Numerical investigation on the cavitating flow in a waterjet pump
Marina Olsson

Department of Applied Mechanics
Division of Fluid Dynamics
CHALMERS UNIVERSITY OF TECHNOLOGY
Göteborg Sweden, 2008

Master’s Thesis [2008 : 44]
Numerical investigation on the cavitating flow in a waterjet pump

Master’s Thesis

Marina Olsson

Department of Applied Mechanics
Division of Fluid Dynamics
Chalmers University of Technology
Göteborg, Sweden, 2008
Numerical investigation on the cavitating flow in a waterjet pump
Master’s Thesis
Marina Olsson

© MARINA OLSSON, 2008

Master’s Thesis 2008:44
issn 1652-8557

Department of Applied Mechanics,
Division of Fluid Dynamics
Chalmers University of Technology
SE-412 96 Göteborg, Sweden
Phone +46-(0)31-7721400
Fax: +46-(0)31-180976

Printed at Chalmers Reproservice
Göteborg, Sweden 2008
Abstract

For waterjet systems operating in marine ships cavitation is a phenomenon that often occur. The presence of vapor in the flow affects the performance of the pump and as the cavity grows the pump efficiency drastically reduces to a level where the pump cannot operate normally. Due to this influence on the pump performance it is of main interest to be able to predict the behaviour of the cavitation process. At Rolls Royce Hydrodynamic Research Center in Kristinehamn experiments have been performed to observe the cavitating flow in such a waterjet system. With growing demand of detail information and the ability to run faster simulations, the desire of a reliable numerical tool has increased. The aim of this work is therefore to analyse and compare different numerical cavitation models and to find a model that can be used as a complement to experiments. First, four different cavitation models have been used for solving the cavitating flow around a hydrofoil. This rather simple geometry gives fast converged solutions and hence, is a fast way of testing the different models. Two of the tested cavitation models gave promising results and have therefore been used for the simulation of the flow in a pump to a waterjet. The efficiency calculated from the solution obtained with each model has been compared to experimental data and it is concluded that one of the cavitation models shows good agreement.

Keywords: cavitation, cavitation models, computational fluid dynamics, NACA 0015, waterjet, mixed flow pump
Acknowledgement

Working with this project has given me a lot of experience in the field of computational fluid dynamics. It has been a fun and demanding time with many challenges. I have been privileged to work close to talented people whose help, suggestions and encouragement have helped me find solutions. I would like to express my gratitude to the employees of the fluid mechanics group at Epsilon High Tech in Gothenburg. Special thanks are given to my supervisor, Dr. Jonas Bredberg, who has been a great support throughout this work.

I would also like to thank the staff at Rolls Royce Hydrodynamic Research Center, especially Jonas Eriksson who has always taken time to answer my questions. I have furthermore to thank my examiner, Prof. Lars Davidson, head of the Division of Fluid Dynamics at Chalmers University of Technology for his help and support.

Last but not least, I would like to thank my family whose supporting love has helped me to complete this work.
<table>
<thead>
<tr>
<th><strong>Roman</strong></th>
<th><strong>Greek</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td>$D$</td>
<td>$\Gamma$</td>
</tr>
<tr>
<td>$E$</td>
<td>$\phi$</td>
</tr>
<tr>
<td>$L$</td>
<td>$\alpha$</td>
</tr>
<tr>
<td>$M$</td>
<td></td>
</tr>
<tr>
<td>$Q$</td>
<td></td>
</tr>
<tr>
<td>$R$</td>
<td></td>
</tr>
<tr>
<td>$R_e$</td>
<td></td>
</tr>
<tr>
<td>$R_c$</td>
<td></td>
</tr>
<tr>
<td>$S$</td>
<td></td>
</tr>
<tr>
<td>$T$</td>
<td></td>
</tr>
<tr>
<td>$U_i$</td>
<td></td>
</tr>
<tr>
<td>$U_\infty$</td>
<td></td>
</tr>
<tr>
<td>$V$</td>
<td></td>
</tr>
<tr>
<td>$c$</td>
<td></td>
</tr>
<tr>
<td>$f$</td>
<td></td>
</tr>
<tr>
<td>$k$</td>
<td></td>
</tr>
<tr>
<td>$k$</td>
<td></td>
</tr>
<tr>
<td>$l$</td>
<td></td>
</tr>
<tr>
<td>$\dot{m}$</td>
<td></td>
</tr>
<tr>
<td>$n$</td>
<td></td>
</tr>
<tr>
<td>$n_0$</td>
<td></td>
</tr>
<tr>
<td>$p$</td>
<td></td>
</tr>
<tr>
<td>$t_\infty$</td>
<td></td>
</tr>
<tr>
<td>$u_i$</td>
<td></td>
</tr>
<tr>
<td>$u'_i$</td>
<td></td>
</tr>
<tr>
<td>$v$</td>
<td></td>
</tr>
<tr>
<td>$q_i$</td>
<td></td>
</tr>
</tbody>
</table>
Abbreviations

CFL Courant, Friedrichs and Lewy number, CFL = |ui|Δt/|Δxi|
We Weber number, We = ρu^2D/S

Subscripts

l Liquid
v Vapor
g Non condensable gases
1 Introduction

Cavitation is the formation of vapor within a liquid due to decrease of pressure below the saturated vapor pressure. It can be observed in devices such as turbomachinery and hydraulic constructions. When the velocity is increased there may be areas where the pressure drops below the vaporization pressure and cavitation is developed. If these cavitating areas grow, the vapor/liquid structures can affect the performance of a pump significantly. As the vapor reaches a higher pressure region the cavity collapses, a phenomenon strongly connected to noise and vibration. These drawbacks make cavitation an important issue in turbomachinery design and operation.

1.1 Background

Waterjet propulsion systems in marine ships are exposed to high water flow velocities and the probability for cavitation to occur is high. With evaporation occurring in the flow the pump efficiency decreases slowly before it drops drastically. To avoid this efficiency drop during pump operation it is important to be able to predict when it occurs. At Rolls Royce Hydrodynamic Research Center experimental observations are used to determine the critical condition for this behaviour. However, as the advantages of numerical simulations have increased, it is desired to have a mathematical cavitation model that can predict the cavitating flow in a waterjet pump.

![Figure 1: Waterjet propulsion system, [10].](image-url)
1.2 Aim and method

The goal of this thesis is to analyse and compare different mathematical cavitation models in order to find a model that can predict the cavitating flow in a waterjet pump in a satisfying way. To reach this goal extensive literature studies of existing cavitation models will be needed. Computational fluid dynamics will then be used to solve the fluid equations and deliver a numerical solution.

1.3 Thesis outline

The first chapters will explain the basics of computational fluid dynamics and deal with the physics behind cavitation. Having this as a base a number of mathematical models for cavitating flows can be derived. The next part of this thesis is a numerical case study of the flow around a single hydrofoil, where four different cavitation models will be compared. This case study is then extended to the flow in a waterjet pump. Numerical simulations with use of two different cavitation models are performed and the solutions are compared with experimental data obtained from Rolls Royce Hydrodynamic Research Center in Kristinehamn.
2 Fluid dynamics

Fluid dynamics is the study of fluids in motion and the forces acting on them. In theory, there are fluid flows which are simple and easy to describe. However, practical flows are almost always varying and complex. This complexity is connected to different phenomena among which turbulence and cavitation are the main interests in this work.

2.1 Turbulence

A short and concise definition of turbulence is impossible to formulate. Instead a long list of properties related to turbulent flows can be made:

- Turbulent flows are three-dimensional.
- Turbulent flows are unsteady.
- Turbulent flows contain vortices. As a matter of fact, vortex stretching is the main mechanism that increases the turbulent intensity.
- Turbulent flows increase the mixing of the fluid properties, a process called turbulent diffusion.
- Turbulent flows fluctuates over a wide range of length and time scales.

Depending on application, the effects of turbulence may or may not be positive. In chemical applications where intense mixing is needed, turbulence can speed up the process with orders of magnitude. On the other hand, turbulence is not desired in a pump or a propeller. The mixing property results in increased frictional forces with the consequence of higher power requirements, [6].

2.2 Cavitation

Cavitation is the process of phase transition from liquid to vapor as a consequence of pressure decrease below the vapor pressure. A brief description of the main features of this process is given in this section.

2.2.1 Phase transition

The phenomenon of phase transition from liquid to vapor occur because the attractive force between the molecules are insufficient to counteract a force in the opposite direction. The attractive force is equal to $\partial \phi/\partial x$, where $\phi$ is the
potential energy of the molecular binding and $x$ is the distance between the molecules. The strength of $\phi$ varies with the distance $x$ according to Figure 2. At the distance $x_0$ the liquid is in equilibrium, and at $x_1$ the attractive force is at its maximum.

![Figure 2: Potential energy of a molecular binding as a function of the distance between the molecules, [1].](image)

The attractive force between molecules decreases as the number of nucleation sites in the fluid increases. Typical nucleation sites are micro-sized bubbles of non-condensable gases or surfaces on which cavitation bubbles can grow. The effect of these weaknesses is that less energy is needed to rupture the molecules. This energy can be received either from temperature increase or from pressure decrease, see Figure 3. Depending on energy source the phenomenon is called boiling (temperature increase) or cavitation (pressure decrease), [1].

As a measure of the resistance of the flow to cavitation, the dimensionless number $\sigma$, called the cavitation number, is introduced. The lower the cavitation number is, the more likely is cavitation to occur.

$$\sigma = \frac{p - p_v}{\frac{1}{2} \rho u^2}$$ (1)
2.2.2 Bubble dynamics

In the previous section, the initial formation of bubbles was considered. In order to understand the whole process of cavitation, the growth and collapse of such bubbles must be explained. The simplest case is to consider a single spherical bubble with the time-dependent radius $R(t)$ in an infinite domain. The temperature in the domain, $T_\infty$, is assumed to be constant and the pressure, $p_\infty$, assumed to be known, regulates the growth and collapse of the bubble. Using the equations for conservation of momentum and mass and considering evaporation or condensation at the bubble interface an equation which describes the dynamics of the bubble radius can be obtained as:

$$\frac{p_B - p_\infty}{\rho_l} = R \frac{D^2 R}{Dt^2} + \frac{3}{2} \left( \frac{DR}{Dt} \right)^2 + \frac{4\nu_l}{R} \frac{DR}{Dt} + \frac{2S}{\rho_l R} \tag{2}$$

This equation is known as the Rayleigh-Plesset equation and is the base of many cavitation models, which mathematically describe the dynamics of bubbles in a fluid flow, see Section 3.5. The complete derivation of the Rayleigh-Plesset equation is obtained in Appendix A, [1].
2.2.3 Cavitation types

Depending on the flow configurations different types of cavitation can be observed. There are two main categories into which the different cavitation types can be divided; attached cavitation and convected cavitation. Attached cavitation is when a part of the cavity is connected to a surface, e.g. sheet cavitation and tip vortex cavitation. Convected cavitation corresponds to a cavity, which is exclusively carried by the convecting flow, e.g. bubble cavitation and cloud cavitation.

Bubble cavitation: Individual bubbles are travelling with the convecting flow as they expand and collapse. This phenomenon occurs for low pressure gradient flows, corresponding to a flow around a hydrofoil at a low incidence angle (Figure 4).

Sheet cavitation: As the incidence angle of a hydrofoil increases a separated zone of vapor starts to take form. This cavity can be of different size. A sheet cavity that closes on the suction surface of the foil is called ”partial cavitation” (Figure 5) whereas ”supercavitation” covers the entire foil. Where the cavity closes and meets the surface a stagnation point appear and a part of the flow, the so called re-entrant jet,
Figure 5: Sheet cavitation on the upper surface (suction side) of a hydrofoil, [1].

is turned back towards the leading edge of the hydrofoil. As the re-entrant jet propagates along the foil surface the sheet cavity is shedding away from the hydrofoil.

Cloud cavitation: After the collapse of the sheet cavity the bubble density increases and a cavitation cloud moves downstream with a rotating motion (Figure 6). Cloud cavitation may be seen in other flows with temporal periodicity. In a pump this periodicity may occur due to fluctuations caused by the rotor-stator-interaction.

Vortex cavitation: Flows of high Reynolds number often contain regions of concentrated vorticity. In the vortex core the pressure is much smaller than in the rest of the flow, therefore this is a critical zone for cavitation. In pump flows this phenomenon can occur at the tip vortices and be seen as a helix traveling downstream (Figure 7).
2 Fluid dynamics

Figure 6: Cloud cavitation on a hydrofoil seen from above, [1].

Figure 7: Tip vortex cavitation, [1].
3 Mathematical model of fluids in motion

3.1 Governing equations

The mathematical basis for a model of fluid flow is the fundamental principles of conservation of mass, momentum and energy. For the analysis of these conservation laws the macroscopic properties of the fluid, such as velocity, pressure, density and temperature, are considered. To avoid microscopic influences of individual molecules of such properties the fluid is regarded as a continuum with the smallest element consisting of a significant large amount of molecules. This fluid element can be considered in cartesian coordinates as a cube with side length $\delta x$, $\delta y$ and $\delta z$ (Figure 8).

![Figure 8: A fluid element.](image)

The mathematical formulation of the change of mass, momentum and energy due to flux in and out of this element give the desired fluid equations. To express the fluxes the derivatives of the fluid properties need to be formulated. Due to the small size of the element a good enough approximation is reached using the first two terms in the Taylor expansion. As an example the $x$-derivative of the pressure at $(x - \delta x/2, y, z)$ can be expressed as:

$$\frac{\partial p(x - \delta x/2, y, z, t)}{\partial x} = p(x, y, z, t) - \frac{\partial p(x, y, z, t)}{\partial x} \cdot \frac{1}{2} \delta x$$  \hspace{1cm} (3)

The pressure in the center of the fluid element $(x, y, z)$ is here explicitly written as a function of space and time. This notation will from now on be omitted, a property in the volume center will only be denoted with the letter assigned to that property.
3.1.1 Mass conservation

The conservation of mass can be described as the balance between rate of increase of mass in a fluid element and the net flow rate of mass into a fluid element. The rate of increase of mass can be written as the time derivative of the product of density and element volume:

\[
\frac{\partial (\rho \delta x \delta y \delta z)}{\partial t} = \frac{\partial \rho}{\partial t} \delta x \delta y \delta z \tag{4}
\]

The mass flow rate into or out of the fluid element is the sum of the mass fluxes over each face. The mass flow across one face is given as the product of the flow vector and the face area. Figure 9 shows the mass flow vectors across each face and the net mass flux can be expressed as:

\[
\begin{align*}
\rho u - \frac{\partial (\rho u)}{\partial x} \cdot \frac{1}{2} \delta x & \quad \delta y \delta z - \left( \rho u + \frac{\partial (\rho u)}{\partial x} \cdot \frac{1}{2} \delta x \right) \delta y \delta z + \\
\rho v - \frac{\partial (\rho v)}{\partial y} \cdot \frac{1}{2} \delta y & \quad \delta x \delta z - \left( \rho v + \frac{\partial (\rho v)}{\partial y} \cdot \frac{1}{2} \delta y \right) \delta x \delta z + \\
\rho w - \frac{\partial (\rho w)}{\partial z} \cdot \frac{1}{2} \delta z & \quad \delta x \delta y - \left( \rho w + \frac{\partial (\rho w)}{\partial z} \cdot \frac{1}{2} \delta z \right) \delta x \delta y
\end{align*} \tag{5}
\]

Expression (4) and (5) can now be equated and divided by the element volume \(\delta x \delta y \delta z\) giving the final equation for the three-dimensional mass conservation in a compressible fluid as:

\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u)}{\partial x} + \frac{\partial (\rho v)}{\partial y} + \frac{\partial (\rho w)}{\partial z} = 0 \tag{6}
\]
or in a more compact way using tensor notation:

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

(7)

where \( i \) is a summation index. For an incompressible fluid the density is constant and the mass conservation equation, also known as the continuity equation, can be simplified to:

\[
\frac{\partial u_i}{\partial x_i} = 0
\]  

(8)

3.1.2 Momentum conservation

The momentum balance in a fluid element is based on Newton’s second law, \( F = ma \). The sum of forces acting on a fluid element must equal the rate of increase of momentum of that element. The rate of increase of momentum in each direction is given by:

\[
\rho \frac{Du}{Dt}, \rho \frac{Dv}{Dt} \text{ and } \rho \frac{Dw}{Dt}
\]  

(9)

where the operator \( D/Dt = \partial/\partial t + u_i \partial/\partial x_i \) is the material derivative of a fluid particle. The forces acting on a fluid element can be either body forces or surface forces. A typical representative of body forces is gravity and is usually represented by a source term in the momentum equation. Surface forces, on the other hand, results from pressure, \( p \), or viscous stresses, \( \tau_{ij} \), acting on each face of the fluid element (Figure 10).

![Figure 10: Stress components acting on a fluid element in x-direction.](image)

The total amount of surface forces in the \( x \)-direction is the summed product of each stress and the area on which the stress acts:
3 Mathematical model of fluids in motion

\[
\begin{align*}
&\left[ (p - \frac{\partial p}{\partial x} \cdot \frac{1}{2} \delta x) - \left( p + \frac{\partial p}{\partial x} \cdot \frac{1}{2} \delta x \right) \right] \delta y \delta z + \\
&\left[ (\tau_{xx} + \frac{\partial \tau_{xx}}{\partial x} \cdot \frac{1}{2} \delta x) - \left( \tau_{xx} - \frac{\partial \tau_{xx}}{\partial x} \cdot \frac{1}{2} \delta x \right) \right] \delta y \delta z + \\
&\left[ (\tau_{yx} + \frac{\partial \tau_{yx}}{\partial y} \cdot \frac{1}{2} \delta y) - \left( \tau_{yx} - \frac{\partial \tau_{yx}}{\partial y} \cdot \frac{1}{2} \delta y \right) \right] \delta x \delta z + \\
&\left[ (\tau_{zx} + \frac{\partial \tau_{zx}}{\partial z} \cdot \frac{1}{2} \delta z) - \left( \tau_{zx} - \frac{\partial \tau_{zx}}{\partial z} \cdot \frac{1}{2} \delta z \right) \right] \delta x \delta y
\end{align*}
\]

In the above expression some of the terms can be cancelled out. Division by the element volume gives the total force in \(x\)-direction per unit volume as:

\[
\frac{\partial(-p + \tau_{xx})}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z}
\]

With the body forces gathered in a source term \(S_x\) the momentum conservation in \(x\)-direction can now be written as:

\[
\rho \frac{D u}{D t} = \frac{\partial(-p + \tau_{xx})}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + S_x
\]

An analogous derivation can be applied to the \(y\)- and \(z\)-direction. Using the definition of the material derivative the three equations can be written in tensor notation as:

\[
\rho \frac{\partial u_i}{\partial t} + \rho u_j \frac{\partial u_i}{\partial x_j} = - \frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + S_i
\]

In addition to these equations a constitutive equation is needed for the viscous stresses \(\tau_{ij}\). For a Newtonian fluid these stresses are proportional to the rate of deformation of the fluid element giving:

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

Inserting this expression into the momentum equation gives the so called Navier-Stokes equations:

\[
\rho \frac{\partial u_i}{\partial t} + \rho u_j \frac{\partial u_i}{\partial x_j} = - \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] + S_i
\]
For incompressible flows the continuity equation \( \partial u_i / \partial x_i = 0 \) can be used to simplify Navier-Stokes equations to:

\[
\rho \frac{\partial u_i}{\partial t} + \rho u_j \frac{\partial u_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \mu \frac{\partial u_i}{\partial x_j} \right) + S_i \tag{16}
\]

### 3.1.3 Energy equation

The energy equation is derived from the first law of thermodynamics, which balances the rate of increase of energy of a fluid particle and the sum of the heat and work added to the particle. The rate of increase of energy is given by:

\[
\frac{DE}{Dt} \tag{17}
\]

where \( E \) is the specific energy of a fluid defined as the sum of internal and kinetic energy. The rate of work done on a fluid element by a surface force is the product of that force and the velocity component in the direction of the force. The calculated forces in equation (11) can be used to give the net rate of work done in the \( x \)-direction as:

\[
\left[ \frac{\partial (u(-p + \tau_{xx}))}{\partial x} + \frac{\partial (u\tau_{yx})}{\partial y} + \frac{\partial (u\tau_{zx})}{\partial z} \right] \delta x \delta y \delta z \tag{18}
\]

Analogous expressions for the rate of work done in the \( y \)- and \( z \)-direction can be written as:

\[
\left[ \frac{\partial (v\tau_{xy})}{\partial x} + \frac{\partial (v(-p + \tau_{yy}))}{\partial y} + \frac{\partial (v\tau_{zy})}{\partial z} \right] \delta x \delta y \delta z \tag{19}
\]

and

\[
\left[ \frac{\partial (w\tau_{xz})}{\partial x} + \frac{\partial (w\tau_{yx})}{\partial y} + \frac{\partial (w(-p + \tau_{zz}))}{\partial z} \right] \delta x \delta y \delta z \tag{20}
\]

The sum of these three expressions is the total rate of work done by surface stresses on a fluid particle. The work done by body forces is added as a source term \( S_E \) to the energy equation.

The rate of heat added to the particle is the sum of the product of the face area and the heat flux vector, \( q \), acting on it (Figure 11). The same approach as with the mass flow in the derivation of the continuity equation is used to give the total rate of heat added to a fluid particle per unit volume as:
The heat flux vector can be related to the local temperature gradient by use of Fourier’s law \( q_i = -k \frac{\partial T}{\partial x_i} \), where \( k \) is the heat coefficient. The total rate of heat can now be written as:

\[
- \frac{\partial q_i}{\partial x_i} = \frac{\partial}{\partial x_i} \left( k \frac{\partial T}{\partial x_i} \right) = -\frac{\partial q_i}{\partial x_i}
\]

All terms needed for the energy conservation are now derived and they can be put together to give the energy equation:

\[
\rho \frac{DE}{Dt} = -\frac{\partial (pu_i)}{\partial x_i} + \frac{\partial (u_i \tau_{ij})}{\partial x_j} + \frac{\partial}{\partial x_i} \left( k \frac{\partial T}{\partial x_i} \right) + S_E
\]

### 3.2 Solution of the governing equations

An exact analytical solution of the governing equations is only available for a few simple flows. For most flows the solution needs to be found numerically. The governing equations are integrated over a control volume (a fluid volume like the one illustrated in Figure 8) and are then discretized into a system of algebraic equations. For laminar flows this equation system can be solved iteratively. However, most flows are not laminar but turbulent. As has been explained in Section 2.1, turbulent flows contain variations over a wide range of length and time scales, which makes the solution of the equations difficult and expensive. Different numerical methods for capturing the effects of turbulence can be grouped in the following three categories:
3 Mathematical model of fluids in motion

**Direct numerical simulation:** The direct numerical simulation (DNS) solves the mean flow and all turbulent velocity fluctuations. The computational grid must be very fine in order to resolve the smallest scales. Therefore the computations are highly time consuming and this method is not used for industrial applications.

**Large eddy simulation:** This simulation resolves the mean flow and the large eddies. The method involves space filtering of the unsteady Navier-Stokes equations, which passes the large eddies. The effect of the smaller eddies are modeled using a subgrid-scale (SGS) model.

**Reynolds-averaged Navier-Stokes equations:** This method focus on the mean flow and the effects of turbulent fluctuations on the mean flow properties. The Navier-Stokes equations are time-averaged (or Reynolds-averaged), which leads to the appearance of an extra term. This term represents the interaction between various turbulent fluctuations and needs to be modeled. In this work this method is being used and will therefore be explained next.

### 3.3 Reynolds-averaged Navier-Stokes equations

With focus directed on the mean flow and effects of turbulent fluctuations on the mean flow properties the flow variables are decomposed into a mean component and a fluctuating component (Reynolds decomposition):

\[
\begin{align*}
  u_i &= U_i + u'_i \\
  p &= P + p'
\end{align*}
\]  

These expressions for the flow variables are inserted in the governing equations, which can be time averaged. When taking the time average of the flow variable there are certain rules which have to be followed:

- The time average of a fluctuating flow property is zero.
- The time average of a mean flow property is again the same mean flow property.
- The time average of a product of two fluctuating flow properties is zero only if they are independent properties. This rule will later on be applied on the product of two fluctuating velocities, which can not be assumed to be independent.
3 Mathematical model of fluids in motion

- The time average of the product of a mean flow property and a fluctuating flow property is zero.

Applying these rules to the incompressible continuity equation gives:

\[
0 = \frac{\partial u_i}{\partial x_i} = \frac{\partial}{\partial x_i}(U_i + u'_i) = \frac{\partial U_i}{\partial x_i} + \frac{\partial u'_i}{\partial x_i} = \frac{\partial U_i}{\partial x_i} \tag{26}
\]

A similar process can be carried out to get the incompressible momentum equations as:

\[
\rho \frac{\partial U_i}{\partial t} + \rho U_j \frac{\partial U_i}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \mu \frac{\partial U_i}{\partial x_j} \right) - \frac{\partial (\mu u'_i u'_j)}{\partial x_j} + S_i \tag{27}
\]

Comparing this equation with the unsteady Navier-Stokes equations (16), a new term can be identified. This term, \(-\partial(\mu u'_i u'_j) / \partial x_j\), involves products of the fluctuating velocities and is associated with the convective transport due to turbulent eddies. Because of these six new unknowns; \(\rho u'_i u'_i\), \(\rho u'_i u'_j\), \(\rho w'_i w'_i\), \(\rho u'_i v'_i\), \(\rho u'_i w'_i\) and \(\rho v'_i w'_i\), called the Reynolds stresses, the equation system is no longer closed and these terms need to be modeled.

### 3.4 Turbulence models

In order to model the new term in the Reynolds-averaged Navier-Stokes equations Boussinesq assumed that the effect of the Reynolds stresses can be represented as an increased viscosity:

\[
-\rho u'_i u'_j = \mu_t \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} \rho \delta_{ij} k \tag{28}
\]

where \(k\) is the turbulent kinetic energy:

\[
k = \frac{1}{2} w'_i w'_i \tag{29}
\]

and \(\mu_t\) is the turbulent viscosity which is an unknown function of flow variables and has to be modeled. The simplest way of doing this is to use a turbulent length scale, \(L\), and a turbulent velocity scale, \(v = \sqrt{k}\) and perform a dimensional analysis giving:

\[
\mu_t = C_\mu \rho v L \tag{30}
\]

where \(C_\mu\) is an empirical dimensionless constant (\(C_\mu = 0.09\)). The turbulent kinetic energy can be determined from a transport equation for \(k\). As a
new unknown, the turbulent dissipation, $\varepsilon$, appear in this equation. As a first approximation $\varepsilon$ is determined from $\varepsilon = \frac{k^3}{2L}$. However, a transport equation for the turbulent dissipation can be formulated, and the turbulent viscosity is then expressed as:

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

(31)

The transport equations for the turbulent kinetic energy, $k$, and for the turbulent dissipation, $\varepsilon$, can be written as:

$$\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho U_i k)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + 2\mu_t S_{ij} S_{ij} - \rho \varepsilon$$  

(32)

$$\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho U_i \varepsilon)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} 2\mu_t S_{ij} S_{ij} - C_{2\varepsilon} \frac{\varepsilon^2}{k}$$

(33)

where $S_{ij}$ is the viscous strain-rate defined as:

$$S_{ij} = \frac{1}{2} \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right)$$

(34)

The commonly used values of the four constants are $\sigma_k = 1.00$, $\sigma_\varepsilon = 1.30$, $C_{1\varepsilon} = 1.44$ and $C_{2\varepsilon} = 1.92$. This model is called the standard $k - \varepsilon$ model. As has been shown the turbulent viscosity, $\mu_t$, is determined from a single turbulence length scale, $L$. The calculated turbulent diffusion occurs only at that specific scale. To account for the effects of smaller scales Yakhot et al [18] developed the renormalization group (RNG). This RNG approach is a mathematical technique, which can be used to derive a turbulence model similar to the standard $k - \varepsilon$ model. The resulting difference lies in the transport equation for the turbulent dissipation, $\varepsilon$:

$$\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho U_i \varepsilon)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C^*_{1\varepsilon} \frac{\varepsilon}{k} 2\mu_t S_{ij} S_{ij} - C_{2\varepsilon} \frac{\varepsilon^2}{k}$$

(35)

where

$$C^*_{1\varepsilon} = C_{1\varepsilon} - \frac{C_{\mu} \eta^3 \left(1 - \eta / \eta_0 \right)}{1 + \beta \eta^3}$$

(36)

and
3 Mathematical model of fluids in motion

\[ \eta = \sqrt{2S_{ij} \cdot S_{ij} \frac{k}{\varepsilon}} \]  

(37)

In the \( k - \varepsilon \) RNG model the empirical constants are: \( C_\mu = 0.0845 \), \( \sigma_k = 0.7194 \), \( \sigma_\varepsilon = 0.7194 \), \( C_{1\varepsilon} = 1.42 \), \( C_{2\varepsilon} = 1.68 \), \( \eta_0 = 4.38 \) and \( \beta = 0.012 \).

3.4.1 Modified turbulent viscosity

The unsteady cyclic behaviour in a cavitating flow is strongly dependent of turbulence model. A standard two-equations RANS turbulence model cannot resolve this unsteadiness, the re-entrant jet is stopped before breaking the cavity interface and the oscillating behaviour of the cavity never starts. This makes the solution quasi-steady, i.e. the sheet cavity remains attached and is oscillating around a stable state. This restrained behaviour is probably due to an over-estimated diffusivity at the rear of the cavity. Using the \( k - \varepsilon \) model this problem can be prevented by modification of the turbulent viscosity. Coutier-Delgosha et al. [3] proposed a model that reduces the turbulent viscosity, \( \mu_t \), in the areas with low vapor volume fraction:

\[ \mu_t = f(\rho)C_\mu \frac{k^2}{\varepsilon} \]  

(38)

where

\[ f(\rho) = \rho_v + \left( \frac{\rho - \rho_v}{\rho_l - \rho_v} \right)^{10} (\rho_l - \rho_v) \]  

(39)

where \( \rho_v \) is the vapor density, \( \rho_l \) is the liquid density and \( \rho \) is the density of a mixture of liquid and vapor. Figure 12 shows the difference of the standard function \( f(\rho) = \rho \) and the modified function \( f(\rho) \).

3.5 Cavitation models

Computational modeling of cavitation has been pursued for years. Early studies based on potential flow theory are still used, but the use of Navier-Stokes equations has emerged the last decade. There are two main categories which separate studies of computational cavitation. The first one is the interface tracking method, where equations are solved for the liquid phase only. By using the assumption that the pressure inside the cavity is constant and equal to the vapor pressure of the corresponding liquid, the liquid-vapor interface can be tracked. The grid is continuously regenerated in order to match the calculated cavity shape. These models are suitable for sheet cavities, but limited when other types of cavities appear. In addition, the use is
restricted to 2D flows only because of the difficulties of tracking complex 3D interfaces. The second category is the most common one and based on the assumption of a single fluid with varying density. The key challenge for these models is how to define the variable density. There are two main approaches of doing this:

I: To account for the cavitation dynamics a transport equation for the vapor mass fraction is being introduced. The rate of evaporation and condensation in the source term are modeled and the density is then computed as a weighted sum of the vapor density and the liquid density.

II: The density is calculated according to a barotropic state law $\rho = \rho(p)$, which describes the density in the two pure phases (liquid and vapor) and in the two-phase fluid.

A detailed description of these two methods will be presented in the following chapters.

3.6 Cavitation model based on a transport equation for vapor mass fraction (I)

In this method the density, $\rho$, of the fluid is calculated as a weighted sum of the vapor density and the liquid density. As a weight the vapor volume fraction $\alpha$ is introduced:
3 Mathematical model of fluids in motion

\[ \alpha = \frac{V_v}{V_l + V_v} \]  
\[ \rho = \alpha \rho_v + (1 - \alpha) \rho_l \]  
(40)  
(41)

The subscripts \( v \) and \( l \) represent the vapor and liquid phase respectively. With this additional unknown, \( \alpha \), the equation system with mass and momentum transport is no longer closed, i.e. there are more unknowns than equations. Therefore a transport equation for \( \alpha \), or rather for the vapor mass fraction \( f \), is introduced.

\[ \alpha \equiv f \frac{\rho}{\rho_v} \]  
(42)

\[ \frac{\partial}{\partial t}(\rho f) + \frac{\partial}{\partial x_i}(\rho u_i f) = \frac{\partial}{\partial x_i}(\Gamma \frac{\partial f}{\partial x_i}) + R_e - R_c \]  
(43)

where the terms from left to right represent unsteady term, convective transport, diffusive transport (\( \Gamma \) is the diffusion coefficient) and phase transfer rates, i.e. vapor generation (evaporation) and condensation rates. These last two terms can be functions of flow parameters and fluid properties and are of main interest when modeling cavitation. In the literature there are a large amount of models for these phase transfer rates, models of which some will be discussed further on.

3.6.1 Singhal’s cavitation model

The first model which will be presented is the so called ”full cavitation model” developed by Singhal et al., [12]. This model involves two phases and a certain fraction of non-condensable gases, whose mass fraction is known in advance. Singhal’s cavitation model is the standard one used in Fluent version 6.3 and the focus is on bubble dynamics. In section 2.2.2 the equation for bubble growth and collapse, the Rayleigh-Plesset equation, was given as:

\[ \frac{p_B - p_\infty}{\rho_l} = R \frac{D^2 R}{D t^2} + \frac{3}{2} \left( \frac{D R}{D t} \right)^2 + \frac{4 \mu_l}{R} \frac{D R}{D t} + \frac{2S}{\rho_l R} \]  
(44)

In Singhal’s model, however, the viscous damping and the surface tension are ignored, so is the second-order derivative of \( R \). With these approximations the simplified Rayleigh-Plesset equation can be written as:

\[ \frac{D R}{D t} = \sqrt{\frac{2}{3} \frac{p_B - p}{\rho_l}} \]  
(45)
To obtain formulations for the source terms $R_c$ and $R_e$ in the transport equation for the vapor mass fraction (43), the continuity equations for the separate phases and the mixture are used.

Liquid phase:

$$\frac{\partial}{\partial t}[(1 - \alpha)\rho_l] + \frac{\partial}{\partial x_i}[(1 - \alpha)\rho_l u_i] = -\dot{m}$$  \hspace{1cm} (46)

Vapor phase:

$$\frac{\partial}{\partial t}(\alpha\rho_v) + \frac{\partial}{\partial x_i}(\alpha\rho_v u_i) = \dot{m}$$  \hspace{1cm} (47)

Mixture:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i) = 0$$  \hspace{1cm} (48)

where $\dot{m}$ is the net phase change rate $R_e - R_c$. The vapor volume fraction, $\alpha$, is defined as:

$$\alpha = n \frac{4}{3} \pi R^3$$  \hspace{1cm} (49)

where $n$ is the number of vapor bubbles per unit volume of flowing fluid. The continuity equation for the vapor phase (47) can be rewritten with the material derivative as:

$$\frac{D\alpha}{Dt} + \alpha \frac{\partial u_i}{\partial x_i} = \frac{\dot{m}}{\rho_v}$$  \hspace{1cm} (50)

To get an expression for $\partial u_i / \partial x_i$ the continuity equation for vapor (47) is divided by the constant vapor density $\rho_v$. In the same way the continuity equation for liquid (46) is divided by the constant liquid density $\rho_l$. The two new equations can now be added and the result is:

$$\frac{\partial u_i}{\partial x_i} = \dot{m} \left( \frac{1}{\rho_v} - \frac{1}{\rho_l} \right)$$  \hspace{1cm} (51)

By inserting this in (50) and using the definition of the mixture density (41) the phase transfer rate can be written as:

$$\dot{m} = \frac{\rho_v \dot{\rho}_l}{\rho} \frac{D\alpha}{Dt}$$  \hspace{1cm} (52)

The total derivative of $\alpha$ can easily be calculated from equation (49) to:
3 Mathematical model of fluids in motion

\[
\frac{D\alpha}{Dt} = \frac{n}{3} \pi R^2 \frac{DR}{Dt} = \frac{3\alpha DR}{R \cdot Dt}
\]

(53)

The expression for \( \dot{m} \) in equation (52) can now be written as:

\[
\dot{m} = \frac{3\alpha}{R} \cdot \frac{\rho_v \rho_l}{\rho} \sqrt{\frac{2 p_v - p}{3 \rho_l}}
\]

(54)

where the expression for the derivative of \( R \) in equation (45) has been inserted. The bubble pressure \( p_B \) has been approximated by the vapor pressure \( p_v \). All terms in (54) are known constants or dependent variables except the bubble radius \( R \). A typical bubble radius taken to be equal to the maximum possible is therefore chosen to approximate \( R \). The maximum bubble radius is determined by the Weber number, which is a balance between aerodynamic drag and surface tension:

\[
\text{We} = \frac{\rho v^2 D}{S}
\]

(55)

where \( v \) is a characteristic velocity scale, \( D \) a characteristic length scale and \( S \) is the surface tension. Inserting the bubble radius, \( R \), as the characteristic length scale and the relative velocity between the two phases, \( v_{rel} \), as the characteristic velocity scale the following expression for the bubble radius can be obtained:

\[
R = \frac{C_R \cdot \text{We} \cdot S}{\rho_l v_{rel}^2}
\]

(56)

where \( C_R \) is a dimensionless constant. Inserting this expression into (54) and gathering the constants gives:

\[
\dot{m} = C \frac{\rho l v_{rel}^2}{S} \cdot \frac{\rho_v \rho_l}{\rho} \sqrt{\frac{2 p_v - p}{3 \rho_l}}
\]

(57)

To distinguish between expressions for evaporation and condensation (remember that \( \dot{m} = R_e - R_c \)) the fact that each phase change rate should be proportional to the volume fractions of the donor phase is used. For the condensation rate, \( \alpha \) in equation (57) is exchanged to the liquid volume fraction \( \gamma = f_l \rho / \rho_l \) and for the cavitation rate, \( \alpha = f_v \rho / \rho_v \) is inserted. The evaporation and condensation rates can now be written separately as:

\[
R_e = C_e \frac{v_{rel}^2}{S} \rho_l \rho_v \sqrt{\frac{2 p_v - p}{3 \rho_l}} f_l
\]

(58)
3 Mathematical model of fluids in motion

\[ R_c = C_c \frac{v_{rel}^2}{S} \rho_l \rho_l \sqrt{\frac{2}{3} \frac{p - p_v}{\rho_l} f_v} \]  (59)

Singhal argues that a linear rather than a quadratic velocity dependence is more relevant, hence the square of \( v_{rel} \) is ignored. Singhal also assumes that the relative velocity between the phases is of the same order as the turbulence velocity fluctuations. Therefore \( v_{rel} \) is approximated by \( \sqrt{k} \).

The full cavitation model also accounts for the effect of turbulent pressure fluctuations. This is implemented by simply raising the vapor saturation pressure to:

\[ p'_v = p_v + \frac{p'_{turb}}{2} \]  (60)

where \( p'_{turb} = 0.39 \rho k \) according to Singhal. The heterogeneous nucleation of non condensable gases is also included in Singhal’s cavitation model. The liquid mass fraction is then obtained by subtracting the mass fraction for the vapor and for non condensable gases \( (f_g) \) from the total mass fraction:

\[ f_l = 1 - f_v - f_g \]  (61)

The final form of the evaporation and condensation rates can now be written as:

\[ R_e = C_e \frac{\sqrt{k}}{S} \rho_l \rho_l \sqrt{\frac{2}{3} \frac{p'_v - p}{\rho_l} (1 - f_v - f_g)} \]  (62)

\[ R_c = C_c \frac{\sqrt{k}}{S} \rho_l \rho_l \sqrt{\frac{2}{3} \frac{p - p'_v}{\rho_l} f_v} \]  (63)

The recommended values of the empirical constants \( C_e \) and \( C_c \) are 0.02 and 0.01 respectively, [12].

3.6.2 Sauer’s cavitation model

Another model based on bubble dynamics is Sauer’s cavitation model, [13][8]. As in the full cavitation model the following simplified version of the Rayleigh-Plesset equation is used.

\[ \frac{DR}{Dt} = \sqrt{\frac{2}{3} \frac{p_B - p}{\rho_l}} \]  (64)
The derivation made for the full cavitation model to get the expression for
the net phase change rate in equation (52) can also be applied here. The
main difference between Singhal’s and Sauer’s model is that Sauer assumes
that a constant number of vapor bubbles per unit volume of flowing liquid,
\( n_0 \), are present in the fluid during the entire computation. The vapor volume
fraction \( \alpha \) is therefore defined as:

\[
\alpha = \frac{n_0 \frac{4}{3} \pi R^3}{1 + n_0 \frac{4}{3} \pi R^3}
\]  

with the total derivative:

\[
\frac{D\alpha}{Dt} = \frac{3\alpha}{R} \frac{DR}{Dt} (1 - \alpha)
\]  

The phase transfer rate can now be written as:

\[
\dot{m} = \frac{\rho_v \rho_l}{\rho} 3\alpha (1 - \alpha) \frac{DR}{Dt}
\]  

To apply the simplified Rayleigh-Plesset equation (64) on both bubble growth
and collapse the following formulation is used for the derivative of the bubble
radius:

\[
\frac{DR}{Dt} = \text{sign}(p_v - p) \sqrt{\frac{2}{3} \frac{|p_v - p|}{\rho_l}}
\]  

where the bubble pressure has been set to the vapor saturation pressure.

With this expression inserted in equation (67) the final form of the phase
change rate is:

\[
\dot{m} = \frac{\rho_v \rho_l}{\rho} 3\alpha (1 - \alpha) \text{sign}(p_v - p) \sqrt{\frac{2}{3} \frac{|p_v - p|}{\rho_l}}
\]  

To eliminate the radius of the bubbles, \( R \) can be computed from equation
(65) to:

\[
R = \left( \frac{3\alpha}{(1 - \alpha)n_0 4\pi} \right)^{1/3}
\]

The only model constant is the number of bubbles, \( n_0 \).
3.6.3 Kunz’ cavitation model

The mass transfer in Kunz’ model [8] is based on two different strategies for creation and destruction of vapor. Vapor production, $R_e$, is modeled proportional to the amount by which the pressure is below the vapor pressure. Vapor condensation, $R_c$, on the other hand, is modeled using a third order polynomial function of the liquid volume fraction $\gamma$. With these assumptions the source terms can be written as:

$$R_e = C_{prod} \cdot \rho_v \alpha_l \min(0, p - p_v)$$

$$R_c = C_{dest} \cdot \rho_v \alpha_l^2 (1 - \alpha_l)$$

The empirical constant $t_\infty$ is a time scale defined as the ratio between the characteristic length scale to the reference velocity scale ($t_\infty = \frac{l}{U_\infty}$) and is used to non-dimensionalize both the production and the destruction terms. Both $C_{prod}$ and $C_{dest}$ are empirical constants. The recommended value for these constants differ between articles. In this work $C_{prod} = C_{dest} = 100$ has been used.

3.7 Barotropic cavitation model (II)

Due to the local presence of two phases the sonic speed reduces dramatically at the cavitation interface and discontinuities such as shock waves occur in the flow. The barotropic model includes the consequence of these effects. This model does not introduce the vapor volume fraction and hence, the additional equation for mass fraction is not needed. Instead, the density of the fluid is computed from a barotropic state law, Figure 13.

When the pressure is higher than the vapor pressure the fluid is supposed to be purely liquid and the density is defined by the Tait equation:

$$\frac{\rho}{\rho_{ref}} = n \sqrt{\frac{p + p_0}{p_{ref}^T + p_0}}$$

where $\rho_{ref} = \rho_0$ and $p_{ref}^T = p_{outlet}$ are reference density and pressure. For water $p_0 = 3 \cdot 10^8$ and $n = 7$. The pure phase of vapor is defined when the pressure is lower than the vapor pressure. The density can then be computed from the ideal gas law:

$$p = \rho RT$$
or when neglecting thermal effects, $p/\rho = \text{const}$. The density of the state between these two limits (mixture of vapor and liquid) is calculated from a smooth curve connecting the two pure phases. This curve is mainly characterized by its maximum slope $1/c_{\text{min}}^2$, where $c_{\text{min}}^2 = \partial p/\partial \rho$. For cold water $c_{\text{min}} \approx 1.5 \text{ m/s}$, vapor pressure $p_v \approx 2300 \text{ Pa}$ and $\Delta p_v \approx 3000 \text{ Pa}$, [4]. The evaporation and condensation processes are treated implicitly by this state law, and hence, no more equations are needed.

When attaching this barotropic cavitation model to the numerical solver the barotropic state law has to be discretized. The equation that describes the mixture state can then be approximated by:

$$c_{\text{min}}^2 = \frac{\partial p}{\partial \rho} = \frac{p - p_{\text{ref}}}{\rho - \rho_{\text{ref}}}$$ (75)

A piecewise continuous function can then be created to add the equations for the two pure phases. This approach turned out to be problematic at the discontinuities where the different curves meet. Instead, the barotropic state law was approximated by one continuous curve that describes the density over the entire pressure interval. This curve was defined by the following function, known as the Sigmoid function:

$$y(x) = \frac{1}{1 + e^{-x}}$$ (76)
Figure 14: Sigmoid function adjusted to describe the barotropic state law.

This function can be tuned with constants in order to match the shape of the barotropic state law, see Figure 14. Also included in this figure is a curve with the slope $\frac{1}{c_{\text{min}}^2}$, representing the minimum speed of sound.
4 Simulation method

For the numerical simulations done in this work the software Fluent 6.3 and Fluent 12 has been used, [7].

4.1 Numerical settings

Fluid properties:
The fluid used in the calculations is water liquid and water vapor with density $\rho_l = 998.2$ kg/m$^3$ and $\rho_v = 0.554$ kg/m$^3$.

Numerical scheme:
The SIMPLE algorithm was used for the pressure-velocity coupling. The flow equations were solved using a first order scheme in time and space.

Turbulence model:
The $k - \varepsilon$ RNG model was used for all simulations. For the transient simulations the turbulent viscosity was modified according to equations (38) and (39).

Multiphase model:
As multiphase model the mixture model was used. For the simulations with the mixed flow pump the slip velocity between the two phases was neglected. This was necessary in order to get a stable solution.

4.2 Moving zones

When simulating pumps the motion of rotating components needs to be included in the model. In Fluent there are three different ways of doing this; multiple reference frame, mixing plane and sliding mesh. These methods differ both fundamentally and also in their ability to accurate capture flow dynamics occurring in turbomachinery.

Multiple reference frame: The multiple reference frame (MRF) model freezes the motion of the rotating zone in one position, hence its common reference as frozen rotor. In this specific position the fluid flow equations are being solved. During the solution process information that is being transferred across the interface between the zones is modified due to the rotation. The solution is therefore a representation of the flow field at one specific position. If the interaction between the rotor and stator is strong this simplification causes a solution that is
not representative for all positions. The great advantage of this model is that it is computationally cheap and could therefore be used to compute a solution that can be used as an initial solution for a sliding mesh calculation.

**Mixing plane:** For flows where the rotor-stator-interaction is strong the mixing plane model gives a reasonable approximation of the time-averaged flow field. The main difference from the MRF model is the information transfer across the interface. The flow data is averaged in the circumferential direction into the downstream side. This gives rise to "bands" of constant flow properties in the circumferential direction on the mixing plane interface.

**Sliding mesh:** The sliding mesh model is a transient model which accounts for the relative motion of stationary and rotating components. During the calculation the adjacent grids slide relative to each other in discrete time steps. The penalty for using this time-correct model is its calculation time, which can be an order of magnitude larger than for the two methods described above. Another disadvantage is that the zones need to match circumferentially. If the number of blades in the rotating row differ from that in the stationary row, more than one blade passage is required. In worst case all of the blades need to be considered in order to get circumferential periodicity.

These three different models were tested on a simplified geometry of the pump and the velocity distributions are shown in Figure 15. For the sliding mesh method the mean velocity distribution is shown. It is clear that the different models show different results and therefore it is important to consider which model to use for the simulations.

In the mixed flow pump investigated in this work the space between one rotor blade and one stator vane is very narrow and hence, the rotor-stator-interaction is strong. As a model for the rotation of the pump the multiple reference frame is therefore not a good option in this specific case. Instead the mixing plane method is used when calculating the steady state flow field. For transient simulations the sliding mesh method would be the choice, however transient calculations are left for future work.
Figure 15: Velocity distribution in the mixed flow pump calculated with three different models for the modeling of the pump rotation.
5 Test case

As an approximation, the cavitating process of a rotating machine such as a pump can be related to a cascade of hydrofoils. The first step in testing and comparing the different cavitation models is therefore to look at a single hydrofoil.

5.1 Simulation setup

5.1.1 Geometry and grid

The geometry used for this test case is a NACA 0015 hydrofoil at an incidence angle of 8°. The chord length is 0.115 m and the dimensions of the surrounding domain is 1.035 x 0.634 x 0.100 m. A mesh consisting of 43120 cells was generated, see Figure 16.

![Figure 16: Computational grid of a NACA 0015 hydrofoil.](image)

5.1.2 Boundary conditions

As boundary conditions the velocity at the inlet was set to 8 m/s and the pressure was set to be constant at the outlet. The operating pressure could then be changed in order to change the cavitation number. The surface of the foil and the horizontal walls were set to walls with no-slip condition. The boundaries in z-direction were set to symmetry boundaries which reduces the case to two dimensions.

5.1.3 Simulation

First, a steady state solution was achieved in order to compare the time averaged cavity shape and pressure distribution on the hydrofoil. The \( k - \varepsilon \)
RNG turbulence model was used and the cavitation number was equal to 1.2. All of the four cavitation models were applied. However, the barotropic cavitation model was too unstable to give a converged solution.

To be able to observe the transient behaviour of the cavitating flow the unsteady solver was activated and the turbulent viscosity was changed to be defined from the function described in Section 3.4.1. To start unsteady calculations, a stationary solution for an operating pressure with no cavitation occurring was carried out. Then, the unsteady solver was activated and the operating pressure was decreased each time step until the desired cavitation number was reached. A time step, $\Delta t$, of 50 $\mu$s was used, determined according to the CFL condition:

$$\text{CFL} = \frac{|u_i| \Delta t}{|\Delta x_i|} \leq 1 \quad (77)$$

where $|\Delta x_i|$ was taken to be the cubic root of the smallest volume in the grid.

5.2 Results

5.2.1 Steady state solution

![Velocity vectors.](image1)

![Pressure contours.](image2)

Figure 17: Steady state solution.

From the steady state solution the velocity vectors in Figure 17(a) show the flow field around the hydrofoil. The fluid flows from left to right and at the leading edge of the hydrofoil a stagnation point appear where the velocity is zero. The fluid accelerates on the suction side of the hydrofoil (the upper side of the hydrofoil in the figure), which reduces the pressure (Figure 17(b)) and this is where cavitation is expected to be seen. On the aft end of the cavity,
where the pressure has increased above the vapor pressure, a re-entrant jet is expected but is not seen in this steady state solution.

Due to the huge changes in density the barotropic model is very demanding for the solver in Fluent. At the cavity interface the density changes from vapor density in one cell to liquid density in the neighbouring cell. This results in an instability, which was seen when trying to attach the barotropic model to the steady state solver and therefore no results can be presented from this case. The other three models, however, gave converged solutions and the obtained liquid volume fractions are shown in Figure 18.

![Figure 18: Liquid volume fraction obtained from the steady state solution.](image)

From the steady state solution also the pressure distribution was obtained and plotted in the same figure, see Figure 19.
in order to capture the oscillating behaviour of the cavity. The flow field obtained from the transient solution is illustrated with velocity vectors in Figure 20(a). The same behaviour as in the steady state flow field can be seen, however, the re-entrant jet is now visible. This is probably a result of the modified turbulent viscosity used for the transient simulations in order to capture the oscillating behaviour of the cavity.

5.2.2 Transient solution

The flow field obtained from the transient solution is illustrated with velocity vectors in Figure 20(a). The same behaviour as in the steady state flow field can be seen, however, the re-entrant jet is now visible. This is probably a result of the modified turbulent viscosity used for the transient simulations in order to capture the oscillating behaviour of the cavity.
For the transient solution results from all four models were obtained. As the volume fraction is not used in the barotropic model it was decided to compare the models from contours of the density, see Figure 21 and 22. For all models the transient behaviour of the cavity can be followed in four pictures (from top to bottom):

1. The sheet cavity starts to grow and a cloud cavity from the previous cycle is convected in the flow direction.

2. The sheet cavity grows and reaches a maximum size.

3. The re-entrant jet breaks the cavity interface and a new cloud cavity starts to grow.

4. The size of the cloud cavity increases and the sheet cavity moves towards the leading edge of the hydrofoil.
Figure 21: Unsteady density distribution calculated with Singhal’s model (left) and Kunz’ model (right).
Figure 22: Unsteady density distribution calculated with Sauer’s model (left) and barotropic model (right).
5.3 Conclusions

The contour plots of vapor volume fraction from the steady state solution show that Kunz’ and Singhal’s cavitation models show the same cavity shape, while the solution from Sauer’s cavitation model differ slightly. Also, when looking at the pressure distribution on the hydrofoil a slight difference between the models can be seen. This difference appear at the aft end of the cavity, and can therefore be a cause of the re-entrant jet. As this re-entrant jet could not be seen from the velocity vectors a transient simulation with a different turbulence model was done.

The main goal with the transient solution was to capture the transient behaviour of the cavity shape when using the $k - \varepsilon$ RNG turbulence model with modified turbulence viscosity. The contour plots of the density show that this cyclic behaviour was captured with all four models. However, the shape of the cavity does not look the same for all models. The three cavitation models based on the transport equation for mass fraction all show similar behaviour. The largest difference can be seen in the density calculated with the barotropic cavitation model. Here the cavity is less diffuse, which is probably a consequence of the fact that this model is local and does not model the transport of the vapor.

Due to the difficulties of achieving a stable solution of the barotropic cavitation model, it was decided not to use this model in the pump simulations. Concerning the other three models, the main difference was seen in the steady state pressure contours where Sauer’s model showed some deviation from the other two models. Between Kunz’ and Singhal’s model there were some differences in the transient behaviour but it is hard to find reasons for which model to choose. So, the most interesting investigation would be to compare Sauer’s model with either of Singhal’s or Kunz’ model. In Fluent version 6.3., which has been used so far for all simulations, Singhal’s model is the only cavitation model available. However, in the next version of Fluent, Fluent 12, two new cavitation models are available, among which Sauer’s cavitation model is one. Therefore, it is decided to use Singhal’s model in Fluent 6.3. and Sauer’s model in the beta version of Fluent 12. Kunz’ cavitation model is left for future investigation.
6 Cavitating flow in a mixed flow pump

The pump considered here is a mixed flow pump with 6 rotor blades and 11 stator vanes. Numerical data from simulations with two different cavitation models (Singhal’s model and Sauer’s model) will be presented next and compared to data received from the experiments performed at Rolls Royce Hydrodynamic Research Center.

![Pump geometry](image)

Figure 23: Pump geometry, [10].

6.1 Experimental setup

In the water tunnel at Rolls Royce the water enters the rotor via a suction bell and an inlet channel before it reaches the rotor blades. After the water has passed the stator vanes an outlet channel leads it out of the system (Figure 24). To measure the effect cavitation has on the pump operation it is decided to use efficiency, which is calculated as a function of cavitation number. To do this the volume flow rate is kept constant through the pump and the tunnel pressure is varied in order to reach a desired cavitation number. First, the efficiency is calculated at a tunnel pressure where no cavitation is seen. The pressure is then lowered successively and the curve of efficiency as a function of cavitation number can be created. The efficiency of a pump is calculated as:

\[
\eta = \frac{Q \Delta p}{M \omega}
\]  

(78)
where $Q$ is the volume flow rate, $\Delta p$ is the pressure difference over the pump and $M$ is the torque needed to keep the pump rotating at the angular velocity $\omega$.

![Experimental setup](image)

**Figure 24: Experimental setup.**

### 6.2 Simulation setup

#### 6.2.1 Geometry and grid

In order to compare the results from the experiment with the ones achieved from numerical simulation, the geometry of the pump inlet and outlet was shaped as in the experiment. A block structured volume mesh consisting of approximately 225000 cells was generated on a passage with one rotor blade and one stator vane, see Figure 25. In the tip gap between the rotor blade and the shroud three cells were used.

#### 6.2.2 Boundary conditions

To keep the volume flow constant throughout the pump domain the velocity at the outlet was set to be constant. At the inlet the pressure was controlled. The hub wall in the outlet channel is a numerical simplification and is not present in the experiment. This wall was therefore set to be a slip wall. On the rest of the solid surfaces appropriate stationary or rotating wall conditions were imposed.
6.2.3 Simulation

The turbulence model used was $k - \varepsilon$ RNG and the multiphase model was the mixture model without slip velocity. As cavitation model, first Singhal’s model in Fluent 6.3 was used. As a second model, Sauer’s cavitation model in the beta version of Fluent 12 was activated and a comparison was made between the two models.

As in the experiment the operating pressure was first set to a value where no cavitation occur. It was then decreased with 5000 Pa at a time and the efficiency and cavitation number were calculated from the converged solutions. A solution was here considered converged when all residuals had reduced to less than $10^{-3}$. 

Figure 25: Computational grid of the pump.

[Image of computational grid of the pump]
6.3 Results

The flow field in the pump is visualized in Figure 26. The pressure distribution in Figure 26(c) indicates that cavitation will occur at the low pressure region on the suction side of the rotor blade.

![Velocity field around a rotor blade.](image1)

![Velocity field around a stator vane.](image2)

![Pressure contours.](image3)

Figure 26: Steady state flow field.
From the experiment the efficiency as a function of cavitation number has been obtained. This curve can be plotted also for the simulation results with each cavitation model. Figure 27 shows this $\sigma - \eta$ curve with normalized values. When determining the limitations of the pump, the cavitation number where the efficiency has decreased with 1%, $\sigma_1$, is important to predict.

![Figure 27: Efficiency as a function of cavitation number.](image)

During the experiment at Rolls Royce Hydrodynamics Research Center photos were taken on the cavity on the rotor blades. Figure 28 shows three pictures where the different cavitation numbers have been non-dimensionalized with respect to $\sigma_1$. Looking at these photos a distinct boundary between the cavity and the surrounding flow can be seen. When illustrating the flow field contours from the simulations it is hard to define such a sharp limit between vapor and liquid. Therefore, different properties must be observed and compared to photos from the experiment.
Figure 28: Sheet cavity on the rotor blades.

Figure 29 show different iso-surfaces of vapor volume fraction obtained from the simulation with Sauer’s cavitation model and cavitation number $\sigma/\sigma_1 = 1.0$. The iso-surfaces are set to a vapor volume fraction at 0.7, 0.5, 0.3 and 0.1 respectively. Comparing these figures with the photo from the experiment at the same cavitation number the volume fraction of 0.1 seems to be the most correct cavity boundary. However, the shape and placement of the cavity is not captured correctly with this property as a boundary. On the photo from the experiment the cavity inception takes place at the rotor leading edge. The iso-surface of vapor volume fraction at 0.1 predicts the cavitation inception to occur farther downstream.
If instead the main cause to cavitation, pressure, is observed the iso-surfaces in Figure 30 are obtained. The cavitation number is still $\sigma/\sigma_1 = 1.0$ and the solution is obtained with Sauer’s cavitation model. The iso-surface boundaries are set to the default vapor pressure used in the Sauer’s cavitation model, 3540 Pa, and two pressure levels above this, 4000 Pa and 5000 Pa.

The size and shape of the cavity can now be recognized, and the placement has been moved towards the leading edge of the rotor blade. Both the limit of 4000 Pa and 5000 Pa seem to correspond good with experimental results and these iso-surfaces are observed from cases with other cavitation numbers (Figure 31).

The main difference between these two iso-surfaces is that the limit of 5000 Pa covers more of the blade at the leading edge and toward the tip gap. This correspond better to the photos from the experiment, and hence the iso-surface of 5000 Pa is here used to define the cavity boundary. In Figure 32 and 33 the obtained cavity shape from experiments and with Sauer’s and Singhal’s cavitation models are shown. The cavity interface is defined with an iso-surface of pressure at 5000 Pa. It is clearly seen that Sauer’s cavitation model shows a much better agreement with experiment than Singhal’s model.
Figure 30: Pressure contours obtained from simulation with Sauer's cavitation model and cavitation number $\sigma/\sigma_1 = 1.0$.

(a) $p = 3540$ Pa.  
(b) $p = 4000$ Pa.  
(c) $p = 5000$ Pa.

Figure 31: Pressure contours obtained from simulation with Sauer’s cavitation model.

(a) $\sigma/\sigma_1 = 1.2$, $p = 4000$ Pa  
(b) $\sigma/\sigma_1 = 1.2$, $p = 5000$ Pa  
(c) $\sigma/\sigma_1 = 1.5$, $p = 4000$ Pa  
(d) $\sigma/\sigma_1 = 1.5$, $p = 5000$ Pa
6 Cavitating flow in a mixed flow pump

Figure 32: Cavitation number $\sigma/\sigma_1 = 1.5$.

Figure 33: Cavitation number $\sigma/\sigma_1 = 1.2$. 
6.4 Conclusions

The $\sigma - \eta$ curve received from the experiment shows that the efficiency is almost constant until a specific cavitation number is reached and the efficiency drops drastically. One of the causes of this efficiency drop can be found when looking at the torque. As the cavity grows on the rotor blade the load gets heavier and the torque increases. The efficiency is inverse proportional to the torque, and therefore it decreases. Another cause is that the large cavity blocks a part of the water way and this makes the velocity in the pump increase. The pressure increase in the pump is therefore reduced and so is the pump efficiency. This behaviour of the efficiency is also seen from simulations with both cavitation models. However, the efficiency calculated with Singhal’s model decreases continuously, whereas the efficiency calculated from the solution with Sauer’s model stays almost constant until it drops drastically. The value of $\sigma_1$ (cavitation number where the efficiency is reduced by 1%) almost coincide for the experiment and the model by Sauer. In the curve from the experiment a small dip can be seen right before the drop. This dip can be a cause of a phenomenon in the flow, but the deviation lies within the accuracy of the measure equipment and no attempt to explain this behavior is made.

The shape and placement of the cavity has also been discussed and a reasonable cavity boundary can be set by an iso-surface with the pressure of 5000 Pa. The reason why the vapor volume fraction was not chosen was that the cavity shape was a bit displaced towards the trailing edge of the rotor blade. This can be a cause of the constant $n_0$ defined in Sauer’s cavitation model as the constant number of vapor bubbles contained in the flow. This constant is found in the source term for both cavitation and condensation and it therefore decides the rates of which these processes occur. If $n_0$ is raised this would increase the rate of cavitation and condensation and the vapor volume fraction would probably look more like the pressure iso-surfaces.
7 Future work

The results obtained from simulations compared to experiments show that Sauer's cavitation model is a promising model for the calculation of cavitating flow in a mixed flow pump. This hopefully gives inspiration to further work within the area and encourage more simulations in a wider range of applications.

Regarding the specific case investigated here the following can be of interest for future work:

- To follow the transient behaviour of the cavity a sliding mesh simulation could be interesting.

- In the simulations made in this work the sheet cavity was captured, but to resolve vortices and effects from the tip gap a finer grid is needed.

- The flow needs to be investigated under other conditions. Here, one volume flow rate was simulated, and hence, the dependence of $\eta$ and $\sigma_1$ on the volume flow rate was never predicted. These are both decisive when estimating the quality and operating point of a pump and it could therefore be of importance to investigate this in further simulations.

- To examine the effects $n_0$ in Sauer's model has on the solution tuning of this constant can also be set on the list for future work. As discussed previously a first attempt could be to raise this constant in order to speed up the cavitation and condensation process.
References


A Rayleigh-Plesset equation

The bubble considered when deriving the Rayleigh-Plesset equation for bubble dynamics can be seen in Figure 34.

![Figure 34: Single spherical bubble.](image)

The radial distance from the center of the bubble to an arbitrary position is denoted as \( r \) and the radial outward velocity is denoted \( u(r, t) \). Further assumptions is incompressible flow (\( \rho_l \) constant) with constant viscosity, \( \mu_l \), and a uniform temperature, \( T_B(t) \), and pressure, \( p_B(t) \), inside the bubble.

The first step of the derivation of bubble dynamics is to consider the mass conservation for incompressible flow expressed in spherical coordinates:

\[
\frac{1}{r^2} \frac{\partial (r^2u)}{\partial r} = 0 \Rightarrow r^2u(r, t) = C(t)
\]  

(79)

where \( C \) is a constant dependent of time \( t \). As a first boundary condition no mass transport across the bubble interface is allowed to occur, i.e. \( u(R, t) = \frac{dR}{dt} \). This gives:

\[
u(R, t) = \frac{C(t)}{R^2} = \frac{dR}{dt}
\]

(80)

\[
\Rightarrow C(t) = R^2 \frac{dR}{dt}
\]

(81)

\[
\Rightarrow u(r, t) = \frac{R^2}{r^2} \frac{dR}{dt}
\]

(82)
A Rayleigh-Plesset equation

In the case of evaporation and condensation, mass transfer across the boundary must be considered. The rate of increase of size of the bubble can be expressed as $4\pi R^2 \frac{dR}{dt}$, and in the case of evaporation the volume rate of production of vapor must equal this value. Hence, the mass rate of evaporation is $\rho_v 4\pi R^2 \frac{dR}{dt}$, and for this an equal mass of liquid is needed. Equalizing the mass rate of evaporation and the mass flow of liquid inward to the bubble interface, $\rho_l u l 4\pi R^2$, the liquid velocity can be expressed as:

$$u_l = \frac{\rho_v}{\rho_l} \frac{dR}{dt}$$  \hspace{1cm} (83)

The new boundary condition at the bubble interface can now be formulated as:

$$u(R, t) = \frac{dR}{dt} - \frac{\rho_v}{\rho_l} \frac{dR}{dt} = \left[ 1 - \frac{\rho_v}{\rho_l} \right] \frac{dR}{dt}$$  \hspace{1cm} (84)

which gives:

$$u(R, t) = \frac{C(t)}{R^2} = \left[ 1 - \frac{\rho_v}{\rho_l} \right] \frac{dR}{dt}$$  \hspace{1cm} (85)

$$\Rightarrow C(t) = \left[ 1 - \frac{\rho_v}{\rho_l} \right] R^2 \frac{dR}{dt}$$  \hspace{1cm} (86)

As $\rho_l \gg \rho_v$ for most cases this term can be ignored and the expression for the constant $C(t)$ can be simplified to the constant received when not counting for mass transfer, see equation (81).

The second step in this derivation is to consider the Navier-Stokes equation for motion in radial direction which can be expressed as:

$$\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial r} = -\frac{1}{\rho_l} \frac{\partial p}{\partial r} + \nu_l \left[ \frac{1}{r^2} \frac{\partial}{\partial r} \left( r^2 \frac{\partial u}{\partial r} \right) - \frac{2u}{r^2} \right]$$  \hspace{1cm} (88)

Body forces are here neglected. The expression for the velocity $u$ achieved from equation (79) can be inserted in (88) giving:

$$\frac{1}{r^2} \frac{\partial C}{\partial t} - 2 \frac{C^2}{r^5} = -\frac{1}{\rho_l} \frac{\partial p}{\partial r} + \nu_l \left[ \frac{1}{r^2} \frac{\partial}{\partial r} \left( -\frac{2C}{r} \right) - \frac{2C}{r^4} \right]$$  \hspace{1cm} (89)

Carrying out the derivation of the second term on the right hand side of the expression, the two terms within the brackets can be cancelled and the viscous term vanishes leaving:
A Rayleigh-Plesset equation

\[
\frac{1}{r^2} \frac{\partial C}{\partial t} - \frac{2}{r^5} C^2 = -\frac{1}{\rho_l} \frac{\partial p}{\partial r}
\]  
(90)

This expression can be integrated in the radial direction from \( r \) to \( \infty \) giving:

\[
\frac{p - p_\infty}{\rho_l} = \frac{1}{r} \frac{\partial C}{\partial t} - \frac{C^2}{2r^4}
\]  
(91)

With the expression for \( C(t) \) from equation (81) inserted and evaluation at \( r = R \) gives the Rayleigh-Plesset equation in terms of the liquid pressure at the bubble interface:

\[
\frac{p_{r=R} - p_\infty}{\rho_l} = \frac{3}{2} \left( \frac{dR}{dt} \right)^2 + R \frac{d^2R}{dt^2}
\]  
(92)

The third and last step to obtain the complete equation for bubble dynamics is therefore to find an expression for \( p_{r=R} \). To do this in a way that accounts for all mechanisms present is very complicated, and most derivations rely on assumptions. Here, the assumption used above with no mass transfer is used and the total force acting on a thin lamina of the bubble can therefore be set equal to zero, see Figure 35.

![Figure 35: Thin lamina of the bubble surface.](image)

The forces acting on the lamina can be balanced to:

\[
\sum F_r = (\sigma_{rr})_{r=R} + p_B - \frac{2S}{R} = 0
\]  
(93)

where the last term is a function of the surface tension \( S \) and the bubble radius. The stress tensor \( (\sigma_{rr})_{r=R} \) for an incompressible Newtonian fluid is expressed as \(-p_{r=R} + 2\mu_l (\partial u / \partial r)_{r=R} \) giving:

\[
-p_{r=R} + 2\mu_l \left. \frac{\partial u}{\partial r} \right|_{r=R} + p_B - \frac{2S}{R} = 0
\]  
(94)
Inserting $u$ from equation (82) leads to an expression for the liquid pressure at the bubble interface:

$$p_{r=R} = p_B - \frac{4 \mu_l}{R} \frac{dR}{dt} - \frac{2S}{R}$$  \hspace{1cm} (95)

This expression can now be inserted in equation (92) to obtain the Rayleigh-Plesset equation for bubble dynamics as:

$$\frac{p_B - p_\infty}{\rho_l} = R \frac{d^2 R}{dt^2} + \frac{3}{2} \left( \frac{dR}{dt} \right)^2 + \frac{4 \nu_l}{R} \frac{dR}{dt} + \frac{2S}{\rho_l R}$$  \hspace{1cm} (96)
B  UDF - Kunz’ cavitation model

#include "udf.h"

#define M 0.029
#define R 8.314472
#define t_inf 1
#define U_inf 8
#define c_prod 100
#define c_dest 100

DEFINE_CAVITATION_RATE(kunz, c, t, p, rhoV, rhoL, mafV, p_v, cigma, f_gas, m_dot) {
real p_vapor = *p_v;
real mafG = *f_gas;
real mafL, rhoG, rho, alphaL, alphaG, alphaV, dp;

mafL = 1.-mafV[c]-mafG;
rhoG = M*ABS_P(p[c], op_pres)/(R*C_T(c,t));
rho = 1./(mafV[c]/rhoV[c]+mafG/rhoG+mafL/rhoL[c]);

alphaL = mafL*rho/rhoL[c];
alphaG = mafG*rho/rhoG;
alphaV = 1.-alphaL-alphaG;
dp = p_vapor - ABS_P(p[c], op_pres);

if(dp > 0.0) /* evaporation */
  *m_dot = c_prod*rhoV[c]*alphaL*MIN(0,-dp)/
            (0.5*rhoL[c]*pow(U_inf,2)*t_inf);
else /* condensation */
  *m_dot = c_dest*rhoV[c]*pow((alphaL-alphaG),2)*
            alphaV/t_inf;
}

56
# UDF - Sauer’s cavitation model

```c
#include "udf.h"

#define min_maf 1.e-5
#define max_maf 0.99999
#define n0 1.e8

DEFINE_CAVITATION_RATE(sauer, c, t, p, rhoV, rhoL, mafV, p_v, cigma, f_gas, m_dot)
{
    real p_vapor = *p_v;
    real mafM, rho, vofV, dp, R;

    mafM = MIN(MAX(min_maf, mafV[c]), max_maf);
    rho = mafV[c]*rhoV[c]+(1.-mafV[c])*rhoL[c];
    vofV = mafM*rho/rhoV[c];

    dp = p_vapor - ABS_P(p[c], op_pres);
    R = pow(3.*vofV/((1.-vofV)*n0*4*M_PI),1./3.);

    if(dp > 0.0) /* evaporation */
        *m_dot = rhoV[c]*rhoL[c]/rho*3*vofV/R*(1-vofV)*sqrt(2./3*dp/rhoL[c]);
    else /* condensation */
        *m_dot = -rhoV[c]*rhoL[c]/rho*3*vofV/R*(1-vofV)*sqrt(2./3*(-dp)/rhoL[c]);
}
```

57
D  UDF - Barotropic cavitation model

#include "udf.h"

#define rhoV 0.5542
#define rhoL 998.2
#define pv 2300.
#define dp 3000.
#define a 1.2
#define t1 -7.
#define t2 7.

DEFINEPROPERTY(superfluid_density, c, t)
{
    real rho, drho;
    real k, dt;
    real p, p1, p2;
    real p_operating;

    drho = rhoL-rhoV;
    k = 1./(pow(a,2.));
    dt = t2-t1;
    p1 = pv-dp/2.;
    p2 = pv+dp/2.;

    p_operating = RP_Get_Real ("operating-pressure");
    p = C_P(c,t) + p_operating;

    /* Sigmoid function */
    rho = rhoV+drho*1./(1.+exp(-k*((p-p1)*dt/dp + t1)));

    return rho;
}

DEFINEPROPERTY(sound_speed, c,t)
{
    return a;
}
E  UDF - Modified turbulent viscosity

#include "udf.h"

#define Cmu 0.0845
#define rho_V 0.5542
#define rho_L 998.2

DEFINE_TURBULENT_VISCOSITY(mut_rev, c, t)
{
    real mu_t;
    real f_rho;
    real gamma;

    gamma = (C_R(c,t)-rho_V)/(rho_L-rho_V);
    f_rho = rho_V+pow(gamma,10)*(rho_L-rho_V);
    mu_t = f_rho*Cmu*SQR(C_K(c,t))/C_D(c,t);

    return mu_t;
}