Numerical simulation of a cavitating jet

Author:
Thijs Schouten

Thesis Committee:
Prof. Dr. ir. C. van Rhee
Dr. ir. G.H. Keetels
Dr. Ir. H.J. de Koning Gans
Dr. ir. A. Nobel
Abstract

In the dredging industry jets are often used to excavate sediments. When jets are applied for cohesive soils high jet pressures are needed to achieve high penetration and production. When jets with high pressures end high exit velocities are used under water and the ambient pressure is not too high the jet will start to cavitate. This occurrence of cavitation has a great influence on the performance of the jet.

Numerous experimental tests regarding cavitating jets have been executed Yahiro et al. (1974), Soyama et al. (1996), Ooi (1985), Yanaida et al. (1985) and Nobel (2013). In these tests several characteristics such as the length of the cavitation cone and the stagnation pressure at multiple points in the jet have been measured and observed.

Beside these experimental tests some efforts have been made in modeling the cavitating jet. The modeling of a cavitating jet is very complex. At the interface of the jet and the ambient water a zone of incompressible water and unsteady compressible vapor bubbles exist. Because of this complexity no general (numeric) code for a fully developed cavitating jet can be found in the literature. The goal of this study is to investigate the mechanisms driving the cavitation and the effect of cavitation on the water jet with a CFD (computational fluid dynamics) software package such as OpenFOAM.

Within OpenFOAM 2 solvers were found to be capable of solving for cavitating flows. Due to the fact that dissolved air content and nuclei diameter is an important parameter the interPhaseChangeFoam solver with the Schnerr-Sauer model was used to perform the simulations.

The first set of simulations where performed with a Reynolds-Average Navier-Stokes method and was based on the assumption that the pressure drop induced by the acceleration of the ambient fluid caused cavitation. It was seen that with this method with multiple different settings and geometries the cavitation cone could not be simulated. This indicates that the pressure drop due to acceleration of the fluid on its own is not enough to induce cavitation. This leads to the conclusion that RANS methods within OpenFOAM are incapable of solving for a cavitating jet.
The second set of simulations were performed with the assumption that the cavitation develops within the vortices produced by the shear between the ambient fluid and the jet fluid. In order to capture these vortices the second set of calculations were performed with a Large Eddy Simulation (LES) method. The results of these simulations showed cavitation bubbles present around the jet. These bubbles start to develop within the core of the vortices and affect the flow of the jet. The presence of the cavitation bubbles hinders the exchange of momentum with the ambient fluid resulting in a longer potential core. The longer potential core gives rise to the stagnation pressures of the jet.

To support this observation the stagnation pressures of the different simulations were recorded. It could be observed from these measurements that in case of a central LES simulation the stagnation pressure at a distance of 12 Dn was found to follow the same trends as observed in experimental test.

This leads to the conclusion that it is possible to simulate cavitating phenomena that occur within a cavitating jet with the interPhaseChangeFoam solver of OpenFOAM and an LES simulation technique.
Nomenclature

\( \alpha_{ent} \)  | entrainment coefficient  
\( \alpha_l \)  | fraction of liquid  
\( \alpha_v \)  | fraction of vapor  
\( \chi \)  | modification factor  
\( \eta \)  | constant  
\( \mu_n \)  | discharge coefficient  
\( \mu_t \)  | eddy viscosity  
\( \nu_l \)  | kinematic viscosity liquid phase  
\( \rho \)  | density  
\( \rho_l \)  | density liquid fraction  
\( \rho_m \)  | mixture density  
\( \rho_v \)  | density vapor fraction  
\( \rho_w \)  | water density  
\( c \)  | sonic speed  
\( C_\mu \)  | constant  
\( c_{\rho} \)  | ratio of liquid density and gas density  
\( D_n \)  | nozzle diameter  
\( I_0 \)  | flow rate  
\( k_1 \)  | constant  
\( k_2 \)  | constant  
\( n_b \)  | number of nuclei  
\( P \)  | static pressure  
\( P_\infty \)  | Pressure infinitely far form bubble  
\( P_b \)  | Pressure inside the bubble  
\( P_r \)  | ambient pressure  
\( P_v \)  | vapor pressure  
\( Q_0 \)  | Momentum flux  
\( r \)  | radial distance  
\( R_b \)  | bubble radius  
\( S \)  | surface tension  
\( s \)  | distance from the nozzle  
\( U_0 \)  | jet outlet velocity  
\( U_m \)  | constant  
\( V \)  | velocity in y-direction for plane jet  

\( v \)
# Contents

Abstract ii

Nomenclature v

1 Introduction 1

2 Jets and cavitation 3
  2.1 Cavitation 3
  2.2 Single phase turbulent jets 4
    2.2.1 Entrainment: fully developed region 6
    2.2.2 Entrainment: development region 7
  2.3 The cavitating jet 8
    2.3.1 Cavitation models 10
    2.3.2 Schnerr and Sauer (2001) 11
  2.4 Overview CFD 13
    2.4.1 Peng et al. (2013) 13

3 Models and discretization 17
  3.1 discretization 17
    3.1.1 The finite volume discretization 18
    3.1.2 The Pressure-Correction Method 19
    3.1.3 OpenFOAM solvers 20
  3.2 Geometries 21
    3.2.1 Single phase turbulent jet model 21
    3.2.2 Cavitating jet geometry 1 23
    3.2.3 Cavitating jet geometry 2 24
    3.2.4 Cavitating jet geometry 3 26

4 Single phase turbulent jet 29
  4.1 RANS 29
    4.1.1 Turbulence 31
4.2 Results .................................................. 32
  4.2.1 Velocity ........................................... 32
  4.2.2 Mesh ............................................... 34
  4.2.3 Entrainment ....................................... 35
4.3 Conclusion of turbulent jet .......................... 35

5 Cavitating jet .............................................. 37
  5.1 RANS simulations .................................... 38
    5.1.1 Case 1: 5mm inlet tube ........................ 38
    5.1.2 Case 2: 5mm inlet tube 3D ..................... 39
    5.1.3 Case 3: 5mm inlet tube 3D low pressure domain... 40
    5.1.4 Case 4: Peng nozzle standard interphase ......... 41
    5.1.5 Case 5: Peng nozzle central interphase ......... 42
    5.1.6 Case 6: Peng nozzle adjusted RNG $k - \epsilon$ central... 43
    5.1.7 Discussion ....................................... 44
  5.2 LES simulations ...................................... 46
    5.2.1 Case 1: standard case, coarse mesh .......... 46
    5.2.2 Case 2: high ambient pressure ................. 47
    5.2.3 case 3: Standard case fine mesh ............... 48
    5.2.4 case 4 & 5: standard case blended ............. 49
    5.2.5 Case 6: standard case central .................. 51
    5.2.6 Effect of cavitation on the jet ................. 53
    5.2.7 Comparison with experiments .................... 55
    5.2.8 Discussion ....................................... 56

6 Conclusion ................................................. 57

Bibliography ................................................. 61

List of Figures ........................................... 64

List of Tables ........................................... 65

Appendix A: Entrainment .................................. 67

Appendix B: Adjustment turbulence model ................. 69

Appendix C: Stagnation pressure calculation ............... 73
Chapter 1

Introduction

In the dredging industry jets are often used to excavate sediments. When jets are applied for cohesive soils high jet pressures are needed to achieve high penetration and production. When jets with high pressures end high exit velocities are used under water and the ambient pressure is not too high the jet will start to cavitate. This occurrence of cavitation has a great influence on the performance of the jet.

If the jet starts to cavitate a protective layer of vapor bubbles is formed around the jet. The formation of this layer can be explained as follows: at the interface of the jet a momentum exchange between jet and ambient fluid takes place. The ambient water is entrained into the jet. As a result of this entrainment the static ambient pressure can drop below the vapor pressure. If the pressure drops under the vapor pressure vapor bubbles will be created which form a protective layer. As a result of this layer less ambient water is entrained into the jet and the exchange of momentum is reduced. This reduction leads to an increase of the stagnation pressure and the flow velocity of the cavitating jet in comparison to a non-cavitating jet.

Numerous experimental tests regarding cavitating jets have been executed Yahiro et al. (1974), Soyama et al. (1996), Ooi (1985), Yanaida et al. (1985) and Nobel (2013). In these test several characteristics such as the length of the cavition cone and the stagnation pressure at multiple points in the jet have been measured and observed. In the work of Nobel the stagnation pressure is measured with varying ambient and jet pressures. Also different nozzle shapes were used.

To fully understand the cavitating jet and cavitation behavior in general researchers deal with modeling of the cavitation. Modeling of a cavitating jet is very complex. At the interface of the jet and the ambient water a zone of incompressible water and unsteady compressible vapor bubbles exist. Because of this complexity no general (numeric) code for a fully developed cavitating
jet can be found in the literature. However some attempts have been made to couple the fluid dynamics equations to various cavitation models.

**Objective**

Since there is no general numeric code available for a fully developed cavitating jet the objective of this thesis is to model a cavitating jet with an existing CFD package such as OpenFOAM, to study the underlying mechanisms that cause the jet to cavitate and study the effects of cavitation on the jet.

**Contents**

In chapter 2 existing literature on jets and cavitation will be discussed.

In chapter 3 the focus will be on the methods used to solve the cavitating jet with a CFD software package. The second part of this chapter will focus on the geometries used in this thesis calculations.

In chapter 4 the results of the single phase turbulent jet will be discussed and compared to experimental measurements.

In chapter 5 the results for different calculations of the cavitating turbulent jet are shown.

In chapter 6 conclusions are drawn from the results.
Chapter 2
Jets and cavitation

In this chapter existing literature on cavitation, single phase turbulent jets, cavitating jets and the cavitating jet in CFD is discussed. Several books were written on the subject of cavitation, for example (Franc and Michel, 2006). In the first section it is explained what cavitation is and what causes cavitation. The second section is about the single phase turbulent jet. This jet is the base for a cavitating jet. The third section handles the cavitating jet and the models that exist in OpenFOAM to simulate cavitation. The last section handles previous attempts to simulate the cavitating jet.

2.1 Cavitation

To fully understand what drives the cavitating jet the phenomena of cavitation needs to be explained and a thoroughly understanding of the phenomena needs to be in place. In this section the phenomena of cavitation will be discussed as well as governing processes and equations.

If water is boiling at sea level it is evaporating at a temperature of approximately one hundred degrees Celsius. But if water is boiling at the top of the mount Everest the temperature of the water is a merely 70 degrees Celsius. The drop in boiling temperature is caused by the drop in ambient pressure. If the ambient pressure is even lower the water will start boiling/evaporating at a temperature of 20 degrees Celsius. In fluid flows, for example past immersed bodies, it sometimes happens that the pressure drops to a significant level. At this moment the water will evaporate without adding heat to the ambient water. This is called cavitation.

To describe the above story in a more physical manner it is to say that at a certain temperature a fluid has a related evaporation pressure. If the fluid is at this temperature and the pressure drops below the evaporation
pressure the fluid will evaporate. Thus cavitation inception is expected if

\[ P < P_v \]  \hspace{1cm} (2.1)

Where \( P \) is the pressure of the fluid and \( P_v \) is the vapor pressure of the fluid. To give an indication of the sensitivity of cavitation a dimensionless number, the so called cavitation number, is defined as follows

\[ \sigma = \frac{P_r - P_v}{\frac{1}{2} \rho v^2} \]  \hspace{1cm} (2.2)

Where \( P_r \) is the ambient pressure of the fluid, \( \rho \) is the density of the fluid and \( v \) is the velocity of the fluid.

There are two main types of cavitation, cavitation around a submersed body (for example a hydrofoil depicted in Figure 2.1a) in which the cavitation develops due to under pressures generated by the flow around the body. The other type is shear cavitation (Figure 2.1b) where the cavitation develops in the shear layer between a high speed and a stagnant flow region. The second type of cavitation is of interest since that governs the cavitating jet.

2.2 Single phase turbulent jets

At a certain flow velocity a turbulent jet starts to cavitate. The cavitation will develop within the shear zone of the turbulent jet. Therefore it is important to describe the characteristics of a turbulent jet before cavitation sets in. The turbulent free jet is schematized in Figure 2.2.

The flow of the jet is divided into multiple regions. At the inflow there is a velocity profile that is usually assumed to be uniform. Directly after the
2.2. SINGLE PHASE TURBULENT JETS

Figure 2.2: Turbulent jet cross section

jet leaves the nozzle it interacts with the ambient fluid. The water of the jet starts mixing with the ambient fluid in the turbulent mixing layer. Here transfer of mass and momentum takes place and ambient fluid is entrained and accelerated into the jet. The jet core which isn’t directly influenced by the turbulent mixing layer the velocity inside this core remains the same as the nozzle outlet velocity until the core vanishes. This core is called the potential core. After a distance of a few nozzle diameters the mixing layer has penetrated to the center of the jet and the potential core does no longer exist. After this point the flow is a fully developed turbulent flow.

In the region of the fully developed flow the velocity profile is a Gaussian profile. For a circular jet flow this results in the following equation.

\[
U(s, r) = \sqrt{\frac{k_1}{2} \frac{U_0 D_n}{s} e^{-k_2 \frac{r^2}{s^2}}}
\]  \hspace{1cm} (2.3)

Where \(U_0\) is the outlet velocity of the jet, \(D_n\) is the nozzle diameter, \(r\) is the location measured from the center of the jet, \(k_1\) and \(k_2\) are constants and \(s\) is the distance from the nozzle outlet. The centerline velocity of the circular turbulent jet is described by the following function.

\[
U(s) = \sqrt{\frac{k_1}{2} \frac{U_0 D_n}{s}}
\]  \hspace{1cm} (2.4)

For a 2D (also called plane) turbulent jet the regions show similarity but the length of the potential core differs and the entrainment differs as well. This leads to a different velocity profile in comparison to the turbulent jet. This is of importance because the in view of computational time most of the calculations will be done in 2D. For the plane turbulent jet the next velocity profile occurs in the fully developed region.

\[
\frac{U}{U_m} = e^{-0.693r^2}
\]  \hspace{1cm} (2.5)
Where \( U_m \) is the velocity in the center of the jet. The profile of the velocity tangent to the direction of flow of the turbulent jet is described with Goertler’s solution (Abramovich et al., 1984):

\[
\frac{V}{U_m} = \frac{1}{\eta} \left( \frac{\eta y}{x} - \frac{\eta y}{x} \tanh^2 \left( \frac{\eta y}{x} \right) - 0.5 \tanh \left( \frac{\eta y}{x} \right) \right)
\]  \hspace{1cm} (2.6)

Where \( \eta \) has a value of 7.67 and \( V \) is the vertical velocity. The centerline velocity of the plane turbulent jet is described by the following equation.

\[
\frac{U_m}{U_0} = 2.58 \sqrt{\frac{x}{d_0}}
\]  \hspace{1cm} (2.7)

### 2.2.1 Entrainment: fully developed region

In the fundamentals the turbulent jet is momentum driven. The momentum flux of any cross section of the jet flow must be conserved and is described as:

\[
I(s) = \rho u 2\pi \int_0^\infty u_j(s,r)^2 r \cdot dr = I_0
\]  \hspace{1cm} (2.8)

Where \( I_0 \) is the momentum flux and \( Q_0 \) is the flow rate at the nozzle exit and are described as follows.

\[
I_0 = \rho u Q_0 u_0
\]  \hspace{1cm} (2.9)

\[
Q_0 = \mu_n \frac{1}{4} \pi D_n^2 u_0
\]  \hspace{1cm} (2.10)

Here \( \mu_n \) is the discharge coefficient which is dependent on the geometry of the nozzle. Discharge coefficients can be found in White (2009) and are 0.6 for bad geometries and 1 for nice and smooth geometries.

It is assumed that \( \mu_n \) is equal to one, inserting this into eq 2.10 and then substituting into 2.9 and integrating gives that \( k_1 \) must be equal to \( k_2 \). Those are chosen to be 77 the same as is done in Albertson et al.(1950).

Integrating the velocity profile in the region of the fully developed flow gives the flow rate with respect to the distance from the jet exit and is expressed in formula form as follows.

\[
Q(s) = Q_0 \sqrt{\frac{8}{k_n}} \frac{s}{D_n} \approx 0.32 Q_0 \frac{s}{D_n}, \text{ for } s \geq s_{dr}
\]  \hspace{1cm} (2.11)
2.2. SINGLE PHASE TURBULENT JETS

It is seen that the flow rate is increasing linearly. This increase in flow is due to the entrainment of ambient water into the jet. Differentiating Equation 2.11 gives the steepness which is

\[ \frac{dQ}{ds} = 0.32Q_0 \frac{1}{D_n} \]  

(2.12)

Substituting eq 2.10 for \( Q_0 \) with \( \mu_n \) equal to 1 gives

\[ \frac{dQ}{ds} = \alpha_{ent}\pi D_n u_o \]  

(2.13)

Where \( \alpha_{ent} \) represents the entrainment coefficient which is equal to 0.081. If this same approach is applied to the empirical relations for the jet velocity of a plane jet it is found that the entrainment coefficient is equal to 0.085. Given the variation of empirical constants this difference can be neglected.

2.2.2 Entrainment: development region

At the beginning of the jet just after the nozzle exit the mixing layer is almost zero so the entrainment is also zero. The entrainment coefficient will increase in this region just as the mixing layer is increasing.

Albertson et al. (1950) derived the following equation for the flow development region of a free circular jet.

\[ Q_{dr}(s) = Q_0 \left[ 1 + f_1 \frac{s}{D_n} + f_2 \left( \frac{s}{D_n} \right)^2 \right] \]

\[ f_1 = 2\sqrt{\frac{\pi}{k}} - \sqrt{\frac{8}{k}} \approx 0.081 \]  

(2.14)

\[ f_2 = \frac{6^{-\nu \pi}}{k} \approx 0.013 \]

Differentiation of this equation leads to the entrainment equation in the flow development region and from this equation the entrainment coefficient for the flow development can be derived as a function of distance from the jet.

\[ \frac{dQ_{dr}}{ds} = \alpha_{ent,dr}\pi D_n u_0 \]

\[ \alpha_{ent,dr} = \left( \frac{1}{4} f_1 + \frac{1}{2} f_2 \frac{s}{D_n} \right) \]  

(2.15)
2.3 The cavitating jet

Figure 2.3: A cavitating free jet ($Dn = 7\text{mm}, p_j = 2.5\text{MN/m}^2, p_a = 0.4\text{MN/m}^2$) (Nobel, 2013)

In Figure 2.3 an image of a cavitating jet is shown. The cavitating jet is an example of cavitation produced in a shear flow. Shear flow cavitation and inception of the cavitation are mainly controlled by their non-cavitating flow structure. The first hypothesis is that due to the entrainment ambient water is accelerated into the jet. This acceleration of the mean flow causes a drop in pressure that will lead to cavitation. The second hypothesis is that the vortices present in a turbulent flow are the cause of the cavitation. Within these vortices the pressure is low giving cavitation a chance to develop (Xing et al., 2005). As a result the minimum pressure in the vortices can be very different than the mean pressures.

In jets cavitation inception depends primarily on the structure of the non-cavitating flow, axisymmetric jets are rather complicated because several kinds of vortices are generated. Such as toroidal, streamwise and helix vortices.

Furthermore experimental results on the cavitation inception of a cavitating jet are extremely scattered (Franc and Michel, 2006). There are jets that start to cavitate at a cavitation number of 0.1 but there are some jets
that are already cavitating at a cavitation number of approximately 2. This spread is due to the nozzle geometries, cavitation criteria and by the water quality which strongly affects the results of the experiments. Ooi (1985) showed the influence of the dissolved air content on the cavitation number at which the first cavitation took place. Some results from the experiments of Ooi are listed in Table 2.1. It is seen that with an higher dissolved air content the critical cavitation number is higher, which means that cavitation takes place at lower velocities.

Table 2.1: Measurements on cavitating jets by Ooi (1985) showing the influence of nozzle diameter and dissolved air content on the cavitation number at which cavitation is observed, at an ambient pressure of 0.13 MN/m²

<table>
<thead>
<tr>
<th>dissolved air content (ppm)</th>
<th>jet diameter (mm)</th>
<th>Nuclei concentration (per cm³)</th>
<th>Nuclei size</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>3.17</td>
<td></td>
<td>5µm &lt; R &lt; 10µm</td>
</tr>
<tr>
<td>14.1</td>
<td>0.16</td>
<td></td>
<td>150</td>
</tr>
<tr>
<td>10.9</td>
<td>0.08</td>
<td></td>
<td>74</td>
</tr>
<tr>
<td>7.7</td>
<td>0.06</td>
<td></td>
<td>24</td>
</tr>
<tr>
<td>4.2</td>
<td>0.03</td>
<td></td>
<td>5.3</td>
</tr>
</tbody>
</table>

According to the experiments carried out by Ooi (1985) the cavitation inception number is almost the same for a given nozzle geometry independent of the Reynolds number based on the jet exit velocity and the nozzle diameter.

According to the studies of Ooi the cavitation inception number in case of the biggest jet diameter he used doesn’t vary much with the air content. It is also seen that the bigger the nozzle diameter the higher the cavitation inception number. Ooi also observed that for his smaller nozzles the cavitation started at approximately the end of the potential core where in his bigger nozzles cavitation started more close to the exit of the nozzle.

This can be declared by influences such as the turbulent intensity at the nozzle inlet and the length of the nozzle. The turbulent intensity showed an influence of about 3 to 15 percent. The length of the nozzle also showed an influence because it influences the thickness of the boundary layer at the outlet.

In Figure 2.4 the influence of the cavitation number on the cavitation around a jet is depicted. It is seen that if the cavitation number becomes lower the amount of bubbles becomes larger.
2.3. THE CAVITATING JET

Figure 2.4: Influence of the cavitation number on the cavitation cone around the jet (Hutli et al., 2007). The jet flow is from the left to the right. From top to bottom: Jets at different cavitation numbers. From left to right: Shows the jet progressive in time. Top to Bottom: $\sigma = 0.0621$, $\sigma = 0.0303$, $\sigma = 0.0202$, $\sigma = 0.0256$, $\sigma = 0.0127$

All in all there are a lot of factors influencing the cavitation around jets and therefore it is complicated to obtain a comprehensive view and separate the various variables, such as the water air content, that governs the cavitation process of a cavitating jet.

2.3.1 Cavitation models

Cavitation is a problem that occurs in multiple applications such as: Nozzles, along hydrofoils in machinery and along the propulsion screw of a ship. The occurrence of cavitation is mostly undesirable, due to the loss of efficiency and damage to parts of the machinery, therefore a lot of research is done to fully understand and predict cavitation. Numerical analysis is also part of the scope of research for cavitation. Because numerical models have the ability to test certain geometries without making real life prototypes. This research resulted in some models for the numerical modeling of cavitation. The next models are available in OpenFOAM.

- Schnerr and Sauer (2001)
2.3. THE CAVITATING JET

- Kunz et al. (2000)
- Homogeneous mixture model

Selection of cavitation model

In the previous section a few pictures of a cavitating jet were shown. In these pictures it is visible that the cavitation layer doesn’t seem to be a nice and continuous layer of gas but more of a regime full of gas bubbles with pure liquid in between them. Since it is a bubbly flow around the jet the models which use the Rayleigh-plesset equations for bubble growth seem to be the most suitable to simulate the cavitating jet. From these models the model of Schnerr and Sauer (2001) is the only one that uses the amount and radius of nuclei already present in the fluid. Since in Ooi (1985) it is demonstrated that the amount and radius of the already present nuclei are important for the inception of cavitation it is preferable to have control of these variables. The Kunz et al. (2000) model uses $u_\infty$ and $t_\infty$ as an input. These constants vary in space and are hard to determine for the cavitating jet. The homogeneous mixture model is a full thermodynamic model and is takes more time to compute than the other two models. This is why the model of Schnerr and Sauer will be used for the modeling of the cavitation. This model is discussed in further detail in the following section.

2.3.2 Schnerr and Sauer (2001)

In the work of Schnerr and Sauer it is assumed that the vapor is filled with tiny spherical bubbles, so called nuclei, which can be described with the simplified Rayleigh-Plesset equation.

In general a bubble flow is treated as a homogeneous mixture. So only one set of equations is used for description. If the fluid enters or leaves the cavitation region the mixture changes rapidly from the pure liquid value to a value that is much lower or vice versa. To avoid problems due to a discontinuous density distribution the volume of fluid method is used. In addition to the continuity and the momentum equation the solution of a transport equation for the cell vapor fraction needs to be added to the Volume of fluid method. The cell volume vapor fraction is defined as vapor volume divided by the cell volume.

The change of the vapor volume fraction per cell volume is now dependent on the number of bubbles in the cell volume and the expansion rate of these bubbles of vapor. The expansion rate of a bubble can be described with the
Rayleigh-Plesset Equation (equation 2.16) for the dynamics of a spherical body of gas in an infinite amount of liquid.

\[
\frac{P_b(t) - P_\infty}{\rho_l} = \frac{R}{t^2} + \frac{2}{3}(\frac{dR}{dt})^2 + \frac{4\nu_l}{R} \frac{dR}{dt} + \frac{2S}{\rho_l R} \tag{2.16}
\]

Where \(P_b\) is the pressure inside the bubble, \(P_\infty\) is the pressure infinitely far from the bubble, \(R_b\) is the bubble radius, \(\nu_l\) is the kinematic viscosity of the surrounding liquid and \(S\) is the surface tension of the bubble.

The fraction of gas in a unit volume cell can be calculated with the radius of the bubbles that are present in the cell as follows

\[
\alpha_v = \frac{n_b 4\pi R^3_b}{1 + n_b 4\pi R^3_b} \tag{2.17}
\]

Where \(R_b\) denotes the radius of the bubbles and \(n_b\) the number of bubbles per volume of fluid.

With the formula of the mixture density \(\rho_m = \alpha_v \rho_v + (1 - \alpha) \rho_l\), the variation of the mixture density can be expressed as a function of the change in the vapor volume fraction.

\[
\frac{D\rho_m}{Dt} = (\rho_v - \rho_l) \frac{D\alpha_v}{Dt} \tag{2.18}
\]

with the substantial derivative the continuity equation becomes

\[
\frac{\partial \rho_m}{\partial t} + \frac{\partial (\rho_m \bar{u}_i)}{\partial x_i} = \frac{D\rho_m}{Dt} + \rho_m \frac{\partial \bar{u}_i}{\partial x_i} = 0 \tag{2.19}
\]

By substituting the above two equations into each-other the mass transfer source terms can be written as

\[
\frac{\partial}{\partial t} (\rho_v \alpha_v) + \frac{\partial}{\partial x_j} (\rho_v \alpha_v u_{v,j}) = \rho_v \frac{D\alpha_v}{Dt} + \alpha_v \rho_v \frac{\partial \bar{u}_j}{\partial x_j} = \frac{\rho_l \rho_v}{\rho_m} \frac{D\alpha_v}{Dt} = S \tag{2.20}
\]

The derivative of the vapor volume fraction can be derived from Equation 2.17 as follows

\[
\frac{D\alpha_v}{Dt} = \frac{4\pi n_b R^2_B}{(1 + \frac{4}{3} \pi n_b R^3_B)^2} \frac{DR_B}{Dt} = \frac{3}{R_B} \alpha_v (1 - \alpha_v) \frac{DR_B}{Dt} \tag{2.21}
\]

Where \(DR_B/Dt\) denotes the change in the bubble radius which is given by the simplified Rayleigh-Plesset equation

\[
\frac{DR_B}{Dt} = \sqrt{\frac{2}{3} \frac{p_b - p}{\rho_l}} \tag{2.22}
\]
Where the far field pressure $p$ is taken to be the ambient pressure of the computational cell. And the bubble surface pressure is the pressure at the surface of the bubble which is assumed to be equal to the saturation vapor pressure. Now the mass transfer rate $S$ can be expressed as a function of the bubble radius as follows.

$$S = \frac{\rho_v \rho_l}{\rho_m} \frac{3 \alpha_v (1 - \alpha_v)}{R_B} \sqrt{\frac{2 p_v - p}{\frac{3}{\rho_l}}}$$ \hspace{1cm} (2.23)

When $p_v$ is bigger than $p$ the above formula is in the correct form for evaporation. When $p$ is bigger than $p_v$ the square-root term of the above formula changes to

$$\sqrt{\frac{2 p - p_v}{\frac{3}{\rho_l}}}$$ \hspace{1cm} (2.24)

and represents the condensation terms.

### 2.4 Overview CFD

The objective of this thesis is to investigate the possibility of modeling a cavitating jet in available Computational fluid dynamics (CFD) software the focus of this section is on the work done on numerical simulations of cavitating jets. Most of the works done on cavitation are dedicated to cavitation along hydrofoils Li (2012), Shang et al. (2012) and inside nozzles of machinery Salvador et al. (2010) and Zhang and Zhang (2012). But there have been several attempts to model cavitating jets and their mechanisms (Xing et al., 2005), Alehossein and Qin (2007) and Peng and Shimizu (2013) to solve problems that occur during the numerical modeling of a cavitating jet. The most notable work is done by Peng et al. and is discussed in the further part of this section.

#### 2.4.1 Peng et al.(2013)

The dynamics of the submerged bubbles depends on the surrounding temperature, pressure and liquid surface tension. For the simulation of the dynamics of a bubble traveling along the flow of a water jet, Alehossein and Qin (2007), developed a density-based method in combination with a finite difference computation of the Rayleigh-Plesset equation. To solve this equation that governs the growth and collapse of the bubble a variable time-step technique was applied to solve for the highly nonlinear bubble oscillation equation. The pressure and velocity data obtained from a CFD analysis
for a jet flow where used as inputs for this model but the computed bubble dynamics where not taken back to the flow, neglecting bubble effects on the flow. Peng and Shimizu (2013) presented a hybrid method by combining the Rayleigh-Plesset equation with a compressible Reynolds-Average Navier-Stokes (RANS) computation. The effect of the bubble dynamics on the surrounding flow where accounted for by adding a source term to the RANS equations. This method showed the possibility to predict the intensity of cavitation, but is limited to the case of weak cavitation.

For the simulation of intensively cavitating jets a pressure-based compressible mixture flow method was developed by Peng et al. (2011). A simplified estimation of the compressibility of the homogeneous bubble-liquid mixture was proposed and the mean flow was calculated with the RANS equations for compressible flow. The set of the RANS equations was solved with a self developed algorithm Peng et al. (2001).

The liquid vapor mixture is considered as a single continuum with one momentum equation and transport equation for the phase fraction. The density of the fluid is taken as the volume average and is defined as follows

$$\rho_m = \rho_l \alpha_l + \rho_v \alpha_v$$  \hspace{1cm} (2.25)

Where $\rho_m$ is the mixture density, $\rho_l$ is the liquid density, $\alpha_l$ is the volume fraction of liquid, $\rho_v$ is the density of vapor and $\alpha_v$ is the volume fraction of vapor. Taking the derivative of this equation the variation of the mixture density is obtained as

$$\frac{d\rho_m}{dt} = \alpha_l \frac{d\rho_l}{dt} + \alpha_v \frac{d\rho_v}{dt} + (\rho_v - \rho_l) \frac{d\alpha_v}{dt}$$ \hspace{1cm} (2.26)

considering the continuity of the mixture flow Equation 2.26 is rearranged in the following form

$$\frac{1}{\rho_m} \frac{d\rho_m}{dt} = \frac{\alpha_l}{\rho_l} \frac{d\rho_l}{dt} + \frac{\alpha_v}{\rho_v} \frac{d\rho_v}{dt}$$ \hspace{1cm} (2.27)

Then with the definition of the compressibility

$$c^2 = \left( \frac{\rho}{p} \right)$$ \hspace{1cm} (2.28)

Equation 2.27 is rewritten as

$$\frac{1}{\rho_m} \frac{d\rho_m}{dt} = \frac{\alpha_l}{\rho_l c_l^2} \frac{dp_l}{dt} + \frac{\alpha_v}{\rho_v c_v^2} \frac{dp_v}{dt}$$ \hspace{1cm} (2.29)

Where $c$ denotes the sonic speed and $p$ the pressure. For the application in engineering it is assumed that the change in pressure for the vapor phase
is approximately the same as the change in pressure over time in the liquid phase, hence \( \frac{dp_l}{dt} \approx \frac{dp_g}{dt} \). Then the variation of the mixture density is directly linked to the liquid pressure.

\[
\frac{1}{\rho_m} \frac{d\rho_m}{dt} = \frac{1}{\rho_m c_m^2} \frac{dp_l}{dt},
\]

(2.30)

Giving the average sonic speed as follows

\[
\frac{1}{\rho_m c_m^2} \approx \frac{\alpha_v}{\rho_l c_v^2} + \frac{\alpha_l}{\rho_i c_i^2},
\]

(2.31)

As governing equations for the two-phase fluid mixture RANS equations for compressible flow are used. The temperature variation caused by cavitation is supposed to be very small. Conservation of mass and momentum are employed together with the density pressure relation of the mixture. The transport equation of gas mass fraction is used to estimate the volume fraction of gas. The turbulence is evaluated with the RNG k-\( \epsilon \) model for high Reynolds number flows.

Due to the expanding and contraction of the cavitation bubbles some turbulence energy may be absorbed and the eddy viscosity of the bubbly flow region is much lower than the viscosity of the fluid. Therefore a modification of the eddy viscosity is needed to account for the effect that the cavitation exerts on the eddy viscosity.

\[
\mu_t = \frac{\chi \rho_m C_\mu k^2}{\epsilon},
\]

(2.32)

Where \( \mu_t \) is the eddy viscosity \( C_\mu \) is a constant and \( \chi \) is the modification factor for the eddy viscosity dependent on the fraction of vapor given by 2.33.

\[
\chi(\alpha_v) = \frac{1 + (c_p - 1)(1 - \alpha_v)^4}{\alpha_v + c_p(1 - \alpha_v)}
\]

(2.33)

Where \( c_p = \rho_l / \rho_v \) which denotes the ratio of liquid and gas density.

Results of this method is shown in Figure 2.5. It is seen that the cavitation starts to develop inside the nozzle. The cavitation layer then continues into the ambient water. The above presented method is useful to estimate the global properties of the flow field of a cavitating jet. The difference of this calculation in comparison to the calculations performed later on with OpenFOAM is that in this calculation the fluid is taken to be compressible whereas the OpenFOAM solver is based on an incompressible fluid.
2.4. OVERVIEW CFD

Figure 2.5: Steady state void fraction of a 2D simulated cavitating jet as obtained by Peng and Shimizu (2013) at different cavitation numbers. The flow is from the left to the right.
Chapter 3

Models and discretization

The calculations of the turbulent jet and the cavitating jet will be performed in the CFD package OpenFOAM. In order to get an idea of how OpenFOAM works and discretizes the equations the first part of this chapter will discuss the Finite Volume Method (FVM). The second part of this chapter will discuss the domains used for the calculation of the single phase turbulent jet and the cavitating jet. The domains geometry, general boundary conditions and mesh will also be discussed.

3.1 discretization

In OpenFoam the (FVM) is used to discretize the Navier-Stokes equations. In this section the finite volume method is explained.

The Finite Volume Method is a discretisation method that is well suited for fluid mechanics. The FVM is based on cell-average values. These average values are assigned to a grid cell. This cell can have every number of edges and can be of every shape. Once a grid is constructed the FVM consist in associating a finite volume, also called a control volume, to a grid point. Then the integral conservation law is applied to the values at this mesh point.

It is very important to maintain the global conservation of flow quantities such as mass, momentum and energy. In Hirsch (2007) it is pointed out that the integral form of the conservation law (Equation 3.1) is valid for every volume. This is a very important property of the integral form. Since the FVM relies on the direct discretization of the integral form the FVM is also conservative. This is an advantage of the FVM.
3.1. DISCRETIZATION

3.1.1 The finite volume discretization

Consider the integral form of the conservation law and apply it to the control volume $\Omega_J$ associated to mesh point $J$.

$$\frac{\partial}{\partial t} \int_{\Omega_J} U \, d\Omega + \oint_{S_J} \vec{F} \cdot d\vec{S} = \int_{\Omega_J} Q \, d\Omega \quad (3.1)$$

Where $U$ represents any given quantity, $\vec{F}$ the flux of quantity $U$, $S_J$ the surface enclosing the volume and $Q$ represents the source terms within the volume. This equation is then replaced by its discrete form, by replacing the volume integrals with the volume average times the volume and the surface integral is replaced by a sum of the fluxes over the faces of the volume.

$$\frac{\partial}{\partial t} (U_J \Omega_J) + \sum_{faces} \vec{F} \cdot \Delta \vec{S} = Q_J \Omega_J \quad (3.2)$$

In this discretized form a few interesting features are distinguished. $U_J$ is not attached to a fixed place within the volume $\Omega_J$ and can be considered as an average value of $U$ within the control volume. The FVM also allows a natural way of implementing boundary conditions. For example, at a solid wall certain normal components such as the mass flux $eq$ are zero. This can be simply implemented by setting the flux at the corresponding boundary to zero.

For the flux part of Equation 3.2 several different numerical schemes can be used. The most common are the central and the upwind scheme. Central schemes are based on local flux estimations and upwind schemes determine the cell face fluxes according to the direction of the convection velocity. An example of a central and an upwind scheme are given in Equation 3.3 and Equation 3.4

$$\frac{U_{ij}}{\Delta t} + \frac{(f_{i+1,j} - f_{i-1,j})}{2\Delta x} + \frac{(g_{i,j+1} - g_{i,j-1})}{2\Delta y} = Q_{ij} \quad (3.3)$$

$$\frac{U_{ij}}{\Delta t} + \frac{(f_{ij} - f_{i-1,j})}{\Delta x} + \frac{(g_{ij} - g_{i,j-1})}{\Delta y} = Q_{ij} \quad (3.4)$$

These numerical schemes both have their advantages and their drawbacks. The central scheme has the advantage that it is more accurate than the upwind scheme. The upwind scheme is more diffusive than the central scheme. The central scheme is less stable than the upwind scheme. To combine both of the qualities from these schemes a blended scheme could be used. With a blend factor it is determined how much the upwind part and how much the
central part contributes to the solution. These three schemes are used in the calculations in Chapter 4 and 5.

A special discretisation method is used in Chapter 5 for the fraction of water. The fraction of water uses the van leer (van Leer, 1997) discretisation. For the time discretization a first order Euler discretisation is used (Greenshields (2015)).

### 3.1.2 The Pressure-Correction Method

To solve the full Navier-Stokes equations for incompressible flows a method known as the Pressure-Correction Method was developed. The method was first applied by Harlow et al. (1965). The methods in this class can be used for stationary flows as well as to the time-dependent incompressible flows. They consist of a basic iterative procedure of the pressure and velocity field. For an initial approximation of the pressure field the momentum equation can be solved to determine the velocity field. The obtained velocity field is does not satisfy the divergence free, continuity equation and has to be corrected. To satisfy this condition the pressure is corrected.

The fundamental approach of pressure correction methods is the decoupling of the pressure field from the velocity field. The pressure field of the previous time step is used to solve for an intermediate velocity field $\vec{v}^\ast$.

\[
\frac{\vec{v}^\ast - \vec{v}^n}{\Delta t} = -\vec{\nabla} \cdot (\vec{v} \otimes \vec{v})^n - \frac{1}{\rho} \vec{\nabla} p^n + v \Delta \vec{v}^n
\]  

(3.5)

The solution for $\vec{v}^\ast$ of Equation 3.5 does not satisfy the continuity equation. This means that an correction needs to be applied to the final values.

\[
\vec{v}^{n+1} = \vec{v}^\ast + \vec{v}'
\]

\[
p^{n+1} = p^\ast + p'
\]  

(3.6)

Where the final values with superscript $n + 1$ have to be a solution of

\[
\frac{\vec{v}^{n+1} - \vec{v}^n}{\Delta t} = -\vec{\nabla} \cdot (\vec{v} \otimes \vec{v})^n - \frac{1}{\rho} \vec{\nabla} p^{n+1} + v \Delta \vec{v}^n
\]  

(3.7)

Substitution of Equation 3.6 into Equation 3.7 and substracting Equation 3.5 gives the relation between the pressure and the velocity corrections:

\[
\vec{v}' = -\frac{\Delta t}{\rho} \vec{\nabla} p'
\]  

(3.8)

Taking the divergence of Equation 3.7 gives the Poisson pressure equation for the pressure correction.
\[ \Delta p' = \frac{\rho}{\Delta t} \vec{\nabla} \vec{v}^* \] (3.9)

The equations described above can be solved with different procedures. One of these procedures is the PISO (pressure implicit with splitting of operator) algorithm. The PISO algorithm can be summed up as follows:

1. Set boundary conditions
2. Solve the discretized momentum equation to compute an intermediate velocity field.
3. Compute the mass fluxes at cell faces.
4. Solve the pressure equation.
5. Correct the mass fluxes at the cell faces.
6. Correct the velocities on the basis of the new pressure field.
7. Update the boundary conditions.
8. Repeat from 3 for the prescribed number of times.
9. Increase the time step and repeat from 1.

### 3.1.3 OpenFOAM solvers

The solvers used in this report are the pisoFoam solver and the interPhaseChangeFoam solver. The pisoFoam solver is used for all the single phase calculations performed. This is a transient solver that assumes an incompressible fluid. The procedure for solving the Navier-Stokes equations is the above mentioned PISO algorithm. The interPhaseChangeFoam solver is used for all the multi-phase calculations performed. The interPhaseChangeFoam solver for two incompressible, isothermal immiscible fluids with phase-change(cavitation). The interPhaseChangeFoam solver also makes use of the PISO algorithm to solve for the Navier-Stokes equations. The procedure for the interPhaseChangeFoam solver is as follows:

1. The vapor volume fraction is calculated with the Schnerr-Sauer model (Section 2.3.2).
2. Then the mixture density is calculated.
3. The velocity and pressure field for the next step is calculated using the mixture density.

4. Increase the time step and restart procedure at 1.

3.2 Geometries

The calculations of the single phase turbulent jet and of the cavitating jet will be performed on different geometries. These geometries, their boundary conditions and their meshes will be discussed in this section.

3.2.1 Single phase turbulent jet model

The geometry of the 2D domain wherein the calculations for the single phase turbulent jet are performed is shown in Figure 3.1.

![Figure 3.1: Geometry of the 2D domain used for the simulations of the turbulent jet](image)

In the above section the geometry is described with its boundaries. These boundaries define the end of the domain and special conditions must be met at this boundaries in order to get good results from the simulations. For the single phase turbulent jet the following boundary conditions must be specified: velocity, pressure, k, omega and epsilon.

For the velocity the following conditions apply. At the inlet an inlet velocity is specified. This inlet velocity is 15 m/s. For the walls in the domain a no-slip condition is implemented. This means that it is assumed
that the fluid is attached to the wall. Since the walls are not moving a
velocity of zero in all directions is imposed. For the ambient boundaries a
zero gradient boundary is implemented.

The pressure is a zero gradient boundary condition everywhere except for
the boundaries that cut through the ambient fluid. Since the ambient fluid
is in a container and has a pressure the same as the ambient fluid which is
0.13 Mpa.

For \( k, \omega \) and \( \epsilon \) the boundary conditions are zero-gradient except at the
walls. At the walls these functions have a boundary condition that is consis-
tent with the law of the wall (White et al., 2003)

Different meshes were generated for this geometry. Three detail images
of these meshes are shown in Figure 3.2. And mesh properties such as number
of cells and minimum cell size are tabulated in Table 3.1.

Table 3.1: Number of cells and minimum cell size for the turbulent jet ge-
ometry

<table>
<thead>
<tr>
<th></th>
<th>coarse</th>
<th>medium</th>
<th>fine</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of cells</td>
<td>109700</td>
<td>185300</td>
<td>340100</td>
</tr>
<tr>
<td>( \text{min } \Delta x ) [m]</td>
<td>0.0005</td>
<td>0.00025</td>
<td>0.000125</td>
</tr>
</tbody>
</table>

Figure 3.2: Details of the mesh refinement levels for the turbulent jet geom-
etry
3.2. GEOMETRIES

3.2.2 Cavitating jet geometry 1

The first geometry for the cavitating jet is a 2D domain with a simple inlet tube of 2.5mm height. The length of the inlet tube is chosen such that the flow inside is a fully developed turbulent flow.

The geometry mesh for this case is shown in Figure 3.3

![Figure 3.3: Geometry for the case with an inlet tube](image)

In case of a multiphase flow one additional boundary condition for the fraction of water needs to be implemented. This condition is taken to be a zero gradient boundary condition at every boundary except for the inlet where pure water enters the domain. The value of $\alpha_l$ is set to one.

Different meshes were generated for this geometry. Three detail images of these meshes are shown in Figure 3.4. And mesh properties such as number of cells and minimum cell size are tabulated in Table 3.2.

Table 3.2: Number of cells and minimum cell width for the different meshes for geometry 1

<table>
<thead>
<tr>
<th></th>
<th>coarse</th>
<th>medium</th>
<th>fine</th>
</tr>
</thead>
<tbody>
<tr>
<td>number of cells</td>
<td>245700</td>
<td>277200</td>
<td>565800</td>
</tr>
<tr>
<td>$\text{min } \Delta x ,[\text{m}]$</td>
<td>0.0005</td>
<td>0.00025</td>
<td>0.000125</td>
</tr>
</tbody>
</table>
3.2. GEOMETRIES

(a) coarse  (b) medium  (c) fine

Figure 3.4: Mesh refinement levels for the mesh of geometry 1

3.2.3 Cavitating jet geometry 2

The second domain has a 3D geometry. This geometry is used to check whether 3D effects of the turbulent jet are of influence on the inception of cavitation. This geometry is also used later on for the calculations performed with LES. In Figure 3.5 a cross section of the geometry is shown.

Figure 3.5: Cross section of the 3D mesh

The boundaries for this case remain the same as in the previous 2D case. One boundary condition is adjusted to meet the fully developed turbulent flow in the inlet tube. This is the velocity inlet boundary condition. At the inlet a velocity profile is applied that corresponds to the turbulent velocity profile.
3.2. GEOMETRIES

The average inlet speed of the flow remains 71 m/s rendering the same cavitation number as in the previous case. All other settings such as discretisation settings and fluid property settings remain the same as in the previous case.

Table 3.3: Number of cells and minimum cell width for the different meshes for geometry 2

<table>
<thead>
<tr>
<th></th>
<th>coarse</th>
<th>fine</th>
</tr>
</thead>
<tbody>
<tr>
<td>number of cells</td>
<td>608935</td>
<td>872363</td>
</tr>
<tr>
<td>$\text{min } \Delta x \text{ [m]}$</td>
<td>0.0005</td>
<td>0.00025</td>
</tr>
</tbody>
</table>

Figure 3.6: Mesh refinement levels for the mesh of geometry 2
3.2.4 Cavitating jet geometry 3

The third geometry of the mesh resembles the mesh which is used in (Peng and Shimizu, 2013) previously mentioned in chapter 2. The turbulence model is also changed to the RNG-k-\(\epsilon\) to resemble the simulations in this paper. This is done because in this paper it is shown that with this geometry it is possible to simulate a cavitating jet with a RANS method. Also the influence of the nozzle geometry can be seen.

The geometry for this case is shown in Figure 3.7.

![Figure 3.7: Geometry for the Peng case](image)

The inlet is 2.88 times the height of the nozzle diameter and the length of the nozzle is 5 times the diameter of the nozzle.

Different meshes where generated for this geometry. Three detail images of these meshes are shown in Figure 3.8. And mesh properties such as number of cells and minimum cell size are tabulated in Table 3.4

Table 3.4: Number of cells and minimum cell width for the different meshes for geometry 3

<table>
<thead>
<tr>
<th></th>
<th>coarse</th>
<th>medium</th>
<th>fine</th>
</tr>
</thead>
<tbody>
<tr>
<td>number of cells</td>
<td>25100</td>
<td>100400</td>
<td>401600</td>
</tr>
<tr>
<td>min (\Delta x) [m]</td>
<td>0.0005</td>
<td>0.00025</td>
<td>0.000125</td>
</tr>
</tbody>
</table>
3.2. GEOMETRIES

Figure 3.8: Mesh refinement levels for the mesh of geometry 3
3.2. GEOMETRIES
Chapter 4

Single phase turbulent jet

The cavitating jet is a complicated flow mechanism involving phase change, compressibility, viscosity and turbulent fluctuations. Complex mechanisms that occur in the flow, such as bubble generation and collapse, influence the turbulence on different time and space scales. To solve for these turbulent influences different methodologies are available (Hirsch, 2007): Direct numerical simulation (DNS), in which the turbulence is simulated by solving directly the Navier-Stokes equation on a small mesh size. The second option is Large Eddy Simulation (LES), in which is solved for a filtered Navier-Stokes equation, where turbulent effects due to eddies smaller than order of mesh size are modeled. At last there is Reynolds-averaged Navier-Stokes (RANS), where time-averaged Navier-Stokes equations are solved and all turbulence effects are modeled.

In OpenFOAM DNS is not affordable due to its huge computational needs and therefore very long computation times. Although LES is less costly in computational time than DNS it still needs a fair amount of computational resources. Since Peng and Shimizu (2013) have shown that it is possible to predict a cavitating jet using RANS code. Therefore the less costly RANS method will be used for the model of the cavitating jet.

The Reynolds-Averaged Navier-Stokes equations and the turbulence models used for the RANS equations will be discussed and evaluated in section 4.1 Thereafter in section 4.2 the results of the turbulent jet will be discussed and the turbulence for the RANS simulations is selected.

4.1 RANS

For turbulent flow, each velocity term and pressure term is a rapidly varying function of space and time. At the moment our mathematics cannot han-
dle this instantaneous fluctuations of these variables. To work around this Osbourne Reynolds rewrote the navier-stokes equations in means of time averaged variables. The time mean of a turbulent fluctuation is defined by

\[ \bar{u} = \frac{1}{T} \int_0^T u \, dt \] (4.1)

Reynolds’s idea was to split each property into two components, the mean of the variable and the fluctuation of the variable, giving the following relations:

\[ u = \bar{u} + u' \quad v = \bar{v} + v' \quad w = \bar{w} + w' \quad p = \bar{p} + p' \] (4.2)

Where the bar indicates the mean velocity and the accent the velocity fluctuations. If these are substituted into the continuity equation and the time mean of each equation is taken then the continuity equation is written as

\[ \frac{\partial \bar{u}}{\partial x} + \frac{\partial \bar{v}}{\partial y} + \frac{\partial \bar{w}}{\partial z} = 0 \] (4.3)

which is the same form as the laminar continuity equation.

Using the method of Reynolds for the momentum equation leads to the next form per component and is written out only for the x direction

\[ \rho \frac{d\bar{u}}{dt} = - \frac{\partial \bar{p}}{\partial x} + \rho g_x + \mu \left( \frac{\partial \bar{u}}{\partial x} - \rho u' u' \right) \\
+ \frac{\partial}{\partial y} \left( \mu \frac{\partial \bar{u}}{\partial y} - \rho u' v' \right) + \frac{\partial}{\partial z} \left( \mu \frac{\partial \bar{u}}{\partial z} - \rho u' w' \right) \] (4.4)

Where \(-\rho u'^2\), \(-\rho u'v'\) and \(-\rho u'w'\) are the turbulent stresses.

In duct and boundary layer flow the stresses related to the direction normal to the wall are dominant over the stresses in the other directions and therefore the other stresses can be dropped out of the equation. Leading to a simpler form of the momentum equation

\[ \rho \frac{d\bar{u}}{dt} \approx - \frac{\partial \bar{p}}{\partial x} + \rho g_x + \frac{\partial \tau}{\partial y} \] (4.5)

Where

\[ \tau = \mu \frac{\partial \bar{u}}{\partial y} - \rho u' v' \] (4.6)

Where the first term on the right hand side represents the laminar part of the stresses and the second term the turbulent part of the stresses. Since the
flux fluctuations $-\rho u'v'$ are unknown there is additional modelling required to get a correct representation of the turbulence in the CFD model. This will be discussed in the next section.

4.1.1 Turbulence

In the previous section the concept of Reynolds averaging is applied to the Navier-Stokes equation. This resulted in equation 4.5 on the right hand side there remains a non-linear term $-\rho u'v'$ from the convective acceleration, this term is known as the Reynolds stress. To close the RANS equations a model for this term is needed.

In 1887 Bousinesq proposed to relate the turbulent stress to the mean shear to close the RANS equations. Equation 4.7 shows the boussinesq hypothesis applied to the Reynolds stress term.

$$\mu_t \frac{du}{dy}$$ (4.7)

Here it is seen that a new term $\mu_t$ is introduced named the turbulent viscosity. The turbulent viscosity can be modeled with several existing models, such as the Spalart-Allmaras, Smagorinsky, $k-\epsilon$ and the $k-\omega$ model. These models are commonly applied in modern engineering models and widely validated. The $k$-epsilon and the $k$-omega models are two equation models and can be divided in into the following five models.

Table 4.1: RANS turbulence models

<table>
<thead>
<tr>
<th>k-$\epsilon$ models</th>
<th>standard k-$\epsilon$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>RNG k-$\epsilon$</td>
</tr>
<tr>
<td></td>
<td>realizable k-$\epsilon$</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>k-$\omega$ models</th>
<th>standard k-$\omega$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>k-$\omega$ SST</td>
</tr>
</tbody>
</table>
4.2 Results

In this section the results of the simulation of the turbulent jet are discussed. Multiple simulations have been carried out on the above described mesh geometry on meshes with different cell sizes. Also different turbulence models have been tested to check their performance. First the velocity profiles of the turbulent jet are checked thereafter the entrainment of the jet is checked.

4.2.1 Velocity

To check the performance of the different turbulence models a basic simulation of a turbulent flow is executed on the geometry discussed in section 3.2.1 of this chapter. The Turbulence models that were tested were the $k-\epsilon$ model, the $k-\omega$ model, the $k-\omega$ SST and the RNG $k-\epsilon$ model. For those cases velocity of the centerline along the $x$-axis are plotted in Figure 4.1.

![Figure 4.1: Centerline velocity of the turbulent jet with different turbulence models. For comparison with Equation 2.7 the blue line is plotted.](image)

In this figure the blue continuous line is the velocity at the center of the plane jet obtained by several experimental observations of a turbulent jet ((Van der Hegge Zijnen, 1958), (Albertson et al., 1950)). The shape
and especially the length of the potential core is dependent on the shape of the nozzle Ashforth-Frost and Jambunathan (1996). Therefore the relation obtained by Albertson et al. (1950) is used because this one is closest to the simulation.

It is seen from the graph that the k-\(\epsilon\) models and the k-\(\omega\) SST give an overshoot at the beginning of the region of developed flow. The solution with the k-\(\omega\) model tends to follow the empirical relation more precisely. The k-\(\omega\) model gives the best results for the centerline velocity of the jet. Next is the check of the velocity profile of the turbulent jet. As a comparison the velocity profile obtained by Tollmien (Rajaratnam, 1976) is used. This is shown in Figure 4.2.

![Velocity profile of the turbulent jet with different turbulence models at \(x = 20D_n\). The blue line represents Equation 2.5.](image)

**Figure 4.2:** Velocity profile of the turbulent jet with different turbulence models at \(x = 20D_n\). The blue line represents Equation 2.5.

It is seen that for the profile of the jet the results obtained with the k-\(\epsilon\) models are the most consistent with (Rajaratnam, 1976). The k-\(\omega\) model shows an overestimation of the velocity at the outer boundaries of the jet.

For the velocity in the y-direction the Equation 2.6 is used to check the velocity profile of the flow tangent to the direction of the jet. This is shown in Figure 4.3.
4.2. RESULTS

Figure 4.3: profile of the velocity in the y-direction of the jet at a distance of $20D_n$. The blue line represents Equation 2.6.

It is seen that the $k$-$\epsilon$ and the $k$-$\omega$ SST follow approximately the same line. The $k$-$\omega$ overestimates the flow tangent to the jet and the RNG $k$-$\epsilon$ model gives an underestimation of the flow towards the jet. It is seen that the best results are given by the $k$-$\epsilon$ and the $k$-$\omega$ SST models.

4.2.2 Mesh

The mesh has been discussed in section 3.2.1 and it is clarified that different grid sizes were used. If the mesh size changes and becomes smaller the solution tends to get closer to the real solution. To show that the solution is mesh-independent the solutions of the coarse, medium and fine mesh are shown in Figure 4.4 for all the different turbulence models.

It is seen that the solution for the medium mesh is almost the same as that of the fine mesh. This means that the cell size doesn’t influence the solution of the calculation significantly if they become smaller than the cell size in the fine mesh. It is concluded that the fine mesh solution is converged.
4.3 Conclusion of turbulent jet

In this chapter the turbulent jet and its importance for the cavitating jet were discussed. Thereafter the CFD simulation of a turbulent jet. At last the turbulent jet was simulated with different models for the turbulence. Those simulations where compared to experimental obtained results for the

4.2.3 Entrainment

In the previous section the velocity profiles given by the different turbulence models are shown and compared. It is seen that in different regimes different turbulence models give the best results. In order to make a decision of which turbulence model would be the best to use for the simulation of the cavitating jet the entrainment of the different models is checked. The results for the entrainment coefficient is shown in Table 4.2. The calculation of these coefficients can be found in Appendix A.

The first value is the entrainment coefficient derived from previously named experimental tests. It is seen that the standard k-ε and the standard k-ω model give the best approximations of the entrainment of the Turbulent Jet

4.3 Conclusion of turbulent jet

Figure 4.4: Influence on the grid size on the solution of the simulation.
4.3. CONCLUSION OF TURBULENT JET

Table 4.2: Entrainment coefficients

<table>
<thead>
<tr>
<th>Theory</th>
<th>0.085</th>
</tr>
</thead>
<tbody>
<tr>
<td>standard k-(\epsilon)</td>
<td>0.088</td>
</tr>
<tr>
<td>RNG k-(\epsilon)</td>
<td>0.097</td>
</tr>
<tr>
<td>standard k-(\omega)</td>
<td>0.088</td>
</tr>
<tr>
<td>k-(\omega) SST</td>
<td>0.112</td>
</tr>
</tbody>
</table>

turbulent Jet. In these comparisons different turbulence models gave the best results which makes it hard to decide which model is the best to use with the simulation of the cavitating Jet. The k-\(\omega\) model was seen to be the most consistent with Equation 2.7. The other two velocity profiles (Equation 2.6 and Equation 2.5) the k-\(\epsilon\) models and the k-\(\omega\) SST model were most consistent. But most important the entrainment is best estimated by the standard k-\(\epsilon\) model and the k-\(\omega\) model. Since the standard k-\(\epsilon\) model is most consistent with most of the above mentioned comparisons the standard k-\(\epsilon\) is used to calculate the cavitating flows.
Chapter 5
Cavitating jet

Now that the single phase turbulent jet has been discussed and a selection for the turbulence model is made the step is made to a cavitating jet. The cavitating jet is simulated with the model of (Schnerr and Sauer, 2001). This model can be found within the interPhaseChangeFoam solver in OpenFOAM. In these simulations the following fluid properties have been specified for the water and the vapor phase.

For the fluid properties the properties of water and water vapor at a temperature of 15 degrees Celsius is used. This gives a vapor pressure of 2300 Pa (Mills, 1999). A kinematic viscosity of 1e-6 $m^2/s$ for water and of 747e-6 $m^2/s$ for the water vapor. The number of dissolved nuclei is set at the standard number used in interPhaseChangeFoam and is 1.6e+13 $m^{-3}$. The diameter of the nuclei present in the water is 2e-6 $m$ which is also the standard setting for interPhaseChangeFoam.

In the first section the calculations are performed with a RANS method. In the second section the calculations are performed with the a LES method.

Table 5.1: Different cases for the RANS calculations within interPhaseChangeFoam

<table>
<thead>
<tr>
<th>Case</th>
<th>Ambient pressure (MPa)</th>
<th>Jet pressure (MPa)</th>
<th>Cavitation number [-]</th>
<th>Discretisation</th>
<th>Geometry</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.13</td>
<td>2.5</td>
<td>0.04</td>
<td>Upwind</td>
<td>1</td>
</tr>
<tr>
<td>2</td>
<td>0.13</td>
<td>2.5</td>
<td>0.04</td>
<td>Upwind</td>
<td>2</td>
</tr>
<tr>
<td>3</td>
<td>4e-3</td>
<td>2.5</td>
<td>0.6e-3</td>
<td>Upwind</td>
<td>2</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>1.1</td>
<td>0.1</td>
<td>Upwind</td>
<td>3</td>
</tr>
<tr>
<td>5</td>
<td>1</td>
<td>1.1</td>
<td>0.1</td>
<td>Central</td>
<td>3</td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>1.1</td>
<td>0.1</td>
<td>Central</td>
<td>3</td>
</tr>
</tbody>
</table>
5.1 RANS simulations

The calculations of the cavitating jet were performed on three different domain geometries, mentioned before in section 3.2, and with different settings for discretisation and different boundary conditions. These settings are summarized in Table 5.1. The different calculations will be discussed in the next part of this chapter.

5.1.1 Case 1: 5mm inlet tube

In this specific geometry the cavitation must originate in the mixing layer of the turbulent jet and is caused solely by entrainment or by shear. The above boundary conditions give a cavitation number of about 0.04 at the nozzle exit. In Figure 5.1 the results of the simulation are shown.

![Figure 5.1: Liquid fraction for RANS case 1 at different simulation times. Red is pure water blue is water vapor mixture. (σ = 0.04, P_a = 0.13 MPa)](image)

For this case it is seen that at the beginning a small vapor bubble starts to develop at the edge of the outflow of the tube. In the second time instant the vapor bubble has grown and has separated from the nozzle outlet. After a while the vapor bubble is transported out of the domain by the flow of the jet (not shown). After this cloud of bubbles disappeared no inception of new cavitation bubbles takes place and the flow that remains is a flow of pure water.
5.1. RANS SIMULATIONS

5.1.2 Case 2: 5mm inlet tube 3D

The second geometry for the turbulent jet is a 3D version of the previous tested geometry. This geometry is generated to see if 3D effects such as vortices can make a change in the outcome of the cavitation simulations. In order to save computational time the tube for development of turbulence is shortened. In Figure 5.2 the results of this computation are shown.

![Figure 5.2: liquid fraction of a cross section for RANS case 2 at different simulation times.](image)

(a) $t = 0.001$
(b) $t = 0.005$
(c) $t = 0.01$

In this case the picture is the same as in the 2D case. Again a cavitation bell exists at the first time instant. At the second time instant the bell has grown. At the third time instant it is seen that the bell of water-vapor mixture is transported out of the domain by the advection which leaves a turbulent jet without cavitation.
5.1.3 Case 3: 5mm inlet tube 3D low pressure domain

In the third case the pressure of the ambient water is lowered to a very low pressure. This results in a decrease of the cavitation number. As discussed in chapter 1 if the cavitation number gets lower the cavitation gets more intense for the same geometry.

The pressure at the outlet boundary on which the ambient pressure is defined is set to 4000 Pa which is approximately 2 times the vapor pressure of water at 15 degrees Celsius. This decrease in ambient pressure results in a cavitation number of 0.6e-3. This corresponds to a jet entering the ambient water with a velocity of approximately 652 m/s.

In Figure 5.3 the results for the lowered ambient pressure is shown.

Figure 5.3: Liquid fraction of a cross section of RANS case 3 with a cavitation number lower than the previous case at different times. ($\sigma = 0.6e - 3, P_a = 4e - 3$ MPa)
5.1. RANS SIMULATIONS

In this figure it can be seen that at the first time instant a water-vapor bubble is present just like in the previous cases. At the second time instant a long water-vapor regime is seen which is still attached to the outlet of the nozzle. This is what is expected to be seen for a cavitating jet. At the third time instant the bubble cone has vanished and does not return anymore. The jet that remains is a single phase turbulent jet without cavitation.

5.1.4 Case 4: Peng nozzle standard interphase

The fourth case to be discussed is the case with the geometry used in the paper of (Peng and Shimizu, 2013). In previous cases it was seen that with a geometry where cavitation can only be produced within the shear layer no cavitation occurred. With this geometry a small nozzle is reconstructed. As is seen in Figure 2.5 the cavitation starts to develop within the nozzle and then continues into the free water. To check if this can trigger the cavitation computations are performed on this geometry.

The results for this simulation are shown in Figure 5.4.

![Figure 5.4: Liquid fraction for RANS case 4 with the geometry of the Peng nozzle with the interPhaseChangeFoam solver set with an upwind discretisation.](image)

Figure 5.4: Liquid fraction for RANS case 4 with the geometry of the Peng nozzle with the interPhaseChangeFoam solver set with an upwind discretisation. ($\sigma = 0.1, P_a = 0.1\text{MPa}$)
5.1. RANS SIMULATIONS

As can be seen in the above figures a vapor boundary layer starts to grow inside the nozzle and a vortex bubble at the end of the nozzle. After a certain amount of time the vapor bubble at the end of the nozzle starts to move in the direction of the flow and slowly disappears. The cavity inside the nozzle remains in place but doesn’t stretch or slink. None of the bubbles is advected into the ambient water.

5.1.5 Case 5: Peng nozzle central interphase

The previous simulation is also executed with a central discretisation scheme for U, k, epsilon and the pressure. As mentioned before in chapter 3 a central scheme has a higher accuracy than an upwind scheme and is less diffusive. To see the effect of the discretisation technique all other variables are kept the same.

The results for this simulation are shown in Figure 5.5

![Figure 5.5: Liquid fraction for RANS case 5 with the geometry of the Peng nozzle with the interPhaseChange solver set with a central discretisation.](image)

It can be seen that in the first two images the solution is approximately the same as with the upwind discretisation. A vapor layer starts to develop...
5.1. RANS SIMULATIONS

in the nozzle and a small bubble is seen at the exit of the nozzle. However in the third picture of this case there is still a big bubble of water-vapor mixture present in the domain. In the end the result of this simulation is the same as in the previous case. No vapor layer exists. Only a vapor regime at the wall inside the nozzle which sometimes releases a small vapor bubble.

5.1.6 Case 6: Peng nozzle adjusted RNG $k - \epsilon$ central

In literature about cavitation such as (Peng and Shimizu, 2013), (Li, 2012) there is an adjustment in the turbulence model for the places where a water mixture is present. At a cell with a water mixture the turbulent viscosity has to be adjusted with a factor dependent on the fraction of water in this cell. The correction factor is the same as used in (Peng and Shimizu, 2013) and is computed with Equation 2.33. This was implemented in the RNG $k - \epsilon$ model in OpenFOAM and its implementation can be found in Appendix C. All the settings for discretisation and all other variables are the same as in the Peng central case.

In Figure 5.6. The results for this calculation are shown. It is seen that the first two steps give approximately the same result as in the previous two cases. There is a slight difference inside the tube where the adjustment to the turbulence model is visible in the form of more shedding from the wall. In general the flow pattern is the same as the previous models in the rest of the time. The vapor bubble disappears and sometimes minuscule bubbles are released from the cavity inside the nozzle.
5.1. RANS SIMULATIONS

Figure 5.6: Liquid fraction for RANS case 6 with the geometry of the Peng nozzle with the interPhaseChangeFoam solver set with an central discretisation scheme and an adjusted turbulence model as described in Equation 2.33. ($\sigma = 0.1, P_a = 0.1 \text{MPa}$)

5.1.7 Discussion

Above the results of the RANS calculations where depicted and analyzed. It is seen that in every different simulation cavitation occurred for a small amount of time but vanished afterwards. If there was a nozzle geometry incorporated, as in the cases with the geometry used in (Peng and Shimizu, 2013), a water-vapor layer was present inside the nozzle. Sometimes a bubble is shed from this layer and is advected into the ambient water. But still there is no full cavitation layer around the jet as is seen in experiments. This might be due to the fact that the interPhaseChangeFoam solver handles both vapor and liquid fractions as incompressible fluids. The code of (Peng and Shimizu, 2013) handles the fractions as compressible fluids.

Pressure plots from the above calculations show that at the edge of the jet there is indeed a pressure drop due to the acceleration of the water in the entrainment region as was assumed. This drop is due to the fact that ambient fluid is entrained into the jet. This pressure drop is shown for the
5.1. RANS SIMULATIONS

case of the 2D inlet tube in Figure 5.7 and for the last case with the Peng nozzle in Figure 5.8. It can be seen that the pressure does not drop below the vapor pressure (2300 Pa) and thus no cavitation occurs.

Figure 5.7: Pressure in kPa for RANS case: \((\sigma = 0.04, \quad P_a = 0.13\text{MPa})\)

Figure 5.8: Pressure in kPa for RANS case: \((\sigma = 0.04, \quad P_a = 0.13\text{MPa})\)

Thus it seems that the interPhaseChangeFoam solver within OpenFOAM in combination with a RANS technique is unable to give appropriate results for the cavitating jet.

The most probable reason for not being able to solve the cavitating jet with a RANS technique is the fact that the RANS equations gives a time average at every cell. This gives a representation of the mean flow which is not accelerated enough to give pressures fluctuations that are low enough to induce cavitation.
In the previous section it was tried to simulate the cavitating effect in a cavitating jet with a RANS method. It was seen that OpenFOAM was unable to give correct results with the interPhaseChangeFoam solver. The main issue seems to be the complex flow structures around the jet as described in chapter 2. These structures are not incorporated in the RANS technique. Another method discussed before is the LES technique which stands for Large Eddy Simultaion. This method also simulates the complex eddies created in the shear region of the flow. An example can be seen in Figure 5.9 where on the left a typical RANS outcome is shown and on the left a typical LES solution. Small eddies can be distinguished in the LES case. Within these eddies an under pressure is generated. As can be seen in Figure 5.9 If these pressures drop under the vapor pressure limit there will be cavitation at this point. To investigate the influence of these eddies with respect to cavitation some LES simulations have been performed. An overview of these simulations is given in Table 5.2.

The interPhaseChangeFoam solver is capable to handle both RANS and LES equations. Due to this capability this solver is also used for the LES calculations. The turbulence model that is used for this calculation is the homogeneousDynSmagorinski turbulence model.

Table 5.2: Different cases for the LES calculations within interPhaseChangeFoam. The fractions associated with the blended schemes represent the amount of central discretisation.

<table>
<thead>
<tr>
<th>Case</th>
<th>Ambient pressure (MPa)</th>
<th>Jet pressure (MPa)</th>
<th>Cavitation number [-]</th>
<th>Discretisation</th>
<th>Geometry</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case 1</td>
<td>0.13</td>
<td>2.5</td>
<td>0.04</td>
<td>Upwind</td>
<td>3</td>
</tr>
<tr>
<td>Case 2</td>
<td>0.5</td>
<td>2.5</td>
<td>0.2</td>
<td>Upwind</td>
<td>3</td>
</tr>
<tr>
<td>Case 3</td>
<td>0.13</td>
<td>2.5</td>
<td>0.04</td>
<td>Upwind</td>
<td>3</td>
</tr>
<tr>
<td>Case 4</td>
<td>0.13</td>
<td>2.5</td>
<td>0.04</td>
<td>Blended(0.9)</td>
<td>3</td>
</tr>
<tr>
<td>Case 5</td>
<td>0.13</td>
<td>2.5</td>
<td>0.04</td>
<td>Blended(0.99)</td>
<td>3</td>
</tr>
<tr>
<td>Case 6</td>
<td>0.13</td>
<td>2.5</td>
<td>0.04</td>
<td>Central</td>
<td>3</td>
</tr>
<tr>
<td>Case 7</td>
<td>0.13</td>
<td>0.18</td>
<td>0.04</td>
<td>Central</td>
<td>3</td>
</tr>
</tbody>
</table>

5.2.1 Case 1: standard case, coarse mesh

The results of the standard case are depicted in Figure 5.10. The contours shown are the 3D contours of the vapor bubbles. The blue and red represent
5.2. LES SIMULATIONS

Figure 5.9: Velocity field and pressure of a single phase RANS simulation versus a single phase LES simulation. From left to right: velocity field of a RANS calculation, pressure field of a RANS calculation, velocity field of a LES calculation and the pressure field of a LES calculation.

the vapor fraction. In the standard case it is seen that the water-vapor mixture bubbles start of with two separate toroidal bubble clouds. But in contrast with the RANS studies the bubbles do not stop to exist. Instead new bubbles are formed inside the vortices where the pressure is lower than the vapor pressure of water.

5.2.2 Case 2: high ambient pressure

By giving rise to the ambient pressure the cavitation number is higher than in the standard case. The higher the cavitation number the less cavitation is expected to occur. To investigate if this trend holds the results calculations performed with the higher ambient pressure are shown in Figure 5.11.

It is seen in these images that in the begin one large toroidal bubble cloud exists. This bubble is smaller than the first bubbles in the previous case. At further stages this bubble has disappeared. In the last image a single small bubble is seen. This bubble will eventually disappear as well and a domain without any cavitation bubbles will remain. This shows that an increase of
5.2. LES SIMULATIONS

pressure indeed decreases the inception of cavitation bubbles.

Since every LES simulation is very sensitive to the size of the grid mesh size is a factor with a huge impact on LES simulation. To research the influence of the mesh size on the LES calculations of the turbulent jet the calculations were also performed on a finer mesh.

5.2.3 case 3: Standard case fine mesh

The first results shown in Figure 5.12 are those of the calculation preformed with the standard solver settings as set in the OpenFOAM tutorials. For the discretisation a second order upwind scheme is used.
5.2. LES SIMULATIONS

Figure 5.11: Liquid fraction for LES case 2, upwind scheme, at different time steps. ($\sigma = 0.04, P_a = 0.5\text{MPa}$)

In the images of the results it is seen that after the first time sequence there are more separate bubble clouds present in the domain. In further steps these bubbles keep increasing in numbers and are bigger than was the case with the coarser mesh.

5.2.4 case 4 & 5: standard case blended

To investigate the influence of the discretisation on the solution of the calculation a calculation with central discretisation is performed. In the case of a full central discretised solver the time steps became very small. Therefore it is chosen to calculate with an blended discretisation. The blend factor for this calculation is set to 0.9 which means that 0.9 is central discretized and
5.2. LES SIMULATIONS

Figure 5.12: liquid fraction of LES case 3, upwind scheme, at different time steps. \((\sigma = 0.04, P_a = 0.13\text{MPa})\)

0.1 is an upwind discretisation.

In Figure 5.13 the results for this central case are shown. It is visible that in contrast to the upwind scheme the bubbles of the first time step are more spread and more irregular, and the bottom bubble has been separated. In the second image the bubble regime is way bigger than in the upwind case. In the last step the cone remains as wide as in the second step but is still bigger than the third step in the upwind case. Also the cavitation bubbles start to occur closer to the nozzle exit.
An additional simulation with a blended scheme with 99 percent central and 1 percent upwind was performed to investigate the change in the solution. Results for this additional simulation can be found in Figure 5.14. It can be seen that the bubbles don’t penetrate the core of the jet anymore. This was also observed by Ran and Katz (1994).

### 5.2.5 Case 6: standard case central

At last a short central simulation was performed for the cavitating jet. The results of this simulation is shown in Figure 5.15.

In these figures it is seen that the bubbles don’t penetrate the core of the
jet and that the bubbles are less intense as in simulations performed with an upwind scheme and the blended scheme with 90 percent central and 10 percent upwind. It shows comparison with the blended scheme in which is 99 percent central and 1 percent upwind.

**low jet pressure** To see the effect of interPhaseChangeFoam on low speed jets that do not cavitate a slow jet with an high cavitation number was simulated on the fine mesh. The results of this simulation can be found in Figure 5.18. On the left hand is an image of the volume fraction of vapor. As expected no cavitation bubbles are visible in case of a slow jet with a high cavitation number. The jet velocity field is depicted in the middle of the figure. This field shows comparison with the image of the single phase LES picture in Figure 5.9. On the right hand side the pressure field of the jet is visible. As can be seen the lowest pressure is $1.22 \times 10^5$ Pa which is way higher than the vapor pressure of water.
5.2. LES SIMULATIONS

Figure 5.18: Results for LES case 7: A jet with a low jet pressure ($\sigma = 0.2$, $P_a = 0.13\text{MPa}$, liquid fraction = 1 (not shown)). (a) Velocity [m/s] field (b) Pressure field [Pa]

5.2.6 Effect of cavitation on the jet

In the previous sections calculations for a cavitating jet where performed with a LES calculation method. In these calculations it was seen that cavitation starts to develop in the under-pressures of the vortices. These cavitation bubbles affect the flow of the jet. In this section the effects of cavitation on the jet are discussed.

The first and main difference in the cavitating jet in comparison with the single phase turbulent jet is seen in the velocity field of the jet. In Figure 5.16 and Figure 5.17 this difference is depicted. In the left image the velocity field of the single phase turbulent jet is shown and in the right image the velocity field of the cavitating jet is shown. It is clearly visible that the velocity, with respect to the distance from the nozzle, in the center of the cavitating jet is
higher than the velocity of the non-cavitating jet.

\[
Q = \frac{1}{2}||\Omega||^2 - |S|^2
\]  

(5.1)

The second difference can be found in the cross section of the vorticity. These are shown in Figure 5.19. It can be seen that in the case of a non-cavitating jet there are vortices everywhere whereas in the cavitating jet it is clearly visible that the vorticity is strongly reduces inside the potential core which now resides inside a cone of cavitation. This reduction of vorticity reduces the mixing with ambient water which can explain the rise of the velocity in the potential core.

The last difference can be seen in the size of the vortices. To depict this change in size the Q-criterion is used. The Q-criterion defines the vortex as a spatial region.
5.2. LES SIMULATIONS

Where $\Omega$ is the magnitude of the vorticity and $S$ is the mean strain rate. The Q-criterion has been applied to both the single phase turbulent jet as the cavitating jet at a threshold level of $10\text{e}+6$. In Figure 5.20 it can be seen that if cavitation is present the vortex structures are bigger than if no cavitation is present.

![Figure 5.20: Isocontour for the Q-criterion at a threshold level of $10\text{e}+6$ visualizing the vortices. On the left hand side the contour for the single phase jet is shown, on the right hand side the cavitating jet.](image)

5.2.7 Comparison with experiments

The increase of the velocity in the jet leads to an increase in the stagnation pressure of the jet. To compare these different calculations with experimental tests the stagnation pressure at a distance of 12 nozzle diameters was recorded. In appendix C it can be found how this is done. In Figure 5.21 the results of these samples are presented.

In the above figures it is seen that trends measured by Shen et al. (1988) are followed by the central and the blended (99 central, 1 upwind) schemes. The other blend (90 percent central, 10 percent upwind) factor and the full upwind calculations give a stagnation pressure that is even lower than that of a non-cavitating jet. This is due to the large amount of air bubbles that is present in the core of the jet, as can be seen in Figure 5.13. Which gives large decrease in density and thus in stagnation pressure.
5.2. LES SIMULATIONS

Figure 5.21: Stagnation pressures from the Simulations recorded at $12Dn \ (P_a = 0.13\text{Mpa})$

Figure 5.22: measurements by Nobel and Talmon (2012)

5.2.8 Discussion

In this section the results for the cavitating jet simulated with an LES calculation have been shown and discussed. It was seen that with a LES calculation method it is possible to simulate cavitation phenomena of a cavitating jet. This is due to the fact that with an LES technique the eddies and corresponding low pressures in the center of the eddies are simulated as well. It is seen that the pressures within these eddies are low enough to induce cavitation.

It is also shown in the results that the jets simulated follow several trends observed in empirical test such as: The decrease of cavitation inception in an environment with an higher ambient pressure and a decrease, or absence, of cavitation with a lower jet pressure. Which both lead to an higher cavitation number.
Chapter 6

Conclusion

In this thesis the possibility to simulate a cavitating jet with an existing CFD software package was investigated. The calculations were performed with the open-source software package OpenFOAM. Within OpenFOAM 2 solvers where found to be capable of solving for cavitating flows. Due to the fact that dissolved air content and nuclei diameter is an important parameter the cavitation model of (Schnerr and Sauer, 2001) was used to solve for the cavitation. Within OpenFOAM This cavitation model is found within the interPhaseChangeFoam solver.

The first set of calculations were performed with a Reynolds-Average Navier-Stokes (RANS) method on different geometries. The first geometry is based on the assumption that the acceleration of the ambient fluid would lead to under-pressures that induces cavitation. The second geometry is based on the assumption that the cavitation develops in the nozzle of the jet and is then transported with the jet fluid into the ambient fluid. With both geometries it was seen that with multiple settings no cavitation layer exists around the jet. This indicates that the pressure drop due to the acceleration of the ambient fluid is not enough to induce cavitation and that interPhaseChangeFoam is not capable of solving a cavitating jet with a RANS method.

The second set of simulations were performed with the assumption that the cavitation develops within the vortices produced by the shear between the ambient fluid and the jet fluid. In order to capture these vortices the second set of calculations were performed with a Large Eddy Simulation (LES) method. The results of these simulations showed cavitation bubbles present around the jet. These bubbles start to develop within the core of the vortices and affect the flow of the jet. The presence of the cavitation bubbles hinders the exchange of momentum with the ambient fluid resulting in a longer potential core. The longer potential core gives rise to the stagnation pressures of the jet.
To support this observation the stagnation pressures of the different simulations were recorded. It could be observed from these measurements that in case of an central LES simulation the stagnation pressure at a distance of 12 Dn was found to follow the same trends as observed in experimental test.

This leads to the conclusion that it is possible to simulate cavitating phenomena that occur within a cavitating jet with the interPhaseChangeFoam solver of OpenFOAM and an LES simulation technique.
Bibliography


E. Hutli, M. Nedeljkovic, and V. Ilic. Visualization of submerged cavitating jet: Part two–influences of hydrodynamic conditions, nozzle geometry and


**List of Figures**

2.1 Two different types of cavitation .................. 4
2.2 Turbulent jet cross section .......................... 5
2.3 Cavitating Jet ....................................... 8
2.4 influence cavitation number .......................... 10
2.5 Steady state void fraction of a 2D simulated cavitating jet as obtained by Peng and Shimizu (2013) at different cavitation numbers. The flow is from the left to the right .......... 16

3.1 Geometry of the 2D domain used for the simulations of the turbulent jet .............................. 21
3.2 Details of the mesh refinement levels for the turbulent jet geometry ................................. 22
3.3 Geometry for the case with an inlet tube ...................... 23
3.4 Mesh refinement levels for the mesh of geometry 1 ............... 24
3.5 Cross section of the 3D mesh ........................... 24
3.6 Mesh refinement levels for the mesh of geometry 2 .............. 25
3.7 Geometry for the Peng case ........................... 26
3.8 Mesh refinement levels for the mesh of geometry 3 .............. 27

4.1 centerline velocity ..................................... 32
4.2 Velocity profile ...................................... 33
4.3 Y-velocity ........................................... 34
4.4 Mesh dependency ...................................... 35

5.1 Liquid fraction for RANS case 1 ....................... 38
5.2 liquid fraction of a cross section for RANS case 2 ............. 39
5.3 Liquid fraction of a cross section of RANS case 3 ............. 40
5.4 Liquid fraction for RANS case 4 ........................ 41
5.5 Liquid fraction for RANS case 5 ........................ 42
5.6 Liquid fraction for RANS case 6 ........................ 44
5.7 Pressure in kPa for RANS case: (σ = 0.04, \( P_a = 0.13 \text{MPa} \)) ........ 45
<table>
<thead>
<tr>
<th>Figure Number</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.8</td>
<td>Pressure in kPa for RANS case:($\sigma = 0.04, P_a = 0.13\text{MPa}$)</td>
<td>45</td>
</tr>
<tr>
<td>5.9</td>
<td>comparison RANS and LES velocity fields and pressure fields</td>
<td>47</td>
</tr>
<tr>
<td>5.10</td>
<td>Liquid fraction for LES case 1</td>
<td>48</td>
</tr>
<tr>
<td>5.11</td>
<td>Liquid fraction for LES case 2</td>
<td>49</td>
</tr>
<tr>
<td>5.12</td>
<td>Liquid fraction for LES case 3</td>
<td>50</td>
</tr>
<tr>
<td>5.13</td>
<td>Liquid fraction for LES case 4</td>
<td>51</td>
</tr>
<tr>
<td>5.14</td>
<td>Liquid fraction for LES case 5</td>
<td>52</td>
</tr>
<tr>
<td>5.15</td>
<td>Liquid fraction for LES case 6</td>
<td>52</td>
</tr>
<tr>
<td>5.16</td>
<td>Velocity field in $m/s$ for LES case 6.</td>
<td>52</td>
</tr>
<tr>
<td>5.17</td>
<td>Velocity field in $m/s$ for the non-cavitating jet</td>
<td>52</td>
</tr>
<tr>
<td>5.18</td>
<td>results for LES case 7</td>
<td>53</td>
</tr>
<tr>
<td>5.19</td>
<td>Vorticity field for the turbulent single phase jet versus the cavitating jet. The single phase jet is depicted on the left side, the cavitating jet on the right side.</td>
<td>54</td>
</tr>
<tr>
<td>5.20</td>
<td>Isocontour for the Q-criterion at a threshold level of $10e+6$ visualizing the vortices. On the left hand side the contour for the single phase jet is shown, on the right hand side the cavitating jet.</td>
<td>55</td>
</tr>
<tr>
<td>5.21</td>
<td>Stagnation pressures from the Simulations recorded at $12Dn (P_a = 0.13\text{MPa})$</td>
<td>56</td>
</tr>
<tr>
<td>5.22</td>
<td>measurements by Nobel and Talmon (2012)</td>
<td>56</td>
</tr>
<tr>
<td>B.1</td>
<td>Original RNG k-$\epsilon$ model</td>
<td>69</td>
</tr>
<tr>
<td>B.2</td>
<td>Original RNG k-$\epsilon$ model</td>
<td>70</td>
</tr>
<tr>
<td>B.3</td>
<td>adjusted RNG k-$\epsilon$ model</td>
<td>71</td>
</tr>
<tr>
<td>B.4</td>
<td>adjusted RNG k-$\epsilon$ model</td>
<td>72</td>
</tr>
<tr>
<td>C.1</td>
<td>sampling lines used to record the stagnation pressures</td>
<td>73</td>
</tr>
</tbody>
</table>
List of Tables

2.1 Measurements on cavitating jets by Ooi (1985) showing the influence of nozzle diameter and dissolved air content on the cavitation number at which cavitation is observed, at an ambient pressure of $0.13 \text{MN/m}^2$ ........................................ 9

3.1 Number of cells and minimum cell size for the turbulent jet geometry ....................................................... 22

3.2 Number of cells and minimum cell width for the different meshes for geometry 1 ........................................ 23

3.3 Number of cells and minimum cell width for the different meshes for geometry 2 ........................................ 25

3.4 Number of cells and minimum cell width for the different meshes for geometry 3 ........................................ 26

4.1 RANS turbulence models ........................................... 31

4.2 Entrainment coefficients ........................................... 36

5.1 Different cases for the RANS calculations within interPhase-ChangeFoam ...................................................... 37

5.2 Different cases for the LES calculations within interPhase-ChangeFoam. The fractions associated with the blended schemes represent the amount of central discretisation. ............... 46
Appendix A: Entrainment

In chapter 3 entrainment has been discussed and the entrainment of obtained with various turbulence models were compared. This comparison is obtained by performing the following calculation.

First the velocity profiles of the different where sampled from the different simulations at the same places. To obtain the volume flux the velocity profile is integrated using the trapezoidal rule (matlab command 'Trapz').

When the volume fluxes of the jet are determined at the different locations the volume fluxes are subtracted from one another. Then they are divided by the distance to obtain the gradient \( \frac{dQ}{dS} \).

This gradient is equal to (for a plane jet):

\[
\frac{dQ}{dS} = \alpha_{ent} \cdot f1 \cdot 2L_n \cdot u_0 \sqrt{\frac{h_n}{s}}
\]  

(A.1)

Where \( f1 \) is a coefficient between 2.37 and 2.45 (Fisher et al., 1979). Filling in the variables in equation A.1 and dividing by them gives \( \alpha_{ent} \).
Appendix B: Adjustment
turbulence model

Figure B.1: Original RNG k-ε model
Figure B.2: Original RNG k-ε model
Figure B.3: adjusted RNG k-ε model: the additions to the model are highlighted
Figure B.4: adjusted RNG k-ε model: the additions to the model are highlighted
Appendix C: Stagnation pressure calculation

To calculate the stagnation pressure of the jet at a certain distance from the jet nozzle additional calculations must be performed. It is known that the stagnation pressure is equal to the dynamic pressure and is defined by

\[ P_s = 0.5 \times \rho_m \times v^2 \]  

(C.1)

Where \( v \) is the velocity in the direction of the jet. In a LES simulation the eddies are present and thus there are velocity fluctuations present. In order to get a good result for the mean velocity at the point of measurement it is preferred that the measurement is performed in time and a time average of the sample can be taken.

However the run time was not long enough to obtain a statistically stable time average and thus another method was used to estimate the stagnation pressure of the jet.

Figure C.1: sampling lines used to record the stagnation pressures

Four sampling lines were drawn across the center of the jet with a length of 5mm and 20 measurement points each. This is shown in Figure C.1. The
80 velocity samples generated by this where individually substituted into Equation C.1.

The volume fraction of water and air influence the density of the fluid at the point of measurement. Therefore the volume fraction of water was also sampled at the same points as the velocity. This volume fraction was inserted into Equation C.2

$$\rho_m = \alpha_l \cdot \rho_l + (1 - \alpha_l) \cdot \rho_{hv}$$  \hspace{1cm} (C.2)

This mixture density was then substituted into Equation C.1 for the corresponding velocity. This procedure gave 80 stagnation pressures over the center of the jet. Adding them up and dividing by 80 gives an average stagnation pressure which are used for Figure 5.21.