for the flow regimes for the Winklhofer nozzle is in the range of $Re_{\Delta p=60 \text{ bar}} \simeq 4500$ to $Re_{\Delta p=90 \text{ bar}} \simeq 5500$. This indicates that the flow is not very turbulent and running preliminary simulations without a turbulence model should not yield large errors to the final solution.

All simulations in this part of the report is incompressible. A quick estimation of the Mach number from equation 1.4 based on the Bernoulli velocity scale $U_b$ from equation 1.2 calculated for the highest pressure drop used in this part of the report, $\Delta p = 90 \text{ bar}$ showed that the maximum Mach number for liquid would be $\sim 0.1$ and the maximum Mach number for vapor would be $\sim 0.2$. The consensus is that flows with Mach numbers below 0.3 can be treated as incompressible.

### 4.3.2 Fluid properties

The objective for this study is to investigate the flow in diesel fuel nozzles so the fluid properties used is those corresponding to diesel fuel. Diesel fuel, as for most petrochemicals, are a fluctuating entity and the fluid properties vary from type to type depending on their use. Winklhofer does not specify any fuel properties, but only refers to it as "Diesel". Fortunately, Kärrholm provides a full disclosure of the physical properties of his fuel, so it was decided to employ these properties as they had already been used to reproduce the Winklhofer experiments. Kärrholm does however state that there are
relatively large margins of error in his fuel data as they are a combination of conventional diesel and n-heptane, but still perform in accordance with the experimental data of winklhofer.

Table 4.2: Physical properties of Diesel Fuel

<table>
<thead>
<tr>
<th>Property</th>
<th>Liquid phase</th>
<th>Vapor phase</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density</td>
<td>832</td>
<td>0.1361</td>
<td>$\text{kg/m}^3$</td>
</tr>
<tr>
<td>Dynamic Viscosity</td>
<td>0.0065</td>
<td>$5.953 \cdot 10^{-6}$</td>
<td>$\text{Pa} \cdot \text{s}$</td>
</tr>
<tr>
<td>Speed of sound</td>
<td>1400</td>
<td>632</td>
<td>$\text{m/s}$</td>
</tr>
<tr>
<td>Vapor Pressure</td>
<td>5400</td>
<td></td>
<td>$\text{Pa}$</td>
</tr>
</tbody>
</table>

Kärrholm states that the speed of sound for the vapor phase at room temperature is estimated from the ideal gas law. The speed of sound for liquid is taken from measurements on diesel fuel from the journal of the American Oil chemist society [18]. The evaporation pressure and the vapor density is taken of n-Heptane while the liquid density is given by winklhofers experiments. Further he states that most of these parameters does not affect the results, apart from the liquid viscosity. It was therefor decided to use these fuel properties for the remainder of the project.

4.3.3 Cavitation model parameters

There are several model parameters to be set in order to perform the simulation. The most integral one being the parameters for the cavitation model. As stated in section 3.5 the cavitation model is based on a homogenous distribution of bubbles with a fixed number density $n_0$ and the variable bubble radius $R$. The input parameters, to be specified by the user, for the cavitation model is

- The initial bubble radius $R_0 \ [\text{m}]$
- The bubble density $n_0 \left[ \frac{1}{\text{m}^3} \right]$

The two above parameters constitute the initial vapor volume fraction

$$\alpha_{v0} = \frac{n_0 \frac{4}{3} \pi R_0^3}{1 + n_0 \frac{4}{3} \pi R_0^3} \quad (4.1)$$

These three parameters should be balanced to emulate real fluid properties. These parameters where the most difficult to determine. Winklhofer did not
perform any investigation of presence of bubbles and does not give nuclei content any attention. The numerical model used by Kärholm is not based on a homogenous distribution of bubble seeds as shown in section 3.9 so no attention towards bubbles in his work either. When referring to the literature the most common values are within the same range as the values in table 4.3.

Table 4.3: Typical values for the cavitation model (Giannadakis,2005)

<table>
<thead>
<tr>
<th>Variable</th>
<th>$n_0 \left[ \text{nuclei/m}^3 \right]$</th>
<th>$R_0 \left[ \text{m} \right]$</th>
<th>$\alpha_0$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Typical values</td>
<td>$10^{12} \sim 10^{14}$</td>
<td>$0.3 \cdot 10^{-6} \sim 2 \cdot 10^{-6}$</td>
<td>$5 \cdot 10^{-6} \sim 5 \cdot 10^{-5}$</td>
</tr>
</tbody>
</table>

The values in table 4.3 is taken from the PhD thesis of Emmanouil Giannadakis (2005) [10], but similar values are presented in other work like the one Yuan (2001) [20] and Martynov (2005)[14]. As stated in 4.3.1 turbulence models was not included at when tuning the cavitation parameters. Kärholm writes: “Turbulence is not modelled due to the relatively weak influence it is believed to have on the nozzle flow compared to cavitation.” (Kärholm et al. Modelling injector flow including cavitation effects for diesel applications, FEDSM2007-37518, 2007) [8]. The literature proposed different combinations for the cavitation parameters so it seemed as the parameters for the cavitation model is dependent on many factors and would be somewhat unique for each case. A condition for hydrodynamic similarity for cavitating flows was presented in the ph.d thesis of S.Martynov [14] and is cited to the work of Lecroffre and Bennin [12] and Lecroffe [13]. Assuming a bubbly type of cavitating flow, the necessary condition for hydrodynamic similarity of cavitating flow is scaling the number density $n_0$ according to the hydrodynamic scale of the flow like

$$n_0 \sim \frac{1}{\ell_\infty^3} \quad (4.2)$$

where $\ell_\infty$ is the hydrodynamic scale of the flow i.e. the hydraulic diameter of the rectangular duct. Since the throttle is slightly contracting the average nozzle height $H_{avg}$ is used, the hydraulic diameter for the nozzle, based on the dimensions shown in figure 4.2 is

$$\ell_\infty = D_h = \frac{2H_{avg} W}{H_{avg} + W} = 296.2 \mu m \quad (4.3)$$

By use of the scaling (4.2) the number density was set to

$$n_0 = \frac{1}{(296.2 \cdot 10^{-6})^3} = 3.85 \cdot 10^{10} \left[ \frac{1}{m^3} \right] \quad (4.4)$$
which is slightly lower than the values presented in table 4.3. The initial volume of fraction was set to be $\alpha_0 = 1 \cdot 10^{-5}$ and equation (4.1) was solved with respect to $R_0$ to calculate the initial bubble radius

$$R_0 = \sqrt[3]{\frac{\alpha_0}{(1 - \alpha_0) n_0 \pi}} = 3.96 \, [\mu m] \quad (4.5)$$

The initial radius and the seed density where used as input in the STAR-CCM+ calculation. The flow field was initiated as stagnant with initial pressure equal to $p_{inlet}$ as stated in section 4.3.1 and the simulation was run transient with a time step of $\Delta t = 3 \cdot 10^{-8} \, s$ until the solution reached a steady state solution. The simulation did not converge numerically as the residuals was high. Numerical convergence was a big problem during the development phase, but the physics from the simulations made good sense and the simulations was considered trustworthy at this point of the development process. It will be shown later that the convergence problems originated from the initial conditions. The convergence problem vanished if a fully developed flow was obtained before activating the cavitation model. It was decided to use a pressure drop $\Delta p = 85 \, bar$ for the development process since this pressure difference yielded a supercavitating regime. Both Winklhofer and Kärholm had cavitation visualizations at this pressure drop. Figure 4.5 shows the distribution of cavitation obtained by winklhofer 4.5a and Kärholm 4.5b by use of a pressure difference corresponding to $\Delta p = 85 \, bar$ and a cavitation number $CN = 5.68$. Both figures shows the liquid as red and the vapor as blue, the cavitation regime for this pressure drop is clearly supercavitation as the cavitation cloud stretches all the way through the nozzle. It was decided that these images would be used as benchmark for the CCM+ cavitation model. Note that figure 4.5b uses cavitation probability to show the cavitation field. The value he presents is a average value over the last 20 time steps. As the simulations in this report reaches a steady state, instantaneous distributions of the volume fraction of vapor $\alpha_v$ is shown instead. The cavitation fields shown in figure 4.5 is not equal and it seems like the numerical model of Kärholm overestimates cavitation. For this study it was decided that a model that could predict supercavitation and mass flow rates within a sensible range of the mass flow rates presented by Winklhofer and Kärholm would be satisfactory.
4.3 Model parameters

Figure 4.5: The $dp = 85$ bar supercavitation regimes of Winklhofer and Kärrholm (Kärrholm, 2007)

Figure 4.6: Distribution of cavitation using hydroscaled values for seed distribution and radius, $dp = 85$ bar

Figure 4.6 show the results from the simulation using hydroscaled values for the cavitation model. Notice that the colors in figure 4.6 are opposite of the ones presented by Winklhofer and Kärrholm. This is because the author found it more interesting to show the volume of fraction for the
Development and tuning of the numerical model

vapor phase than the liquid phase. As one can see the cavitation field for
the hydrodynamic scaled cavitation model noes not nearly produce the same
cavitation field as the experimental data. Figure 4.6 is a two dimensional
plane at the symmetry plane and the plane has been mirrored over the
bottom symmetry plane to show the whole side view at the symmetry plane.
It is not possible to see, but there is vapor present in a few cells right at the
nozzle entrance. The literature states that the key parameter for cavitation
is the seed density, it was decided to keep the initial volume fraction of vapor
fixed at $\alpha_v = 1 \cdot 10^{-5}$ and vary the initial radius according to the seed density
when choosing cavitation parameters.

Based on the results shown in figure 4.6 it was decided that the seed
density should be increased so that the values of the cavitation parameters
are within the recommended values from table 4.3. Inspired from the values
initial radius where set to 0.5 $\mu$m corresponding to a seed density of $n_0 =
1.9 \cdot 10^{13}$ $[1/m^3]$. The simulation where run transient from a stagnant flow
field using $\Delta t = 3 \cdot 10^{-8}$ s as for the previous case.

![Figure 4.7](image)

**Figure 4.7:** Distribution of cavitation and velocity using $R_0 = 0.5 \mu m, n_0 = 1.9 \cdot 10^{13}$ $1/m^3$ and $\alpha_0 = 1 \cdot 10^{-5}$, $CN = 5.68$

Figure 4.7a show the volume of fraction for the vapor phase for the sim-
ulation using the new values for the cavitation model. The nozzle is still
not experiencing supercavitation, but it has improved significantly from the
one cavitation field in figure 4.6 and does now show the characteristics of
sub-cavitation or traveling cavitation where vapor is produced and limited
to the recirculation zone just after the nozzle entrance. The velocity dis-
tribution shown in figure 4.7b clearly show this recirculation zone and the
vena contracta that form after the nozzle entrance. Normally, these values
for the cavitation parameters should be sufficient to obtain the cavitation
regime, but it is believed that the relatively high dynamic viscosity of the
diesel fuel that is approximately 6.5 times higher than for tap water the
viscous stresses are higher and that can result in higher critical pressures for
the onset of cavitation. After an iterative process the following values for
$R_0$, $n_0$ and $\alpha_0$ was determined.

**Table 4.4**: The final parameters for the cavitation model

<table>
<thead>
<tr>
<th>Variable</th>
<th>$n_0$ [$\text{nuclei/m}^3$]</th>
<th>$R_0$ [m]</th>
<th>$\alpha_0$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Values</td>
<td>$2 \cdot 10^{18}$</td>
<td>$1.061 \cdot 10^{-8}$</td>
<td>$1 \cdot 10^{-5}$</td>
</tr>
</tbody>
</table>

Table 4.4 show the final parameters for the cavitation model and these values will be applied to every model presented in this report from here on out. Note that the values in table 4.4 is outside the recommended values from table 4.3. The decision of exceeding the recommended values for the cavitation model is based on the PhD study of S. Martynov (2005)[14]. Martynov encounters the same difficulties obtaining the cavitation model parameters for the Winklhofer nozzle. Martynov proposes the high liquid viscosity of the Diesel fuel as the reason and proposes to use the seed density of $n_0 = 2 \cdot 10^{18}$. As for the previous simulations, the flowfield was initiated at rest and the simulation was run transient with $\Delta t = 3 \cdot 10^{-8}$ s until it reached a steady state solution.

![Figure 4.8: Distribution of cavitation using $R_0 = 0.01061 \mu m, n_0 = 2 \cdot 10^{18}$ $1/m^3$ and $\alpha_0 = 1 \cdot 10^{-5}$, $CN = 5.68$](image)

The cavitation field for the simulation is shown in figure 4.8. The figure shows a long field of cavitation bubbles stretching from the nozzle entrance towards the end of the nozzle. One can discuss whether this is a transitional cavitation regime or supercavitation. Higher values of the seed density ranging up to $n_0 = 2 \cdot 10^{22}$, but the effect of the cavitation was negligible and it was decided at this point to keep the cavitation parameters from table...
4.4 and to focus the attention on numerically converging the solution and to implement turbulence models.

4.3.4 Turbulence

As stated earlier the iterative process of obtaining the cavitation parameters have been conducted without applying a turbulence model. The turbulence model applied in this study is the standard k-epsilon Two-Layer model. This model was chosen because it provided the best results for the simulations. The standard k-epsilon Two-Layer is also very flexible and can be applied for meshes where the viscous sub layer is fully resolved and for meshes where the cells close to walls are within the logarithmic region. Turbulence is specified by the turbulent length scale

\[ l_{turb} = 0.1 \cdot H_{avg} \simeq 0.03 \, mm \]

and turbulence intensity

\[ I = 0.01 \]

at the inlet and outlet boundary as stated in section 4.3.1. The initial turbulence condition is specified with the same \( I \) and \( l_{turb} \) as for the boundary with an additional input for the to

\[ u_{turb} = 2 \, m/s \]

since the flow is initialized as stagnant fluid the initial conditions for turbulence is considered to have little effect of the final solution.

To include the effect of turbulence on the production of cavitation Singhal et. al (2002) [3] proposes to incorporate the turbulent pressure fluctuations to the critical pressure for cavitation. Up until now the assumption has been that \( p_{cr} = p_{vapor} \), but to include the effect of turbulence the critical pressure for cavitation becomes

\[ p_{cr} = p_v + \frac{1}{2} p'_{turb} \tag{4.6} \]

where the turbulent pressure fluctuation \( \frac{1}{2} p'_{turb} \) is based on the local turbulent kinetic energy like

\[ p'_{turb} = 0.39 \rho k \tag{4.7} \]

Figure 4.9 shows the cavitation field after incorporating the turbulence model and the turbulent pressure fluctuations into the critical pressure. The cavitation field shown in figure 4.9 is fully developed supercavitation. Including the turbulent pressure fluctuations in the critical pressure for cavitation was shown to have little effect on the cavitation regimes. The maximum
value for $p_{cr}$ in the domain was $p_{cr} \sim 4 \cdot 10^5 \, Pa$, located at the nozzle entrance where the local pressure is much lower than the vapor pressure anyway. Through the nozzle the pressure fluctuations did no make any significant difference for the development of cavitation. Despite the relatively weak influence it had on the results it was decided to keep the modification to the critical pressure.

4.4 Solution Procedure

In the previous section the most important model parameters are presented for the cavitation model and a tiny portion of the iterative process obtaining them have been presented. All results so far are obtained by initializing the flowfield as stagnant. This has in turns led to poor convergence with respect to the residuals as stated in section 4.3.1. The physics for all the simulations are consistent and the models behave in a sensible way. It is assumed that the model development can be trusted, but the simulations must be numerically converged before the model can be approved. The rule of thumb within CFD is that a simulation is converged if the residuals is below $1 \cdot 10^{-4}$. Several strategies was tested in order to bring down the residuals. As stated earlier the simulations so far have been initialized from a stagnant fluid and the cavitation model was activated during the entire simulation. Using the low time step of $\Delta t = 3 \cdot 10^{-8} \, s$ was time consuming to obtain the steady state solution since the time evolution for the initial transient was very slow. The first step was to run the simulation using a larger time step, in the order of 10-100 times larger than the original time step, in the beginning so that the
flow could develop into a steady state before the time step was set back to \( \Delta t = 3 \cdot 10^{-8} \) s. Although this strategy was successfully in terms of saving CPU time and obtaining the cavitation regimes presented in section 4.3.3 the residuals where very high. Figure 4.10a show the residual monitor for a simulation initialized as stagnant and where the time step has been varied to lower the residuals.

![Residual monitor for a simulation where the flow field was initiated from a steady state solution](image1)

![Residual monitor, flow initialized from a steady solution](image2)

(a) Residual monitor for a simulation where the flow field was initiated from a steady state solution

(b) Residual monitor, flow initialized from a steady solution

**Figure 4.10:** Residual monitors for different solution strategies

Something that should be noted is that running a cavitation simulation is very time demanding due to the required time step and even when running the simulations parallel on 8 processor finding a steady state solution could take as much as two days. This in turn made it difficult to start a new simulation from scratch whenever changing time step or number of inner iterations. Several time steps was used ranging from \( \Delta t = 1 \cdot 10^{-7} \) s to \( \Delta t = 5 \cdot 10^{-9} \) s. The final time step that was decided to use when quantifying the simulations to reproduce the mass flow versus pressure drop presented by Winklhofer was \( \Delta t = 1 \cdot 10^{-8} \) s. This time step is used for all simulations involving cavitation for the remainder of this report. All simulations so far have been run transient. It was discovered that using a steady solver to obtain the steady state solution was very effective as converging a steady solver only required a few hours of CPU time. The strategy of using a steady solver to obtain the initial flow field was therefor chosen to be the best strategy and the was used for the remainder of the study. This did not solve the convergence problems, but the CPU time was reduced. Figure 4.10b show a simulation where the flow is initialized from a steady state solution. The figure show that the residual for the Diesel phase is still very high, the convective discretization scheme was changed from second order upwind to first order upwind as this should have a positive effect on the residuals. The sudden drop in the residuals in figure 4.10b marks when the discretization scheme was changed. As one can see changing the the
convective scheme from second order to first order had a positive effect and the residuals was lowered. It was therefore decided to continue to use this scheme. The strategies presented in this section is only a portion of the iterative work performed when developing the model. At the end it was discovered that the convergence problems originated from the cavitation model. It was discovered that if a steady state solution was obtained without activating the cavitation model solved the convergence problems for the transient simulations. All previous simulations was run with the cavitation model activated at all times. When initializing the flow field for the transient simulation based on a steady state solution obtained without activating the cavitation model the convergence problems was gone. Figure 4.11 show the final solution obtained when employing the strategy presented here. The initial flow field for the transient simulation is obtained from a steady simulation where the cavitation model is not activated, the first order upwind discretization scheme was used for the segregated flow solver and the time step for the transient simulation was $\Delta t = 1 \cdot 10^{-8}$ s. The rest of the simulations presented in this section all employ these settings.

Figure 4.11: Final distribution of cavitation for $dp = 85$ bar, $CN = 5.68$, Super-cavitation
4.5 Results from the tuned cavitation model

As stated early on in this chapter, detailed and precise data on cavitation is scarce so the validation process for this study will be based mostly on global parameters as massflow through the nozzle and comparing images of cavitation regimes. Unfortunately there are not as many flow parameters to use as benchmark for the numerical solver with respect to the flow field. It was therefore decided that if the model developed in this study could reproduce the cavitation regimes and mass flows presented in Winklhofers paper [7] the model would be implemented to the full scale MAN Diesel nozzle under real life operating conditions to give a preliminary estimate of the presence of cavitation.

A range of pressure differences has been used to calculate the mass flows and corresponding cavitation regions on order to validate this model. The key elements are to see if the mass flow chokes, i.e becomes independent of increase in pressure differences, at any point and if the regimes of cavitation inception, sub cavitation, transitional cavitation and supercavitation occurs accordingly to experimental findings of Winklhofe.

![Figure 4.12: Mass flow versus pressure difference](image)

Figure 4.12 shows the mass flow as a function of the pressure drop. It
4.5 Results from the tuned cavitation model

includes the experimental mass flow rates of Winklhofer [7], the numerical mass flow rates of Kärrholm [8] and the mass flow rates obtained with the CCM+ solver. To visualize the choking effect on mass flow both the steady state solution without cavitation and the transient cavitation simulation is presented. The figure clearly show that for a pressure drop between 75 bar and 78 bar the mass flow starts to choke and becomes independent from the increasing pressure difference. The mass flow presented by Winklhofer shows choking at a slightly lower pressure drop around 71 bar. He also states in his report that critical cavitation occurs at 71 bars. This is also confirmed by Kärrholm, who for some reason is able to predict the point of critical cavitation at almost the exact same pressure drop as Winklhofer even though the mass flow is quite a bit higher. It is difficult to draw some precise conclusion as to why the mass flow between the numerical model of Kärrholm and the one presented in this study is so different since they both employ the same fluid properties. One possible explanation is that Kärrholm in his paper [8] states that he uses slip condition for the top and bottom wall because he thinks that ordinary no slip conditions would overestimate the thickness of the boundary layer yielding vortices in the center of the channel. The slip condition would reduce the friction along the walls and can possibly be attributed the higher mass flows. The model developed in this study uses no slip for all walls, but wall roughness was not taken into consideration. Also the fact that Kärrholm uses a different model and the fact that his cavitation fields are denser than the ones obtained with the model developed in this study are possible explanations to the observed differences in mass flow rates. On the basis of mass flow the experimental values from Winklhofer and the numerical values from the CCM+ model does coincide well.

In order to relate the mass flow and nozzle discharge coefficient to the cavitation regime they have been calculated and are presented in figure 4.13 using the cavitation number, from equation (2.2), as reference. The discharge coefficient $C_d$, from equation (1.5), is based on the theoretical mass flow rate using the bernoulli velocity scale $U_b$ from (1.2) and the average cross sectional area of the contracting nozzle and the actual mass flow. The point for critical cavitation, where the flow chokes, has not been determined directly, but it is fairly obvious that the flow chokes somewhere between a pressure drop of 75-78 bar corresponding to a cavitation number in the range of CN=3 to CN=3.5. As for the discharge coefficient, it is fairly stable until the critical point where it starts to decay. This is expected as the theoretical mass flow is a function of the pressure difference while the choked flow is not pressure dependent any more.
Development and tuning of the numerical model

Figure 4.13: Discharge coefficient and mass flow compared to the Cavitation number

Figure 4.14 shows the cavitation fields corresponding to the pressure drops and mass flows discussed in this section. Starting from cavitation inception shown in 4.14a where cavitation is formed only at the nozzle entrance due to a sudden pressure drop due to the change in geometry. The cavitation at the nozzle inlet for figure 4.14a is difficult to see when observing the plane, but when observing the nozzle form above it is clear that cavitation bubbles are forming in the very thin prism layer cells close to the wall. Further increasing the pressure drop and cavitation number to 4.14b the cavitation is starting to form in the recirculation zone at nozzle entrance. This correspond to the regime called inlet cavitation or subcavitation. Transitional cavitation where clouds of vapor bubbles are convected downstream the nozzle with the flow is shown in figure 4.14d. This cavitation regime is very close to the point of critical cavitation and is very sensitive changes in the pressure drop. Only a slight rise or fall in the back pressure is believed to have significantly impact on the extent of the vapor cloud. There was unfortunately not enough time in this study to elaborate more on this matter than isolating the critical point within a small range of pressure drops. The remaining regimes are all supercavitation as the cavitation number is past the CCN (critical cavitation number). The vapor cloud in all of them are approximately of the same size and density and one can see the contraction of the flow is approximately the same size for all of them, explaining that the mass flow does not change when increasing the pressure drop.
4.5 Results from the tuned cavitation model

To investigate the pressure distribution inside the nozzle, the pressure has been plotted along the centerline and at a distance of $0.04 \cdot L$ from the upper wall. The nozzle entrance is where $x = 0$ to include the decay and recovery of pressure at inlet and outlet. Figure 4.15 show the pressure distributions along the centerline and close to the wall for two different cavitation regimes, namely the supercavitating flow of $dp=85$ bar and the sub cavitating flow of $dp=75$ bar. For the supercavitating flow the pressure inside the nozzle almost instantly drops below the vapor pressure, denoted with a blue line, and does not recover before entering the back chamber. When the pressure in the nozzle is below the vapor pressure the bubbles will not collapse and
Development and tuning of the numerical model

Figure 4.15: Pressure distribution inside the nozzle

(a) Pressure in nozzle for pressure difference $dp = 85$ bar, $CN = 5.68$

(b) Pressure in nozzle for pressure difference $dp = 75$ bar, $CN = 3.00$

is convected through the nozzle. For the sub cavitating flow the pressure recovers not so far downstream of the entrance and thus the bubble collapses. From figure 4.15b it may seem like the pressure never drops below the vapor pressure at all, but this is because the probe measuring the pressure close to the wall is placed with a $0.04 \cdot L$ offset to the wall.

To relate the pressure levels inside the nozzle to the production of
bubbles the source term of cavitation $S_{cav}$ from equation (3.33) is shown as a scalar function in figure 4.16. The positive cavitation production term correspond to growth of bubbles and it shows that bubbles are formed at the nozzle entrance where the local pressure is lowest, and then the bubbles are convected downstream the nozzle. Figure 4.16b show the negative cavitation production, corresponding to bubble collapse. The figure also verify that the bubbles are convected downstream the nozzle and collapses at the nozzle exit where the pressure rises above the vapor pressure. The negative bubble production rate observed at the nozzle entrance is most likely caused by the flow direction as bubbles exits the low pressure zone right at the entrance. Comparing with the positive production rate in figure it seems like the is a defined line between the cells where bubbles grow and collapses.

Figure 4.16: Cavitation production rates for dp=85bar CN = 5.68

4.6 Investigation of grid dependence

As stated earlier, several grids have been applied at the initial stages of the development period. However, there have been little focus on developing a solution that is independent of the grid size. The grid used as the final model consist of 140 641 cells. To check if the solution from this grid is dependent on the grid size a new grid, consisting of twice as many cells have been generated.

Figure 4.17: The refined Winklhofer grid

Figure 4.17 show a grid consisting of 309 440 cells. The flow conditions used
in this simulation is the same as used in the development, namely the \( \Delta p = 85 \text{ bar} \) pressure drop. This pressure drop should induce supercavitation in the nozzle.

Figure 4.18: Cavitation field for refined grid, \( \Delta p = 85 \text{ bar} \), \( CN = 5.68 \)

The cavitation field for the grid refined model is shown in figure 4.18. The model predicts supercavitation as for the original grid but the bubble cloud stretches a little bit further for the refined grid.

Table 4.5: Parameters for the cavitation model

<table>
<thead>
<tr>
<th></th>
<th>140K</th>
<th>300K</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \dot{m} )</td>
<td>8.149</td>
<td>8.12</td>
</tr>
<tr>
<td>( C_d )</td>
<td>0.78</td>
<td>0.778</td>
</tr>
</tbody>
</table>

Based on the visual cavitation field in figure 4.18 and the mass flow and discharge coefficient data in table 4.5 it is concluded that the original grid is of adequate quality and the model validation can be trusted. It is therefore decided that the model developed in this study is valid and can be incorporated in the real life application for the MAN Fuel injector. It should be stated that the 300K grid used here is exactly similar to the 140K, Whereas the 140K grid have two refinements zones the 300K grid only have one, spanning the entire nozzle. As a result of this the cells at the nozzle inlet are not refined, but are approximately the same size as for the 140K grid. This was unfortunately discovered late in the project and there was no time left to check this again. This adds uncertainty to the validity of the investigation of grid dependence.
4.7 Discussion for the model development

The cavitation model has now been developed and validated. However, some simplifications are made with respect to the design of the nozzle. Winklhofer states in his report that the total height of the geometry is 1 cm, but the total height of the geometry used in this study is $2 \cdot L$. As a result of this large recirculation zones form in the back chamber due to the no-slip boundary condition for the walls.

![Vector plot of the entire computational domain](image)

**Figure 4.19:** vector plot of the entire computational domain $dp = 85$, $CN = 5.68$

The recirculation zones in the back chamber can be seen in figure 4.19 where a jet enters the duct at high velocity and due to friction from the wall and the stagnant fluid creates a pair of symmetric vortices between the wall and the center of the duct. The vortices generate significantly back flow in regions of the back chamber. It is unclear how these recirculation zones affect the nozzle flow, but measurements on the pressure at the nozzle exit suggest that the back pressure for the nozzle is more or less equal to the outlet boundary. This is also shown in figure 4.15 where the pressure rises to approximately the outlet pressure when the flow exits the nozzle. Preliminary simulations were run with a larger back chamber where the total height was $4 \cdot L$ and it was found that this did not make any significant difference to the simulation. However, the simulations did diverge on occasion and when investigating the diverged solution, unphysical values of pressure and velocity were found in cells after the nozzle exit where the back flow from the top and side wall is introduced into the flow leaving the nozzle at high velocities. This did not happen often and there was no clear pattern that leads to any conclusion on the matter. The discretization scheme for the segregated flow solver was changed from a second order upwind scheme to a first order upwind scheme. This was done to improve convergence for the simulations. When a fully converged solution finally was obtained it was decided to keep
the first order convection scheme for the cavitation simulations. Generally it is recommended to use a second order scheme whenever possible since this scheme usually gives more accurate solutions. At the end of the study a simulation was run changing the convective discretization scheme back to second order to check whether using a more accurate discretization scheme would change the simulation.

Figure 4.20: Simulation using second order upwind discretization for the segregated flow solver $dp = 85$, $CN = 5.68$

Figure 4.20 show the cavitation field for the simulation using the second order upwind convection scheme for the segregated flow solver. Judging by the cavitation field in figure 4.20 it seems like there is no big error involved with using the first order scheme. The mass flow rate from the simulation is $\dot{m} = 8.08 \frac{Kg}{s}$ which is within 1% of the mass flow rate obtained when using the first order scheme.

A similar investigation was performed to investigate the effect of the additional term $p_v = p_{cr}$ that was included to account for turbulence pressure fluctuations. A simulation was run using the first order upwind scheme for the segregated flow solver in accordance with the proposed model parameters. Figure 4.21 show the cavitation field when using $p_v = 5400 \ Pa$. Comparing the cavitation field with the one where $p_v = p_{cr}$ in figure 4.11 it does not seem like the additional pressure fluctuations added to the cavitation model made any significant changes to the results. The mass flow rate for the simulation using the standard vapor pressure is $\dot{m} = 8.09 \frac{Kg}{s}$ the change in mass flow is also within 1% of the model using the modified critical pressure. This indicates that the changes made to the model regarding discretization schemes and critical cavitation pressure is redundant and does not affect the flow. This further verify the statement that the difficulties converging
4.7 Discussion for the model development

Figure 4.21: Simulation using $p_v = 5400 \, Pa$, $dp = 85$, $CN = 5.68$

the simulations originated from the cavitation model, and that the strategy of obtaining a fully developed flow field using a steady solver without activating the cavitation model was the most influential change to improve convergence.
Chapter 5

MAN Diesel F0002 Fuel Injector

5.1 Real life operating conditions

The numerical model for two phase flow and cavitation model have now been developed and tuned. The objective for this part of the project is to utilize this model to investigate the internal nozzle flow upstream of the spray in a full scale fuel injector at real life operating conditions. The internal flow in a fuel injector is very complex and the physics of vortex formation and subsequent string cavitation is still not fully understood. Similar studies on fuel injectors, like the one of Gevaises et al [2] suggest the presence of a vortex structure in the sac volume that where string cavitation can form and affect the subsequent mass flow rates though the injectors nozzles. The geometry used in this study is the fuel injector named F0002 employed in MAN Diesel & Turbo's Low Speed, Two-Stroke diesel engines. Unfortunately there has not been performed CFD simulations upstream of the spray on this fuel injector before and there are no quantitative data for mass flow and flow coefficients at operating conditions. There have been done experimental and numerical work on the F0002 nozzle previously, more precise in an industrial PhD project performed by Bjarke Skovgård Dam at MAN Diesel & Turbo and MEK-DTU focusing on the fuel spray from the fuel injector into the cylinder at different flow conditions [5]. The PhD thesis of Bjarke Skovgård Dam contains both experimental and numerical work focusing on the downstream effects of internal nozzle flow using several atomizers including the F0002. The experimental work consisted of spraying fuel through the injector and into a measurement devise that hold atmospheric pressure. The results presented in his work is time averaged correlations of mass flow
and momentum between holes, but there was no hole specific or total mass flow presented for the F0002 atomizer. The simulations presented in this report employ the real life operating pressure for the fuel pump as the inlet boundary condition and real life in-cylinder pressure as the outlet boundary condition as driving force for the flow inside the fuel injector. Consequently the results presented in this section is not validated against experimental data and modeling errors will add uncertainties to the final results, but as a preliminary study of internal nozzle flow and the presence of cavitation bubbles this was considered to be adequate.

The strategy was then to build up a computational grid generated from a neutral CAD file of the fuel injector to not be grid dependent and then use this model to investigate the internal flow and incorporate the cavitation model with the parameters developed in section 4.1. Due to the very small time step required to model the phase change from the cavitation model, it was considered too computationally demanding to run a simulation over a full injection cycle when the cavitation model was activated. Instead the simulation was run using stationary boundary conditions and the cavitation model was activated when the simulation had developed into a steady state to give an indication of the cavitation regime at this condition. The computational domain used for this study includes only the bottom half of the injector and starts from the point marked "Inlet Computational Domain" in figure 5.1. The needle lift is at the fully opened position and is kept at this position for all simulations. This is a large simplification made to avoid having to use a moving or morphing grid.

The real life operating conditions for the simulations using a constant pressure inlet boundary is based on the supply pressure from the fuel pump to the fuel injector and the in-cylinder pressure at operating conditions. In real life neither the pressure for the fuel pump nor the in-cylinder pressure is constant and this assumption adds uncertainty to the validity of the results. The inlet pressure to the computational domain is equal to the pressure applied at the point denoted Head or \( p_h \) in figure 5.1. The pressure loss due to wall friction between the point denoted Head and the Inlet Computational Domain...
5.2 Initial conditions, time step and boundary conditions

Domain in figure 5.1 is neglected and the inlet pressure from the supply pump \(p_h\) is enforced directly on to the inlet of the model. The outlet boundary pressure is based on an average cylinder pressure at operating conditions said to be in the range of 130 bar to 150 bar depending the engine type and load. The mean value of 140 bar is used as the outlet pressure boundary placed at the outlet faces of the nozzles.

Table 5.1: Model pressure boundaries for the constant boundary simulations

<table>
<thead>
<tr>
<th>Boundary</th>
<th>Inlet</th>
<th>Outlet</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure [bar]</td>
<td>800</td>
<td>140</td>
</tr>
</tbody>
</table>

To simulate a full injection cycle for the fuel injector a transient pressure signal was used as a transient pressure boundary condition for the inlet. This was done to see how the mass flow rates changes with a rise in pressure and to see if a transient pressure profile would influence the physical behavior of the flow compared to simulations using constant pressure boundaries. The pressure signal used was given as a pressure vs time plot in the PhD thesis of Bjarke Skovgård Dam [5] and was digitalized and incorporated in to CCM+ as two vectors containing pressure and the corresponding time. A injection cycle for the F0002 fuel injector is in the order of 30 ms.

5.2 Initial conditions, time step and boundary conditions

The numerical model used in this exercise is identical to the one used in the validation section, Stagnation inlet and pressure outlet boundary conditions is used for the inlet and outlet with the values from table 5.1. All walls in the domain have no slip condition. The initial flow field is set to be a stagnant with an initial pressure of 350 bar. This value is equivalent to the pressure when the valve just upstream of the inlet to the computational domain opens, shown in figure 5.1. It was considered to set the initial pressure to 140 bar, equal to the in-cylinder pressure, but the large pressure difference between the inlet boundary and the adjacent cells made the model unstable during the first iterations and led to divergence problems. To avoid this the initial pressure field was set equal to the the release pressure for this upstream valve. This simplification had little influence on the final solution and as will be shown later that the flow field develops into a statistically independent of time steady state after approximately 450 time steps. The time step used for the simulations without cavitation is set to be

\[ \Delta t = 5 \cdot 10^{-6} \]
which is adequate to capture the transient development of the vortices formed inside the SAC volume. The validity of the time step can be checked by comparison to the flow’s characteristic time scale $t^*$, a time scale for transient structures in the flow. It was decided that if the time step was smaller than this scalar the time step is small enough to capture the transients in the flow. The calculation of the time scale, based on the diameter of the SAC volume $D_{SAC}$, and the mean axial velocity, $\bar{U}_{SAC}$, entering the SAC volume, is shown in equation (5.1). The mean axial velocity is collected from the plane denoted Inlet Nozzle SAC volume in figure 5.2. For the simulations using a constant inlet pressure boundary the mean axial velocity $\bar{U}_{SAC}$ is the mean velocity for the last 100 $\tau$ to get a mean value of the fluctuations in mass flow rates. For the simulations using a transient pressure signal as input $\bar{U}_{SAC}$ is the mean velocity over the whole injection cycle.

$$t^* = \frac{D_{sac}}{\bar{U}_{SAC}} = 5.77 \cdot 10^{-5} \text{s}$$  \hspace{1cm} (5.1)

The calculation in equation 5.1 show that the time step $\Delta t \approx 0.1t^*$ and the time step is considered adequate. When plotting transient results the non-dimensional time scale $\tau$ will be used to denote the time evolution. This allows future studies, using other flow rates through the injector, to compare with the results from this study. The real physical time is normalized with the time scale $t^*$ like

$$\tau = \frac{t}{t^*}$$  \hspace{1cm} (5.2)

**Figure 5.2:** Placement of boundaries and planes for data collection

The geometry for the simulations is shown in figure 5.2. It shows the location of the inlet and outlet boundary conditions and the placement of the planes where the total mass flow and the velocity entering the SAC volume is collected. The plane denoted massflow is placed $2 \cdot D_{inlet}$ downstream of the inlet boundary and the plane denoted Inlet Nozzle SAC Volume is placed $3 \cdot D_{SAC}$ upstream the bottom point of the SAC volume.

Compressible effects are also included in simulations of the F0002 atomizer due to the high pressure difference between the inlet and outlet boundary condition. Velocities are expected to be as high as 400 m/s in the nozzles
yielding high local mach numbers. The compressible effects where implemented as described in section 3.8 by coupling the density to pressure and the speed of sound of the medium. Considering the bernoulli velocity scale $U_b$ from equation 1.2

$$U_b = \sqrt{\frac{2}{\rho} \cdot (p_{in} - p_{out})} = 398 \text{ m/s}$$

a quick calculation of the Mach number from equation (1.4) using $c_v = 632 \text{ m/s}$ for the vapor phase and $c_l = 1400 \text{ m/s}$ for the liquid phase the following mach numbers is obtained

$$M_l = \frac{U_b}{c_l} = 0.29$$

for the liquid phase and

$$M_v = \frac{U_b}{c_v} = 0.64$$

for the vapor phase. In the simulations where the cavitation model is applied the local mach number in each cells are based on the local volume of fraction like $M = \alpha_v M_v + (\alpha_v - 1)M_l$. In regions where cavitation occurs is usually subjected to large velocities and the Mach number is expected to be higher than 0.3 which is the boundary value as to where compressibility effects should be included.

The standard k-epsilon two layer model was applied for include turbulent effects in the simulations. The turbulence inlet and outlet boundary condition was defined by the turbulence intensity $I$ and the turbulence viscosity ratio $\mu_t/\mu$. There was not a lot of focus on the turbulence specification when developing the model and the default values of $I = 0.01$ and $\mu_t/\mu = 10$ was used for the simulations. This is a source of error, but it is believed that due to the long straight pipe from the inlet and down to the SAC volume the turbulence will develop making the boundary specification less influential.

The cavitation number for the F0002 fuel injector using the boundary conditions presented in table 5.1 yields the following cavitation number

$$CN = \frac{800 \cdot 10^5 - 140 \cdot 10^5}{140 \cdot 10^5 - 5400} = 4.72$$

A cavitation number of 4.72 indicates that a supercavitating regime should form inside the nozzles. It also support the statement that neglecting cavitation when simulating the flow upstream of the nozzles would lead to errors since part of the physics are left out, e.g resistance in nozzles.

### 5.3 Mesh

The mesh was generated from a neutral CAD file containing the assembly of the full fuel injector, the 3D-CAD function Extract Internal Volume was used
to extract the computational domain for the fuel injector. The geometry extracted is shown in figure 5.3a and can be seen as the internal volume for the nozzle downstream the second valve in figure 5.1, from the point denoted "Inlet Computational Domain" to the nozzle exit.

![Internal volume of the F0002 atomizer](image1)

![Cross section of preliminary mesh](image2)

Figure 5.3: Internal volume and preliminary grid for the F0002 geometry

Figure 5.3 show the extracted internal volume and a preliminary mesh of the F0002 fuel injector. As one can see the geometry of the nozzle is relatively complex and the flow has to travel through several tiny channels before being joined together in the sac volume short before the nozzle exit. The area of interest in this report is primarily the sac volume and the nozzles, but due to the challenging geometry it would be difficult to determine proper inlet boundary conditions at this point so subsequently a computational mesh containing only the sac volume would be difficult to trust. It was later tried to make a smaller grid, including only the sac volume, and then transfer pressure and velocity fields and turbulence specifications from the full model, but this has proven to be difficult and due to time limitations in the project this strategy was abandoned. Some geometry modification had to be made on the extracted internal volume for meshing purposes. Some design features from the nozzle led to difficulties when generating the grid in some pointy areas of the geometry where the wall thickness approached zero, especially when trying to insert cells into these pointy and sharp interphases. The sharp edges was removed using the *remove cut* utility in the CCM+ CAD environment 3D-CAD.

Figure 5.4 show the placement for the sketches used to smoothen the sharp edges on the geometry 5.4a and the modified geometry 5.4b. It is believed that when removing these parts of the geometry, the quality of the grid would improve without influencing the flow field as neither of these edges are acting as obstacles for the flow.

Figure 5.5 show a close up of the lower part of the total geometry, namely the SAC volume and the nozzles. As stated earlier this is the area of the grid that is of interest of this project and most of the investigation and dis-
5.3 Mesh

(a) Sketches for the revolve remove utility in the 3D-CAD environment

(b) Modified internal volume after the sharp interphases and the infinitely thin walls where removed

Figure 5.4: Modifications to the F0002 geometry

(a) Unmodified geometry
(b) Modified geometry

Figure 5.5: Close up of the SAC volume and the nozzle outlets

cussion will be focused on this area. Removing the edges shown in 5.5a and 5.5b had a large influence in cell quality in these regions. The fuel injector has four nozzles and to distinguish them from each other they are numbered counting left to right with reference to 5.5: nozzle 1, nozzle 2, nozzle 3 and nozzle 4. This notation will be applied for the remainder of this study. Note from figure 5.5b that nozzle 4 is not fully open despite the needle lift is at a fully opened position.

The meshing tools used for the F0002 geometry is the same as for the validation case, namely
with two local refinement areas in the SAC volume area to make sure that the flow regimes are properly resolved in the SAC volume and in the four nozzles. 5 prism layers are used in the unrefined area and the first refinement area and 10 prism layers are applied to the third refinement area furthest to the left in figure 5.6.

Two refinement zones was used in order to have a gradual decrease in cell size allowing the majority of the mesh, upstream the SAC volume, to consist of relatively large cells and the SAC volume and nozzles could have small cells without risking interpolation errors between large and small cells.

5.3.1 Grid independent solution

As stated earlier experimental data corresponding to the operating conditions in this report does not exist so the model validation process regarding the F0002 fuel injector has to depend only on consistency in mass flow data for different grid sizes. This means that a grid independent solution should be obtained. Simulations was run with different cell sizes to make sure that an adequate number of cells where used for the final model. Figure 5.7 show the result from the study of grid independence. The time averaged mass flow data is extracted from a plane section placed $2 \cdot D_{inlet}$ downstream of the inlet boundary as shown in figure 5.2 and averaged over the last $5\tau$. the mass flow is extracted using the CCM+ mass flow report function. The mass flow through a boundary is calculated as \cite{1}

$$\dot{m} = \int \rho \mathbf{v} \cdot d\mathbf{a} = \sum_{f} \rho_f \mathbf{v}_f \cdot \mathbf{a}_f$$

(5.4)

where $\rho_f$, $\mathbf{v}_f$ and $\mathbf{a}_f$ is the cell face density, velocity and area vector respectively. Judging by the mass flows rates in figure 5.7 the solution is no
longer grid dependent in terms of mass flow rates for meshes consisting of more than 450 000 cells. It should be noted that the solution never becomes fully independent of time and can therefore not be called a steady state solution, but becomes independent of initial transients and reach a repeatable oscillating state. This oscillating flow pattern will be investigated later in section 7.

5.4 Convergence problems for continuity equation

When developing the model there was difficulties to numerically converge the continuity equation. Despite the grid independence mass flow the residual for the continuity equation did not converge to a satisfactory level, \( r_{\text{continuity}} < 1 \cdot 10^{-4} \). Whereas the residuals for the momentum equation and the turbulence equations converged the continuity residual leveled out at approximately \( r_{\text{continuity}} = 0.002 \sim 0.003 \). The residual monitor for simulation using a 700 000 cell grid is shown in figure 5.8 and as one can see the continuity residual does not meet this convergence criteria.

As stated in section 3.3 the continuity residual is the equivalent of the mass imbalance for each cell given by equation (3.26). The spatial distribution of mass imbalance can shown in a scalar scene in CCM+. The threshold function was applied to show all cells in the domain where the mass imbalance, and hence the continuity residual, is greater than \( r_{\text{continuity}} 1 \cdot 10^{-4} \). This i-
vestigation showed that the continuity equation had problems converging in the cells adjacent to the nozzle inlet. The most obvious solution is to reduce the time step to improve the CFL condition and to increase the number of inner iterations in order to converge the simulation. Both these attempts failed as the continuity residual still resided above the convergence criteria. Although it had been determined that the simulations became physically independent of grid size for grids consisting of more than 450,000 cells it was decided to increase the number of cells to bring down the residuals. Several grid sizes were tried out spanning from 500,000 cells to 1,160,000 cells. The mass imbalance for two of these grids is shown in figure 5.9.

Comparing the mass imbalance for the 700,000 grid and the 1,160,000 grid it seems that increasing the number of cells would eventually reduce the residual, but one must also note that the cells marked with dark blue in figure 5.9a and 5.10b is very close to the convergence criteria and the attention should be directed at the very few red cells just at the inlet for the nozzles as they are the ones presented in the residual monitor plot. The maximum mass imbalance value for the two grids are shown in table 5.2. The discrepancy between the value of $r_{\text{continuity}}$ and the values from table 5.2 is not understood as CD-Adapco at several occasions state that the continuity residual is equal to mass imbalance. The computational cost of increasing the number of cells by nearly half a million did not reduce the residual that much and it was decided to continue using the 700,000 cell grid, from now on just abbreviated 700K. Running a simulation for a model consisting of more than one million cells is very demanding even when running the simulation.
5.4 Convergence problems for continuity equation

Figure 5.9: Spatial distribution of the continuity residual where \( r_{\text{continuity}} > 1 \cdot 10^{-4} \)

parallel on 8 processors.

Table 5.2: Maximum mass imbalance values for the 700K and 1160K grids

<table>
<thead>
<tr>
<th>( N_{\text{cells}} )</th>
<th>700 000</th>
<th>1 160 000</th>
</tr>
</thead>
<tbody>
<tr>
<td>( r_{\text{max}} )</td>
<td>0.0007</td>
<td>0.0005</td>
</tr>
</tbody>
</table>

It is not clear what causes the bad convergence for the continuity equation in these cells. One hypothesis is that the sharp boundary condition at the outlet of the nozzles affected the flow conditions upstream and that including a portion of the cylinder into the domain, as was done for the Winklhofer model developed in section 4.1, would help bring down the residual. The in-cylinder conditions and the effect of the subsequent spray from the nozzles is difficult to incorporate in any sensible way, so it was decided to fit expansion tubes onto the nozzles instead to move the outlet boundary condition further away from the "bad" cells. The inspiration to fit expansion tubes comes from the experimental setup on a similar 5 hole atomizer by Gavaises et al. [2] where expansion tubes is used to capture the flow from each nozzle exit for measuring purposes. The expansion tubes where placed at the end of the nozzles and has approximately twice the radius and twice the length, due to the non uniform placement of the nozzles and difficulties placing sketch planes onto the nozzle exit face in 3D-CAD the placement and dimensions of the expansion tubes are not accurate, but should not affect the flow significantly and is viable for testing this hypothesis.
As shown in figure 5.10 the expansion tubes did not yield lower residuals, and the change in dimension between the nozzle and the expansion tube influences mass imbalance for the upstream cells negatively. Although it is difficult to determine if moving the pressure boundary further downstream would have any effect or if the change in dimension causes the mass imbalance for the cells to rise this strategy was abandoned. It is difficult to say if there is a transient phenomenon at the nozzle entrance causes the mass imbalance for the cell to be high or if it is cased by the numerical model, but anyway the problem has been identified and measures have been made to minimize it. It was therefore decided to accept high residuals at the entrance.
and continue with the 700K model. A close up of the SAC volume and the nozzles are shown for the 700K grid in figure 5.11.

Figure 5.11: The 700K grid
Results from the F0002 Fuel Injector

6.1 Introduction

A numerical model to simulate the internal nozzle flow for the MAN Diesel & Turbo F0002 fuel injector/atomizer has been developed and the results from the simulations is divided into three parts. First simulations run without activating the cavitation model is presented. It will become clear later in this report that the very low time step needed by the cavitation model make it difficult to obtain a comprehensive study of the flow fields inside the nozzle and therefore the main investigation of the flow will be performed for a simulation where the cavitation model is not active. Then an investigation on the cavitation fields inside the nozzles will be presented. Both these simulations are run using constant pressure boundaries for the inlet and outlet. The third simulation presented employs a transient pressure boundary condition for the inlet pressure to simulate a full fuel injection cycle at operating conditions to investigate the behavior of mass flow during the injection time and to compare with the constant boundary simulation. A very brief analyze on the inception of cavitation will be performed for the simulation with a transient pressure profile where probes have been placed in strategic places in the nozzle entrance to see the pressure development as a function of time.
6.2 Results from the full F0002 geometry without activating the Cavitation model

The strategy for this section is to run simulations on the F0002 atomizer, without activating the cavitation model, to investigate the internal flow field in the fuel injector. Not activating the cavitation model means that parts of the physics are neglected from the simulations and subsequently the flow fields might differ since the resistance associated with choked flow is not included. The cavitation number for the given pressure difference between the inlet and outlet is $CN = 4.72$ so according to the validation process in section 4.5 cavitation and choked flow conditions very likely to occur in the F0002 atomizer at operating conditions.

6.2.1 Mass flow and flow parameters

The mass flow have been measured during the transient simulation as area averaged values of planes placed at strategic places in the domain. The total mass flow rate collected from the plane called massflow from figure 5.2 and the mass flow rates for the nozzles are collected at the outlet boundaries. As stated earlier the needle is fully opened and does not move during the simulation and the initial flow is stagnant, the simulation is run transient for approximately $500 \tau$ which in real time correspond to the time for a single injection cycle in a diesel engine which is approximately 30 ms.

Mass flow for each nozzle and the total mass flow is shown in figure 6.1. As stated in section 5.3.1 the solution does not become independent of time, but when the initial transients is no longer present the mass flows stabilize in a oscillating manner which can be considered a statistically time independent steady state solution. The most apparent information to draw from the mass flows rates is that the mass flow rates is not equally distributed between the nozzles. Nozzle 1 and nozzle 2 have the highest mass flow rate while nozzle 4 have a somewhat smaller mass flow rate. In terms of oscillation, nozzle 3 experiences the largest fluctuations of mass flow rate of the four nozzles. This suggests that there are a vortex structure inside the SAC volume that affect the flow through the nozzles as will be shown in the section 6.2.2 where the vector fields and streamlines will be shown to give an explanation of the variation in mass flow. Before running the simulations is was assumed that the presence of a vortex structure inside would affect the mass flow through the nozzles, but it was not clear if this structure would be stable in a fixed position or if it would move around changing the mass flow from nozzle to nozzle. The flow field develop into this state within the first 30-35 $\tau$ before starting to oscillate with what appears to be a stable frequency. The frequency have been found for the individual nozzles by counting peaks manually and is converted into the non-dimensional Strouhal number with
6.2 Results from the full F0002 geometry without activating the Cavitation model

![Graph showing mass flow over time for different nozzles](image)

**Figure 6.1:** Mass flow

...the Bernoulli velocity scale based on the inlet and outlet boundary condition and the diameter of the SAC volume. The frequencies and Strouhal numbers are shown in table 6.1.

\[
U_{b\,tot} = \sqrt{\frac{2}{\rho} (p_{inlet} - p_{outlet})}
\]  

(6.1)

<table>
<thead>
<tr>
<th>Nozzle</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>Total</th>
</tr>
</thead>
<tbody>
<tr>
<td>Frequency [Hz]</td>
<td>1389</td>
<td>1393</td>
<td>1385</td>
<td>2278</td>
<td>1377</td>
</tr>
<tr>
<td>St</td>
<td>0.0243</td>
<td>0.0244</td>
<td>0.0243</td>
<td>0.0399</td>
<td>0.0241</td>
</tr>
</tbody>
</table>

Table 6.1: Frequencies and Strouhal numbers for the nozzles

The frequencies for nozzle 1, nozzle 2 and nozzle 3 are equal, the negligible deviations can be blamed on inaccurate measuring by hand, but the frequency for nozzle 4 is twice as high. The reason for this is that nozzle 1, nozzle 2 and nozzle 3 all have two distinguishable frequencies whereas nozzle 4 only have one. A close up of the mass flows rates from figure 6.1 is shown in figure 6.2 to illuminate these frequencies. The frequencies was collected...
Results

by counting 10 top peaks manually and dividing by the real time difference $\Delta t$. The hypothesis is that these oscillations in the mass flow rate is generated by the movement of the vortex structure inside the SAC volume, this will be discussed and compared to flow fields in section 6.2.2.

![Figure 6.2: Close up of mass flow fluctuations](image)

It is established that the mass flow through each of the nozzles is not equal. To further investigate the distribution between the nozzles some other scalars have been calculated for comparison. To visualize the distribution of mass flow between the nozzles each nozzle’s mass flow rate is the mean of 100 $\tau$ and divided by the mean mass flow for all the nozzles.

![Figure 6.3: Normalized mass flow for individual nozzles](image)

The variation of mass flow between the nozzles are shown in figure 6.3. As
6.2 Results from the full F0002 geometry without activating the Cavitation model

already noticed the highest mass flow goes through nozzle 1 and nozzle 2. Individual discharge coefficients and momentum coefficients for each nozzle have been calculated and is shown in figure 6.4

![Flow coefficients](image)

**Figure 6.4:** Flow coefficients

To disregard the pressure loss due to friction in the fuel injector from the inlet down to the SAC volume, the discharge coefficient \( C_d \) and the momentum coefficient \( C_m \) is based on the pressure at the SAC inlet \( p_{\text{inlet SAC}} \) instead of the the injection pressure \( p_{\text{inlet}} \). The pressure at the SAC inlet is extracted from the plane called *Inlet Nozzle SAC Volume*. shown in figure 5.2.

\[
U_{\text{b SAC}} = \sqrt{\frac{2}{\rho} (p_{\text{inlet SAC}} - p_{\text{outlet}})}
\]  

The loss due to friction can be shown by calculating a second discharge coefficient based on the pressure difference between the inlet boundary and the outlet boundaries, the velocity scale \( U_{\text{b tot}} \) is calculated in equation (6.1). Comparing \( C_d \) and \( C_{d \text{ tot}} \) show that there is a 12% loss from the inlet boundary to the plane called *Inlet Nozzle SAC Volume* in figure 5.2. This show that there are losses due to wall friction from the inlet and down to the SAC volume. This add some uncertainty to the inlet pressure condition used in this study since it does not account for the loss of pressure from the injector head to the inlet of the computational domain as shown in figure 5.1.

A spatial distribution of pressure inside the nozzle is shown in figure 6.5. It is clear from this image that the pressure in the SAC volume is approximately 100 bar lower at the entrance to the SAC volume.
6.2.2 Flow field Visualization

The images shown in this section are instantaneous images of the flow field captured at the final time step of the simulation. Unless anything else is stated all images correspond to the instantaneous flow field at \( t = 505 \tau \).

Figure 6.6 show the trajectories of the fluid as it enters the SAC volume and leaves through the nozzles. The streamlines in both figures show a large recirculation zone inside the SAC volume and thereby confirms the statement that the fluctuations in the mass flow is driven by a oscillating vortex inside the SAC volume. Figure 6.6a also show that the vortex structure forces the main flow field towards nozzle 1 and nozzle 2, this is in accordance with the mass flow rate distribution discussed in section 6.2.1.

Figure 6.7 is a side view of the SAC volume with nozzle 4 visible in on the left hand side. The plane used for figure 6.7b is placed just behind the centerline and perpendicular to the nozzles to avoid including the large velocities at the nozzle inlets. The plane is shown in figure 6.8.
6.2 Results from the full F0002 geometry without activating the Cavitation model

Figure 6.7: Streamline and vector field showing fluid path through the SAC volume and the nozzles

Figure 6.8: Placement of plane used to collect vector field

Figure 6.7a and 6.7b show the main vortex and additional smaller recirculation zones at the SAC volume wall. Something interesting to note is that nozzle 1, 2 and 3 all appear to being fed directly from the main flow the fourth nozzle seems to be fed primarily from the recirculating flow field. Judging by the amplitude of the oscillating mass flows in figure 6.1 it is apparent that nozzle 3 is influenced more by the structure than the other nozzles. Judging by figure 6.6a the core of the vortex seems to go towards nozzle 3 and its movement might influence the mass flow rate.
Figure 6.9: Transient behavior of the vortex

(a) $t = 176 \tau$

(b) $t = 184 \tau$

(c) $t = 191 \tau$

(d) $t = 196 \tau$

(e) $t = 201 \tau$

(f) $t = 210 \tau$
6.3 Results from the full F0002 geometry including the Cavitation model

To capture the transient movement of the vortex a simulation was run and instantaneous vector fields was extracted from the plane shown in figure 6.8 at every tenth time step to observe the transient behavior of the vortex. A series of vector fields for the quasi steady state solution are presented in figure 6.9. From the figures it is clear that the vortex core is moving from side to side inside the SAC volume. This oscillating behavior is very likely to cause the fluctuations in mass flow rates presented in section 6.2.1. The frequency for the movement of the vortex structure was found by manually counting the number of images between reoccurring structures. The frequency was determined by counting the number of images \((10 \cdot \Delta t)\) between four consecutive oscillations and using the mean value to determine the frequency. The frequency and the non-dimensional Strouhal number is shown in table 6.2.

<table>
<thead>
<tr>
<th>Frequency [Hz]</th>
<th>Strouhal number</th>
</tr>
</thead>
<tbody>
<tr>
<td>1081</td>
<td>0.019</td>
</tr>
</tbody>
</table>

The Strouhal numbers for the mass flow rates in table 6.1 and the Strouhal number for the vortex shedding in table ?? are all in the order of 0.02 and it can be concluded that the fluctuations observed in the mass flow measurements originate from the vortex and is not numerically induced as feared due to the mass imbalance problems discussed in section 5.4.

6.3 Results from the full F0002 geometry including the Cavitation model

As stated earlier in this report the strategy on how to utilize the cavitation model is to first obtain a fully developed flow field as this has proven to be important to numerically converge cavitation simulations. The very low time step needed to model the cavitation phenomenon also make it very time consuming to evolve the flow from being stagnant to a time independent solution. It was determined in chapter 4.1 that using a steady solver to obtain a fully developed flow field and then start the transient simulation including the cavitation model was the best strategy when modeling cavitation. Divergence problems for the steady solver during initial iterations, and the fact that the flow through the injector never becomes fully time independent this strategy was changed to first obtain the statistically time independent steady state solution with a transient solver and then apply the cavitation model. The simulation for cavitation presented was run without activating the cavitation model and using the same time step as in section
for approximately 300 $\tau$ and then the cavitation model was activated, the time step was changed to $\Delta t_{\text{cavitation}} = 1 \cdot 10^{-8}$ s and the convective scheme for the segregated flow solver was changed from second order to first order upwind in accordance with the model parameters developed in section 4.1. The very low time step for the simulation make the total process very time consuming and even after running parallel on 8 processors for more than 150 hours of CPU time making approximately 27 500 time steps the model had only developed for approximately $4.5 \cdot \tau$.

The mass flow through the nozzles and the total mass flow through the injector is shown in figure 6.10. Since the cavitation simulation is a restart of a simulation using the same settings as the simulation in section 6.2 the first entry in the figures correspond to the mass flow of the last time step for the model run without activating the cavitation model. Note that these values are not time averaged, but instantaneous mass flow rates placed somewhere arbitrary inside the oscillating flow rate. One cycle of the oscillation in figure 6.9 is calculated using the frequency in table 6.2 to be $\sim 16 \tau$ so the time development for the cavitation simulation have only run for approximately one third of the vortex cycle.

Not having run the cavitation model over several cycles for the oscillating vortex add some uncertainty to the change in mass flow rates due to cavitation, but is believed to be negligible since the difference between max and min values of the total mass flow in figure 6.1 is only 1% for one cycle. The
6.3 Results from the full F0002 geometry including the Cavitation model

![Graph showing percent of original mass flow for different nozzles.](image)

**Figure 6.11:** Change in mass flow \( \left( \frac{\dot{m}_{cav}}{\dot{m}_{no~cav}} \cdot 100 \right) \) when activating the cavitation model

The change in mass flow rates for the nozzle holes are presented in figure 6.13. The Mass flow rate through the nozzles is reduced which is in compliance with the behavior observed from the model developed in section 4.1.

![Volume of Fraction (VOF) images](image)

**Figure 6.12:** Volume of Fraction (VOF)

The cavitation fields are shown as Volume of Fractions (VOF) in figure 6.12. In accordance with the cavitation number of \( CN = 4.72 \) the nozzles experience supercavitation. Only the cells where \( \alpha_v > 0.1 \) are shown in the
figure. The cavitation is more or less equally distributed in the nozzles and there are no major differences in extent or intensity of the vapor clouds. To give a more detailed view on the cavitation field a cross section of nozzle 3 will be used to visualize the distribution of cavitation inside the nozzle.

Figure 6.13: Cross section view of nozzle 3

Figure 6.13 show the volume of fraction of vapor ($\alpha_v$) for nozzle 3. Just as the model developed in section 4.1 would for the equivalent cavitation number, $CN = 4.72$, the nozzle supercavitates and vapor stretches from the inlet of the nozzle and downstream to the outlet, but note that the intensity of the vapor is lower than for the model from section 4.1. Vapor dominated clouds are only present at the nozzle inlet and for the rest of the nozzle there is a mixture of liquid and vapor.

The bubble growth and collapse is shown in form of the scalar field function cavitation production rate shown in figure 6.14. The bubble growth is shown in figure 6.14a and it show that the vapor bubbles are formed at the nozzle entrance, where the pressure is lowest due to the sudden change in geometry. Figure 6.14b show where the bubbles collapse, and as expected the bubbles collapse when reaching the end of the nozzle where the pressure again rises above the vapor pressure. The negative production rate at the nozzle entrance is a result of a steep pressure gradient at the entrance where the pressure changes from being above the vapor pressure to under the vapor pressure. Note that there are no signs of vortex cavitation inside the SAC volume, the local pressure at the core of the vortices detected in this study is much higher than the vapor pressure and cavitation so the numerical model used in this study will not be able to model it.

The pressure distribution inside the SAC volume is shown in figure 6.15 and it clearly show that the pressure inside the SAC volume is much higher
6.3 Results from the full F0002 geometry including the Cavitation model

(a) Positive cavitation production rate  
(b) Negative production rate

Figure 6.14: Cavitation production rates $\frac{m^3}{s}$

Figure 6.15: Pressure distribution in the SAC volume

than the vapor pressure. Gevaises et al. [2] also confirms that their CFD model that is also based on pre existing spherical bubble nuclei that that grow by use of the Rayleigh-Plesset equation is unable to predict string cavitation and can only predict geometric-induced hole cavitation formed at the nozzle entrance. Further they state: "Formation of vortex cavitation as a result of the pressure drop within the core of the vortex has been abandoned as a mechanism on the basis of model predictions which indicate pressure levels much higher than the vapor pressure of the flowing liquid" (Gevaises et al. Vortex flow and cavitation in diesel injector nozzles, Journal of Fluid Mechanics vol 610, 2008 page 209)[2].
6.4 Results from the Full F0002 geometry without activating the cavitation model using a transient pressure input boundary condition

The results presented so far in this part of the report is based on constant boundary conditions. A transient pressure signal has been collected from the PhD thesis of Bjarke Dam [5] to investigate the behavior of the nozzle when the pressure rises during a fuel injection cycle at operating conditions. As for all other simulations the needle is fully open at all time and the outlet pressure boundary is constant at 140 bar. The results presented here will therefore not include effect of needle movement, but is still interesting since it will show the transient behavior for mass flow through the individual nozzles during an injection cycle at operating conditions.

(a) The original Pressure signal (Dam, 2007) (b) The time corrected pressure signal used in the simulations

Figure 6.16: The transient pressure signal used in the simulations

Figure 6.16a show the original pressure signal collected from the PhD thesis of Bjarke Dam [5]. Figure 6.16b show the signal used as input for the inlet boundary condition in this simulation. As stated earlier the valve shortly upstream of the inlet to the computational domain opens at 350 bar and no fuel passes through the injector before this point. The pressure signal used as input in the simulation starts at 350 bar and ends at the point where the fuel pump closes and the pressure starts to decay. The pressure input is digitalized with a software called Digitizer and the time input is corrected so that $t_0 = 0$ as shown in figure 6.16b. The transient pressure signal was loaded into STAR-CCM+ as a table boundary condition and STAR-CCM+ performed the interpolation to fit accordingly with the time step $\Delta t = 5 \cdot 10^{-6}$ s as for the simulations using a constant input pressure boundary condition.
6.4 Results from the Full F0002 geometry without activating the cavitation model using a transient pressure input boundary condition

The mass flow rate through the nozzles are shown in figure 6.17. Note that the non-dimensional time $\tau$ in this plot is not equal to the $\tau$ used for the constant pressure boundary simulations. $\tau$ is determined by equation (5.2) but the time scale $t^*$ is different since the axial velocity entering the SAC volume is no longer independent of time. The axial velocity entering the SAC volume $\bar{U}_{SAC}$ from equation 5.1 is extracted from the plane called Inlet Nozzle SAC volume in figure 5.2 and time averaged over the full cycle. The non-dimensional time is then calculated using equation (5.2)

$$\tau = \frac{t_{\text{transient profile}}}{t_{\text{transient profile}}}$$

The distribution of mass flow is the same as observed for the simulation using a stationary pressure input. Nozzle 2 has the highest mass flow rate, followed closely by nozzle 1. Nozzle 3 experiences the largest fluctuations and the mass flow through nozzle 4 is still lower than the other nozzles. This indicates that the oscillating vortex discovered in the previous section is present when using a transient pressure profile. The flow is developing for the first 75-120 $\tau$ before the vortex develops and start to oscillate. The frequency for the oscillations are found by manually counting peaks through the domain and the strouhal number is calculated based on the Bernoulli
velocity scale $U_b$ and the diameter of the SAC volume. The Bernoulli velocity scale is calculated with equation (1.2) and is based on the time average injection pressure $\bar{p}_{\text{inj}}$ from figure 6.19.

Table 6.3: Frequencies and Strouhal numbers for the nozzles for the simulation using a transient pressure input boundary

<table>
<thead>
<tr>
<th>Nozzle</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>Total</th>
</tr>
</thead>
<tbody>
<tr>
<td>Frequency [Hz]</td>
<td>1271</td>
<td>1379</td>
<td>1327</td>
<td>2673</td>
<td>1369</td>
</tr>
<tr>
<td>St</td>
<td>0.0234</td>
<td>0.0254</td>
<td>0.0244</td>
<td>0.0492</td>
<td>0.0252</td>
</tr>
</tbody>
</table>

The frequencies and Strouhal numbers found for the transient simulation is shown in table 6.3. Comparing the data in table 6.3 with the frequency and Strouhal numbers found for the simulation using a constant pressure boundary in table 6.1 show that they are equal and the oscillation present in the results from this simulation can be attributed to the same vortex structure as the one discovered in the simulations using a constant pressure boundary for the inlet.

![Flow coefficients for the transient simulation](image)

**Figure 6.18:** Flow coefficients for the transient simulation

The nozzle discharge coefficients $C_d$ and momentum coefficients $C_m$ have been calculated for the simulation based on the Bernoulli velocity scale $U_b$ and the time average mass flow for the total injection cycle. The Discharge coefficient and momentum coefficient is shown in figure 6.18.
6.4 Results from the Full F0002 geometry without activating the cavitation model using a transient pressure input boundary condition

6.4.1 Estimation of Cavitation inception

As stated in section 6.1 it is considered too computationally demanding to run a full injection cycle when the cavitation model is activated. This is also confirmed by observing the time span for the simulation where the cavitation model was activated in section 6.3. Running the simulation for over approximately 30,000 time steps only resulted in a time evolution of approximately 4.5 \( \tau \). To give an estimation of cavitation inception for real operating conditions for the injector probes was placed at the nozzle entrance, where the cavitation bubbles are formed, to give an indication of when cavitation occurs. The location of pressure probes are shown in figure 6.19b. The probes are placed somewhat arbitrary so the results can not be fully trusted, but it gives a reasonable estimation of cavitation inception. The probes measure the pressure in these locations and is plotted against the time evolution to see when the local pressure at the entrance drops below the vapor pressure for the liquid. Figure 6.19a show the time evolution for the local pressure at the entrance for the nozzles. From the figure it is clear that cavitation occurs almost instantaneous after the simulation starts. When comparing with the evolution of the mass flow in figure 6.17 it is clear that the flow develops fast after initialization and the nozzles can be expected cavitate within the first 5-15 \( \tau \) \((0.4 - 0.8 \text{ ms})\). Due to the inaccurate placement of the probes further investigation on cavitation inception in terms of which nozzle cavitation starts first will be mere speculations.

![Figure 6.19: Local pressure and placement of probes](image)
Chapter 7

Discussion and Conclusion

7.1 Discussion

In this study a numerical CFD model for simulating nozzle flow and modeling cavitation have been developed, tuned against experimental data from the literature and applied to a real life application to give an initial estimation on the cavitation regimes present in fuel injectors at real life operating conditions. When this master thesis was started the objective was to develop a cavitation CFD model parallel to the experimental work performed for the master thesis project of Katrin Herkert at MAN Diesel & Turbo and then use the experimental data to validate the CFD model. Unfortunately, the experimental data from the cavitation rig at MAN Diesel & was not available for use in the CFD model before long into the project and it was too late to use them for model tuning and validation. In an ideal world, the experimental work should precede the numerical development, but this was unfortunately not the case. Instead the objective for the report was changed to develop a model based on the work made by Winklhofer et al [7] and then apply the model to the F0002 fuel injector provided by MAN Diesel & Turbo. A consequence of this is that the model developed in this study is not properly verified against experimental data and as a result of this the results obtained with this model can only be used as predictions for cavitation and the upstream nozzle flow in fuel injectors, and as inspiration for later work where experimental work for validation is available. A discussion of the development phase of the cavitation parameters have already been conducted in section 4.7 and the remainder of this discussion will focus on the simulations for the F0002 fuel injector.
CAD data for a full scale fuel injector was made available from MAN Diesel & Turbo and the cavitation model was used to perform flow simulations upstream the nozzle spray. Experimental data was not available for this part either so the results presented for the F0002 fuel injector is not viable for design recommendations. This is, as far as the author knows, the first numerical investigation of the flow conditions upstream the spray for fuel injectors like the F0002 performed at both MAN Diesel & Turbo and MEK-DTU. Consequently there was very many uncertain factors and parameters to deal with, and almost the entire project have been based on a trial and error approach.

The strategy was to develop a computational grid that produced consistent results for several grid sizes and to perform a "validation" against itself in terms of mass flow rates for different grid sizes. There where some numerical problems with converging the continuity equation and several approaches was used to reduce this factor of uncertainty. A very large grid, consisting of over one million cells, showed signs of improvements, but it was decided that the computational costs and time consume involved with a grid of this size would limit the development for the project significantly. A compromise to accept a level of numerical error was made to assure progress in the project. The area of interest in the F0002 fuel injector is the SAC volume and the four nozzles. Attempts was made to develop a grid that produced results consistent with the full geometry model. This was proven to be difficult as the mass flow rates for the two models did not coincide. The small SAC volume model was intended to include the SAC geometry starting from the plane called \textit{Inlet SAC volume} in figure 5.2 and use values for pressure and turbulence specification directly from the full geometry model. The model did display similar physics as the full geometry and simulations showed the presence of a vortex structure similar to the one observed in the full geometry. Unfortunately, due to time limitations in this project the work on this smaller SAC volume was terminated and the model including the total geometry was used for the study.

The simulations for the full F0002 geometry, as stated earlier, showed the presence of a large vortex structure in the SAC volume and consistent vortex shedding frequencies was collected and analyzed. The presence of this structure is consistent with numerical and experimental work performed on similar nozzle geometries. The very low time step needed by the cavitation model made investigation of the cavitation in the F0002 injector difficult and the results presented in this study can only be used as an indication of the cavitation phenomenons present in the fuel injector. There was no sign of string cavitation in the simulations, but the results in this study can not confirm or disconfirm presence of this phenomenon as earlier studies on similar geometries \cite{2} show that string cavitation form inside the nozzle even
when the pressure exceeds the vapor pressure.

There have not been performed a study on scaling of cavitation in this project. The strategy was to determine cavitation parameters in the Winklhofer nozzle and apply them directly to the F0002 injector. As mentioned in chapter 4.1 the cavitation parameters presented in the literature did not provide acceptable results for the Winklhofer nozzle and it is possible that adjustments is necessary when applying the cavitation model to a much larger domain which is the case for the F0002 atomizer. This was not investigated in this study.

7.2 Conclusion and future studies

In this report a numerical model capable of reproducing cavitating flow regimes from experimental data have been developed. A long an time demanding process of parameter tuning, model selections and design decisions had to be made before the numerical model and the experimental data from Winklhofer [7] coincided. The model development was based on a supercavitating regime and the tuning process was based largely on comparison of images of cavitation regimes and mass flow rates. The model parameters proposed in this study yielded good results and the cavitation field and mass flow rates over a series of pressure differences did coincide to an acceptable level.

When the CFD model was concluded to perform at an acceptable level, the model was applied to a real life application. The F0002 fuel injector used in MAN Diesel & Turbo’s Two stroke Diesel engines to give an estimate of the cavitation regime during real life operating conditions. The flow fields inside the injector’s SAC volume was investigated and the identification of a oscillating vortex structure is consistent with suggestions from the literature and previous work on similar geometries. Within the scope of this study this vortex was only identified in terms of fluctuations in the mass flow rates and visualizations by vector plots and streamlines. A detailed analysis of the vortex and coupling of upstream effects and this vortex was not performed. The absence of numerical and experimental data for the F0002 injector make it difficult to draw more detailed conclusions regarding this vortex than just identify it. This study includes investigations using constant pressure boundaries for the inlet and outlet and a transient pressure signal corresponding to a full injection cycle. Comparison between these two simulations revealed that a transient increase in pressure did not significantly affect the flow conditions in the nozzle as the vortex shedding frequency was determined to be equal for both cases. This indicates that
numerical and experimental investigations can be performed using constant flow settings to investigate the physics present inside the fuel injector. This will, in the authors opinion, make it easier to build up a experimental setup for a fuel nozzle to investigate the formation of string cavitation since this has proven to be very difficult using numerical simulations alone.

As stated earlier there was an other master thesis project performed at MAN Diesel & Turbo parallel to this study. This master thesis was conducted by Katrin Herkert and focused on building up a test rig for cavitation. Unfortunately it was impossible to couple these projects together due to the time involved with establishing experimental data. A recommendation for future projects would be to use the model developed in this study to compare cavitation regimes with the experimental data. Experimental data for the F0002 geometry is also needed in order to verify the results from this study, since the flow fields and cavitation regimes presented in this study can only be used as indications and identifications of the flow inside the SAC volume. As stated in the discussion, a smaller geometry consisting only of the SAC volume is desirable since more cells can be used in the SAC volume and the nozzles. Future numerical studies on the F0002 nozzle should focus on developing a smaller model.

7.2.1 Acknowledgements

I would like to thank my co-supervisors at MAN Diesel & Turbo, Simon Matlok and Stefan Meyer, for allowing me to use CAD data for their F0002 fuel injector in my project, as this has made my project much more interesting and challenging. I would especially like to thank Simon Matlok for being available for questions and guidance throughout the project. I would also like to thank Kristian Mark Ingvorsen for helpful guidance and feedback throughout the project.


A.1 Turbulence formulation

A.1.1 Two-Layer formulation

The formulation in section 3.6.1 is purely based on the standard k-Epsilon model as it is the basis for the turbulence modeling. When dealing with complex geometries and flow fields like the ones involved with injection nozzle flow it is difficult to find a turbulence model and subsequent wall treatment method that provides good results since there are large differences in the $y^+$ values in different parts of the computational grid. The solution for this project was to employ a turbulence model that was able to handle regions where the viscous sublayer is fully resolved and couple it with regions where wall-functions are used $y^+ > 30$. The two layer model blends a one-equation model, which solves for $k$ but prescribes $\varepsilon$ algebraically with distance from the wall, with the two-equation k-epsilon model. The two layer model is parameterized as a length scale function

$$l_\varepsilon = f(y, Re_y) \quad (A.1)$$

where $y$ is distance from the wall and a turbulent viscosity ratio function

$$\frac{\mu_t}{\mu} = f(Re_y) \quad (A.2)$$

where both the length scale and the viscosity is functions of the turbulent Reynolds number $Re_y$ given by the following equation

$$Re_y = \frac{\sqrt{k}y}{v} \quad (A.3)$$
The turbulent dissipation rate computed from the two layer model is

$$\varepsilon = \frac{k^2}{l_\varepsilon} \quad (A.4)$$

The following blending function is used to combine the two layer formulation with the full two equation model:

$$\lambda = \frac{1}{2} \left[ 1 + \tanh \left( \frac{Re_y - Re_y^*}{A} \right) \right] \quad (A.5)$$

where $Re_y^*$ defines the limit of the applicability for the two layer formulation, the value is set by default in STAR-CCM+ to be $Re_y^* = 60$. The constant $A$ determines the width of the blending function, it is defined to be such a value that the blending function $\lambda$ is within 1% of its far field value at a given variation of $Re_y - Re_y^*$. The formulation is then

$$A = \frac{|Re_y - Re_y^*|}{\tanh 0.98} \quad (A.6)$$

The value for $Re_y - Re_y^*$, denoted $\Delta Re_y$ in the user guide [1], is a user defined input with a default value of 10. The turbulent viscosity $\mu_t$ from the k-epsilon model, equation (3.47), is blended with the two layer value as follows

$$\mu_t = \lambda \mu_t \bigg|_{k-\varepsilon} + (1 - \lambda)\mu \left( \frac{\mu_t}{\mu} \right)_{two-layer} \quad (A.7)$$

Since cell values of $\varepsilon$ may be specified by setting the coefficients of the solution matrix to satisfy

$$\frac{a_p}{\omega} \Delta \varepsilon_p = a_p(\varepsilon_p^{n+1} |_{specified} - \varepsilon_p^n)$$

the discretized transport equation for $\varepsilon$ (3.44) may be combined with this to obtain:

$$\frac{a_p}{\omega} \Delta \varepsilon_p + \sum_n a_n \lambda \Delta \varepsilon_n = \lambda \left( b - a_p \varepsilon_p^n - \sum_n a_n \varepsilon_p^n \right) + (1 - \lambda) a_p(\varepsilon_p^{n+1} |_{two-layer} - \varepsilon_p^n) \quad (A.8)$$

The above formulation denotes how the two layer equations are implemented and blended with the standard k-Epsilon formulation. The actual equations corresponding to equation (A.1) and (A.2) for the length scale and the viscosity ratio will be presented in the remaining part of this formulation. There are two versions of the two-layer model in STAR-CCM+, one for shear driven flows and one buoyancy driven. The shear driven type are the
one applied in this study, so the buoyancy formulation will not be presented. The length scale function $l_\varepsilon$ is calculated as

$$l_\varepsilon = c_l y \left[ 1 - \exp \left( - \frac{Re_y}{A_\varepsilon} \right) \right]$$  \hspace{1cm} (A.9)

with the following relation between the constants $A_\varepsilon$ and $c_l$

$$A_\varepsilon = 2c_l$$  \hspace{1cm} (A.10)

$$c_l = \kappa C_\mu^{-\frac{3}{4}}$$  \hspace{1cm} (A.11)

where $C_\mu = 0.09$ and $\kappa = 0.42$. The turbulent viscosity is calculated by use of the following expression

$$\mu_t \mu = Re_y C_\mu^{\frac{1}{2}} \left[ 1 - \exp \left( - \frac{Re_y}{A_\mu} \right) \right]$$  \hspace{1cm} (A.12)

where $A_\mu = 70$.

### A.1.2 Wall treatment

Wall treatment is a set of modeling assumption applied to the near wall region of the turbulent flow. It is assumed that the turbulence models are only valid outside the viscous dominated region of the boundary layer so that a consistent set of assumptions are made regarding the distribution of velocity, turbulence and other scalar quantities. In general one can divide the wall treatment in to three, the high $y^+$ wall treatment who is the "classic" wall treatment approach where it is assumed that the $y^+$ values are within the logarithmic region ($30 < y^+ < 300$) and all values are modeled from turbulent boundary layer theory. The second treatment is the low $y^+$ wall treatment where it is assumed that the viscous sublayer ($y^+ \sim 1$) is fully resolved and no modeling via wall laws are necessary. The third approach is called the all $y^+$ wall treatment and is basically a compromise between the two latter treatments and seeks to recover the behavior of the two other treatments in the limit of very fine or coarse meshes. It produces good results over the entire range of $y^+$ values, even in the intermediate buffer layer.

First a short formulation of the wall laws and the non-dimensional quantities will be presented followed by a short formulation of the wall treatment used in this study, namely the two-layer all $y+$ wall treatment. First the non dimensional value

$$y^+ = \frac{yu^+}{v}$$  \hspace{1cm} (A.13)
that can be considered the wall distance, where \( y \) is the distance from the cell center to the nearest wall and \( \upsilon \) is the local kinematic viscosity. The reference velocity \( u^* \) is called the wall friction velocity and is often related to the wall shear stress like \( u^* = \sqrt{\frac{\tau_w}{\rho}} \approx \sqrt{\frac{\nu u}{y}} \) \cite{19}, but as will be shown is often defined differently depending on the particular turbulence model\cite{1}. The non-dimensional velocity parallel to the wall is

\[
u^+ = \frac{u}{u^*} \tag{A.14}\]

The wall laws are set up to calculate the value of \( u^+ \), and its first derivative, as a function of \( y^+ \). In the laminar viscous sublayer the \( u^+ \) value is assumed to be linear with \( y^+ \) as

\[
u_{+\text{lam}}^+ = y^+ \tag{A.15}\]

The \( u^+ \) value in the turbulent logarithmic layer are calculated by the wall log-law as

\[
u_{+\text{turb}}^+ = \frac{1}{\kappa} \ln(E y^+) \tag{A.16}\]

Where \( E \) is the wall function constant, who can be used to incorporate wall roughness, and \( \kappa \) who is the Von Karmann constant. The default values for these parameters are \( E = 9.0 \) and \( \kappa = 0.42 \). The blended wall law is intended to represent the buffer layer by approximately blending the viscous sublayer and the logarithmic regions.

\[
u_{+\text{blended}}^+ = \frac{1}{\kappa} \ln(1 + \kappa y^+) + C \left[ 1 - \exp \left( -\frac{y^+}{D} \right) - \frac{y^+}{D} \exp(-b y^+) \right] \tag{A.17}\]

Where

\[D = y_{+\text{m}}\]

and \( y_{+\text{m}}^+ \) is the intersection distance between the viscous and fully turbulent regions, this value is found by Newton iteration. Further

\[C = \frac{1}{\kappa} \ln \left( \frac{E}{\kappa} \right)\]

and

\[b = \frac{1}{2} \left( \frac{D \kappa}{C} + \frac{1}{D} \right)\]

The above equations are the general formulation of the wall laws. As stated earlier the complex flow fields and geometry of an fuel nozzle leads make it difficult to produce a grid that fits the profile of an low \( y^+ \) or a high \( y^+ \) since velocities and grid sizes, and subsequently the \( y^+ \) value, is varying a
lot throughout the geometry. The wall treatment chosen for this study is therefore the two-layer all $y^+$ wall treatment. This formulation is identical to the standard all $y^+$ wall treatment, but contains a wall boundary condition for the dissipation rate $\varepsilon$ to be consistent with the two-layer formulation. For the k-Epsilon turbulence models the wall treatment performs the following functions:

- Specifies the reference velocity $u^*$ to be used in the wall laws (A.15) (A.16) (A.17)
- Computes a special value of turbulent production $G_k$ (3.45)
- Computes a special value of turbulent dissipation $\varepsilon$ in the wall cell

First, a blending formulation for the all $y^+$ wall treatment is defined as a function of the turbulent Reynolds number from equation (A.3)

$$g = \exp\left(-\frac{Re_y}{11}\right)$$

(A.18)

The reference velocity $u^*$ is given by the following equation

$$u^* = \sqrt{g \frac{\nu u}{y} + (1 - g)C_{1/2} \mu k}$$

(A.19)

The wall-cell turbulent production $G_k$ is calculated by

$$G_k = \mu \nu S^2 + (1 - g) \frac{1}{\mu} \left( \rho u^* \frac{u}{u^+} \right)^2 \frac{\partial u^+}{\partial y^+}$$

(A.20)

In accordance with the two layer formulation the turbulent dissipation rate $\varepsilon$ is not modeled, but solved through the entire turbulent boundary layer using equation (A.4) repeated here for convenience

$$\varepsilon = \frac{k^2}{\varepsilon}$$

A.2 Numerical aspects for the cavitation source term

The following section is based on the work of J Sauer [16] and E. Giannadakis [10]. The description of the cavitation model in section 3.5 is as far as the STAR-CCM+ user guide [1] goes. When contacting the CD-Adapco support service they refer to the PhD thesis of Jürgen Sauer for a more detailed description on how the cavitation model is implemented in STAR-CCM+. J. Sauer writes in his PhD study that the cavitation source term, $S_{\text{cav}}$, in equation (3.32) needs a modification to avoid divergence in the SIMPLE
algorithm. The term $S_{cav}$ is an average value of the previous and the current time step since the bubble radius $R$ is from the previous time step while the term $\frac{dR}{dt}$ is from the current time step. This constitutes a mixture of explicit and pseudo implicit approach with respect to the time treatment for the source term\cite{10} while the treatment for the pressure correction is only pseudo implicitly. Treatment for the cavitation source term used by the pressure correction equation is therefore necessary because the pressure correction estimations in the inner iterations of the SIMPLE algorithm leads to divergence since it overcompensates pressure corrections e.g. where the pressure is below the vapor pressure. J. Sauer \cite{16} proposed to include the partial derivative of the cavitation source term with respect to the pressure correction of the current integration step to improve its implicitness and numerical stability.

\[
\frac{\partial S_{cav}^m}{\partial p} = \frac{n_0}{1 + n_0 \frac{4}{3} \pi R^3} \cdot 4\pi R^2 \frac{\partial}{\partial p} \left( \frac{dR}{dt} \right) \tag{A.21}
\]

where

\[
\frac{\partial}{\partial p} \left( \frac{dR}{dt} \right) = \begin{cases} 
\frac{\partial}{\partial p} \left( \sqrt{\frac{2}{3} \frac{p_v - p_\infty}{\rho_l}} \right) = \frac{1}{\sqrt{\frac{2}{3} \frac{p_v - p_\infty}{\rho_l}}} \cdot \frac{2}{3\rho_l} (-1) & \text{for growth} \\
\frac{\partial}{\partial p} \left( \sqrt{\frac{2}{3} \frac{p_\infty - p_v}{\rho_l}} \right) = \frac{1}{\sqrt{\frac{2}{3} \frac{p_\infty - p_v}{\rho_l}}} \cdot \frac{2}{3\rho_l} (+1) & \text{for collapse} 
\end{cases} \tag{A.22}
\]

or more general like

\[
\frac{\partial}{\partial p} \left( \frac{dR}{dt} \right) = \frac{-2}{3\rho_l} \frac{1}{\sqrt{\frac{2}{3} \rho_l |p_v - p_\infty|}} \tag{A.23}
\]

including the above equations into the cavitation source term for the pressure correction the source term becomes

\[
S_{cav}^m = S_{cav}^{m*} + \frac{\partial S_{cav}^{m*}}{\partial p} (p_p^m - p_p^{m*}) \tag{A.24}
\]

Where $m$ is the iteration step for the SIMPLE algorithm.