Abstract

Computational Fluid Dynamics (CFD) is a very important tool for the study of complex fluid flows and the design of hydraulic fluid flow machinery. At the same time, experimental analysis is very difficult to perform. Thus, for a better understanding of the behaviour of such complex flows, including turbulence, unsteadiness and cavitation, a suitable knowledge of CFD is indispensable. Generally, the specific applications of CFD codes for solving this type of engineering problems are not well documented and a previous work for the acquirement of the CFD code capabilities is necessary.

This work presents numerical investigation concerning complex unsteady flows, including turbulence and cavitation. The main objective was to acquire deeper knowledge about the software potentials for solving this kind of flows. To reach this objective several cases have been studied with a commercial CFD code (Fluent v6.1). The turbulence models being used were mainly the Spalart-Allmaras, the Standard k-ε, and the k-ω model [1], [2], [3], [4]. The utilized cavitation model was from Singhal, 2002 [5]. Based on long term considerations, this investigation aims at the application of the acquired knowledge and experience for further investigations relative to the cavitation phenomena in real fluid flow machines.

Several steps were necessary to understand the suitable simulation process of unsteady turbulent cavitating flows. The case of an unsteady and turbulent, non-cavitating flow around a 2D circular cylinder was studied as a first step using different turbulence models at Reynolds numbers around the critical drag-crisis region. Compared with experimental data, the results are quite divergent, but similar numerical researches (J.S.Cox et al., 1997, [6], M.M.Zdravkovich, 1997, [7]) revealed comparable conclusions as does the present work. Mainly 3D effects are the cause of the non accuracy of the findings (e.g. P.D.Ditlevsen, 1996) [8].

As a second step, the cavitation phenomena has been studied in several applications. First, the full cavitation model implemented in Fluent (Singhal, 2002) has been tested comparing findings with corresponding experimental data. The first case was a steady, cavitating flow through a sharp-edged orifice by Nurick, 1976 [9]. Further, the unsteady turbulent flow around a 2D NACA 0015 hydrofoil has been simulated using the cavitation model. This work was based on the publications by Kubota, 1992 [10] and Berntsen et al., 2001 [11]. Results revealed that depending on the case, the cavitation model offers useful results, but only in a qualitative way. Accurate fittings with experiments are obtained only in few cases. The theoretical validity of the present cavitation model could be questioned. Future work consists of the prediction of damage caused by cavitation (comparing numerical results with experimental databases e.g. Escaler, 2001) using adequate software tools. The final goal is to apply the knowledge obtained to damage prediction in turbomachinery.
Contents

Abstract ........................................................................................................................................ 1

Contents ......................................................................................................................................... 3

1. Nomenclature ................................................................................................................................. 5

2. Introduction ..................................................................................................................................... 9

   2.1. Analysis of complex flow fields .............................................................................................. 9
   2.2. Main objectives of the present work ....................................................................................... 10

3. Physical Background ...................................................................................................................... 13

   3.1. RANS equations and turbulence modelling ............................................................................ 13
        3.1.1. The Spalart-Allmaras model ........................................................................................... 18
        3.1.2. The k-ε model .................................................................................................................. 19
        3.1.3. The k-ω model .................................................................................................................. 23
   3.2. Cavitating flows ....................................................................................................................... 25
   3.3. The Cavitation Model ............................................................................................................... 29

4. Numerical results ........................................................................................................................... 33

   4.1. Circular Cylinder ...................................................................................................................... 33
        4.1.1. Preliminary discussion and numerical implementation ..................................................... 37
        4.1.2. Mesh sensitivity control and numerical results ................................................................. 40
   4.2. Sharp-edged orifice ................................................................................................................. 46
        4.2.1. Experimental set-up ......................................................................................................... 46
4.2.2. Numerical implementation and boundary conditions....................... 48
4.2.3. Mesh refinement analysis............................................................... 50
4.2.4. Simulation results........................................................................... 51

4.3. Hydrofoil............................................................................................................... 58
4.3.1. Simulation based on the case of Kubota (1992)............................... 60
  4.3.1.1. Computational domain and boundary conditions.................... 61
  4.3.1.2. Simulation results and discussion........................................... 62
4.3.2. Simulation based on the case of Berntsen et al. (2001)................... 67
  4.3.2.1. Computational domain and boundary conditions.................... 68
  4.3.2.2. Simulation results and discussion........................................... 68

Conclusions ............................................................................................................. 71
Acknowledgements .................................................................................................... 73
Annexes...................................................................................................................... 75
Bibliography .............................................................................................................. 89
1. Nomenclature

Symbols

\( A_c \) effective flow area

\( A_l \) geometric area of the orifice

\( b \) cylinder length

\( c_d \) discharge coefficient

\( c_D \) drag force coefficient

\( c_L \) lift force coefficient

\( c_p \) pressure coefficient

\( c_c \) empirical constant

\( C_{c, e} \) constants in vapour generation

\( C_{b_2}, C_{1x}, C_{2x}, C_{v1} \) constants

\( d \) diameter

\( d_b \) bubble diameter

\( D \) characteristic length/diameter

\( f \) frequency

\( f, f_{br}, f_{ci}, f_g \) mass fraction

\( f_{\beta}, f_{v1} \) viscous damping function

\( F_D \) drag force

\( F_L \) lift force

\( G_v \) production of turbulent viscosity

\( G_k, G_\omega \) production of \( k \) and \( \omega \)

\( k \) turbulent kinetic energy

\( L \) orifice length

\( L_m \) Prandtl mixing length

\( L_t \) characteristic scale length

\( Ma \) Mach number

\( m_b, m_c \) mass

\( m \) mass flow

\( p \) pressure

\( P \) pressure forces

\( P^* \) pressure and volume forces

\( p_{\text{ref}} \) reference pressure

\( p_{\text{sat}} \) saturation pressure

\( p_{\text{turb}} \) local pressure fluctuations

\( p_{\nu}, p_{\text{vap}} \) vaporization pressure

\( R_{cv}, R_e \) source terms for bubble generation

\( Re \) Reynolds number

\( R_k, R_\omega \) constants

\( S_{o, St} \) Strouhal number

\( S_{to} \) Stokes number

\( S \) modulus of the mean rate-of-strain tensor

\( S_k, S_\omega \) source terms
\begin{itemize}
\item \( t \) \quad \text{time} \\
\item \( u_k \) \quad \text{stress velocity} \\
\item \( u_i, u_j, u_k \) \quad \text{component velocity, mean value} \\
\item \( u_i' \) \quad \text{velocity fluctuation quantity} \\
\item \( u_\infty \) \quad \text{free stream velocity} \\
\item \( u^+ \) \quad \text{normalized wall velocity} \\
\item \( U, U_i \) \quad \text{velocity mean value} \\
\item \( U_t \) \quad \text{characteristic mean velocity} \\
\item \( \text{We} \) \quad \text{Weber number} \\
\item \( x_j, x_j \) \quad \text{coordinate (Cartesian)} \\
\item \( C \) \quad \text{chord length} \\
\item \( y \) \quad \text{wall distance} \\
\item \( y^+ \) \quad \text{normalized wall distance} \\
\item \( Y_v \) \quad \text{destruction of turbulent viscosity} \\
\item \( Y_k, Y_\omega \) \quad \text{destruction of } k \text{ and } \omega \\
\item \( \alpha \) \quad \text{angle of attack} \\
\item \( \alpha_0, \alpha_c, \alpha_g \) \quad \text{volume fraction} \\
\item \( \alpha, \alpha^*, \alpha_0, \alpha^*_0, \alpha_v, \alpha_v^* \) \quad \text{constants} \\
\item \( \beta, \beta^* \) \quad \text{constants} \\
\item \( \Gamma_k, \Gamma_\omega \) \quad \text{effective diffusivity of } k \text{ and } \omega \\
\item \( \delta \) \quad \text{boundary layer} \\
\item \( \varepsilon \) \quad \text{dissipation rate of } k \\
\item \( \eta, \eta_b, \eta_c \) \quad \text{dynamic viscosity} \\
\item \( \kappa \) \quad \text{von Karman constant} \\
\item \( \lambda \) \quad \text{constant (Stokes hyp.)} \\
\item \( \nu \) \quad \text{kinematic viscosity} \\
\item \( \nu_s \) \quad \text{turbulent kinematic viscosity} \\
\item \( \tilde{\nu}_s \) \quad \text{modified turbulent viscosity} \\
\item \( \rho, \rho_b, \rho_c \) \quad \text{density} \\
\item \( \sigma \) \quad \text{cavitation number} \\
\item \( \sigma/2\alpha \) \quad \text{composite parameter} \\
\item \( \sigma_s \) \quad \text{surface tension} \\
\item \( \sigma_k, \sigma_v, \sigma_\omega \) \quad \text{constants} \\
\item \( \tau \) \quad \text{integration time variable} \\
\item \( \tau_{\text{dyn}} \) \quad \text{response time} \\
\item \( \tau_{\text{str}} \) \quad \text{characteristic stream term} \\
\end{itemize}
\( \tau_w \)  
wall stresses

\( \omega \)  
specific dissipation rate

\( \Omega_b \)  
bubble radius

\( \psi \)  
volume forces

**Subscripts and Superscripts**

- \( b \)  
bubble (phase)
- \( c \)  
liquid (phase)
- \( e \)  
element
- \( g \)  
non-condensable gases
- \( i,j,k \)  
tensor notation

**Mathematical tools**

\[ \delta_{ij} \rightarrow \begin{cases} 
0 & \text{for } i \neq j \\
1 & \text{for } i = j 
\end{cases} \quad \text{(Kronecker delta)} \]

\( \nabla, \vec{V} \rightarrow \nabla = \sum_{i=1}^{n} \frac{\partial}{\partial x_i} \quad \text{(Nabla = divergence operator)} \)

\[ \frac{D}{Dt} \rightarrow \left\{ \frac{\partial}{\partial t} + (\vec{V} \vec{u}) = \frac{\partial}{\partial t} + u_i \frac{\partial}{\partial x_i} \right\} \]

---

1 Note that the number format in tables and graphics is the European (i.e. 0,1 is “0 point 1”; 10,000 is “ten thousand”).
Tensor notation: \[ \frac{\partial u_i}{\partial x_j} = \sum_{i=1}^{n} \frac{\partial u_i}{\partial x_j} \] for n-dimensional vectors

Example: \[ u_j \frac{\partial u_i}{\partial x_j} = \sum_{j=1}^{n} u_j \frac{\partial u_i}{\partial x_j} \quad \text{or} \quad \frac{\partial u_i}{\partial x_j} \delta_{ij} = \sum_{i=1}^{n} \sum_{j=1}^{n} \frac{\partial u_i}{\partial x_j} \delta_{ij} \]
2. Introduction

2.1. Analysis of complex flow fields

Cavitation is a very complex vapour-liquid two-phase flow phenomenon including phase changes and viscous effects. Cavitation strongly affects the performance of hydraulic turbomachinery. When the local pressure within a flow with water as medium becomes lower than the water vapour pressure, bubbles begin to form and are further transported to regions of higher pressure where they collapse. This bubble collapse, when it occurs near foil surfaces, is the main source of material damage, erosion and noise in hydraulic nozzles and pumps. In spite of many studies, the structure of cavitation, its different forms as well as its influences on the flow field are poorly understood ([9], [10], see also most of complementary literature treating on the cavitation phenomena). Especially when it proceeds in unsteady flows, its mechanism becomes very complex.

The current tendencies in the design of hydraulic turbomachinery of every type, especially those requiring a high grade of optimisation as well as functional guarantee at critical regions, are strongly influenced by the CFD ( Computational Fluid Dynamics) simulation techniques. In this context there have been developed many methods and software programmes for the elaboration and analysis of fluid mechanical problems.

CFD has been applied for about 25 years and has been spread as a very promising tool for present and future design and development of turbo-engines. The main interest is to establish a solid and reliable shifting system between pure theory and experimental research, providing a tool for calculating flow phenomena and enabling engineering applications. The development of CFD strongly depends on the progress made by the computer science. In spite of its rapid development, experiments still remain of crucial importance. The present work is an example of the close connection and dependence that exist between theory and practice.

Several important works analysing cavitation phenomena have been done in the past (e.g. Kubota, 1992 [10]; Arndt et al., 2000-2002 [12], [13]; Kunz et al. 2000 [14]; Song and Qin, 2002 [15]). In the case of the cavitating flow around a hydrofoil, the work of Kubota shows interesting results of a numerical investigation of the unsteady structure (velocity distribution) of cavitation around a 2D-hydrofoil. These results lead to more detailed information about processes related to cavitation as well as further theories upon cavitation and its classification. Later on, other works and numerical investigations found that the cloud
cavitation observed in experiments using hydrofoils was a large-scale vortex with many small cavitation bubbles. The importance of the interaction between large-scale coherent vortices in the flow field and cavitation bubbles was recognized. A bridge between cavitation, unsteadiness and turbulence was established.

Regarding unsteady turbulent flows, much more extensive works are available. However, theories about turbulence modelling are strongly based upon empiric data and acknowledgement. The first chapters of this work offer several aspects of the current tendencies of CFD and the methods used by Fluent to simulate unsteady, turbulent flows. Further, the cavitation model that has been used for all computations will be discussed. For details upon cavitating flows please refer to the chapters describing the cases treated within the present work or to the listed bibliography.

2.2. Main objectives of the present work

Within this work the main goal was to provide advanced knowledge for two-phase fluid flow simulations. By means of the software program FLUENT for flow simulation, an unsteady, incompressible two-phase fluid-flow has been simulated and analysed. The results will contribute to the main project, which aims to analyse cavitating flows in hydraulic turbomachinery. The detection of cavitation phenomena and the understanding of the influence of the streaming processes on the collapse of the cavitation bubbles and the material damage caused by this phenomenon, would lead to major progress in the development of hydraulic turbines. The results would enable the simulation and the prediction of the position and the intensity of bubble collapse in higher pressure zones, at a given flow set-up. Experimental research has been carried out by Escaler (2001) upon a cavitating flow around a hydrofoil containing a cylindrical obstacle at its leading edge. In order to accelerate the obtaining of data, numerical simulation must replace experimental research.

As a first step, steady and unsteady one-phase flows on simple 2D geometries have been modelled (circular cylinder, sharp-edged orifice) in order to acquire experience before simulating more difficult cases. Changing characteristic flow parameters, like the Reynolds number, made it possible to compare the results obtained to existing experimental data. Concerning the flow past a circular cylinder, many questions about the physical processes that occur still exist [6], [7], [8]. Experimental and previous numerical research data was used for judging the best method (turbulence model) to be applied.
As a further step, a two-phase flow has been analysed for a sharp-edged orifice and a hydrofoil using the cavitation model for multiphase flows. The sharp edged orifice could be simulated with the steady state equations, while using the Nurick correlation to validate the results. In the case of the hydrofoil, the flow was unsteady, turbulent and depending on the settings, composed by two phases.

Before discussing the mentioned cases that are treated within this work, an introduction into state-of-the-art methods for turbulent flow simulations will be described from a physical point of view. The turbulent models provided in FLUENT as well as different theories and approaches for the simulation of unsteady, two-phase fluids will be exposed.
3. Physical Background

3.1. RANS equations and turbulence modelling

Turbulent flows are characterized by fluctuating velocity fields. These velocity fluctuations also lead to fluctuations of momentum or energy. Since these fluctuations can be of small scale and high frequency, they are too computationally expensive to be simulated directly in practical engineering calculations. Instead, the instantaneous (exact) governing equations can be time averaged, ensemble-averaged, or otherwise manipulated to remove the small scales, resulting in a modified set of equations that are computationally less expensive to solve (e.g.: Reynolds averaged). However, the modified equations contain additional unknown variables, and turbulence models are needed to determine these variables in terms of known quantities (closure’s problem).

We generally distinguish several main turbulence models, being the Standard k-ε model the mostly used to use engineering tasks. The following turbulent models provided in FLUENT (2D) have been used:

- Spalart-Allmaras model
- Standard k-ε model
- RNG k-ε model
- Realizable k-ε model
- Standard k-ω model
- SST k-ω model

The basic equations which lead to the Reynolds Averaged Navier-Stokes equations and the introduction of turbulence models are the momentum equations (2nd Newton’s law) and the continuity equation. The continuity equation postulates that the change in mass in the volume is equal to the fluxes through the surfaces of a volume element. In tensor notation and after dividing by the control volume, this leads to:
\[
\begin{align*}
\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} &= -\frac{1}{\rho} \frac{\partial p}{\partial x_i} - \frac{\partial \Psi}{\partial x_i} + \frac{1}{\rho} \frac{\partial \tau_{ji}}{\partial x_j} \\
\frac{\partial \rho}{\partial t} + \nabla (\rho \cdot \vec{u}) &= 0 ; \quad \text{for incompressible fluids:} \quad \nabla (\rho \cdot \vec{u}) = 0
\end{align*}
\]  
(3.1)  
(3.2)

This equation set (3+1) leads to following unknowns:

- 3 velocity components \( u_i \)
- pressure \( p \)
- density \( \rho \)
- stress tensor \( \tau_{ij} = \begin{bmatrix} \tau_{11} & \tau_{12} & \tau_{13} \\ \tau_{21} & \tau_{22} & \tau_{23} \\ \tau_{31} & \tau_{32} & \tau_{33} \end{bmatrix} \)

The number of unknown variables is superior to the given equations, which obliges to assume several simplifications. The main goal is to find an acceptable relation for the stress tensor:

1) The fluid is \textit{newtonian}. In analogy to structure mechanics, the proportionality between stress and the rate of share are proportional.

\[
\varepsilon_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right),
\]

With the strain rate in tensor notation, the stress-strain relationship is:

\[
\tau_{ij} = \lambda \cdot \varepsilon_{kk} \cdot \delta_{ij} + 2 \cdot \eta \cdot \varepsilon_{ij},
\]

where \( \delta_{ij} \) is the Kronecker-Delta.
2) The Stokes hypothesis which assumes, that the pressure is the average normal stress. This results in:

\[
\frac{\lambda}{\eta} = -\frac{2}{3}
\]

Introducing these assumptions into the momentum equations (3.1), and with \( \nu = \eta/\rho \), one obtains the Navier-Stokes equations [3]:

\[
\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} - \frac{\partial \Psi}{\partial x_i} + \nu \left[ \frac{\partial (u_i + u_j)}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right] - \frac{2}{3} \rho \frac{\partial u_i}{\partial x_j} \delta_{ij}
\]

Together with the continuity equation (3.2), 4 equations and 5 unknowns are obtained. By adding a supplementary transport equation for a scalar quantity and the relation between pressure, density and the transported variable, one obtains a closed set of equations. An example would be the use of the transport equation for the temperature together with the ideal gas law (for a gas flow). Another one is the simulation of two-phase flows with the cavitation model, where the volume fraction equation brings the relation between the state variables of the two phases, and closes the Navier-Stokes equation set [chapter 2.4].

The Navier-Stokes equation computes the exact velocity field in a flow. Simulating a turbulent flow using the theory described above leads to a computational technique called the DNS-method (Direct Numerical Simulation). Somehow, the present-day possibilities of such computations quickly reach the actual computer limits, even in basic engineering problems. It is not even of major interest to calculate the exact magnitude of fluctuations within the whole eddies’ scale [3]. CFD-methods use models, the so called turbulence models, introducing further equation sets in order to simplify and reduce the computation problem. The LES-method (Large Eddy Simulation) uses the Navier-Stokes equation set for computing big vortexes within a turbulent flow. In the smaller scales, the turbulence is simulated with the help of turbulence models. For complex flow-fields, this method is not yet applicable in complex industrial problems. Mostly used is the RANS-method (Reynolds Averaged Navier-Stokes) which has been developed for the use of turbulence models into the simulation of turbulent flows, and builds the basis of the major part of computational software for fluid mechanics at the same time. It largely leads to best results and approaches for complex engineering tasks. Somehow, its limits can also be quickly reached as it has been experienced within this work.

For simplicity, the present work is restricted to incompressible flows with constant temperature: the compressibility of the flow is treated separately within the cavitation model. How the cavitation model treats the fluid’s compressibility will be treated in later chapters.
every case for which the cavitation model was not enabled, the flow was set to be incompressible. Therefore, the continuity equation is reduced to:

\[ \nabla \cdot \vec{u} = 0 \]  

(3.5)

The momentum equations are then:

\[
\begin{align*}
\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} &= \frac{1}{\rho} \cdot \frac{\partial p}{\partial x_i} - \frac{\partial \Psi}{\partial x_i} + \nu \left[ \frac{\partial (u_i + \partial u_j)}{\partial x_j} \right]
\end{align*}
\]

(3.6)

As said before, turbulence is characterized by fluctuations within the velocity-field around the steady/unsteady fluid flow. The instantaneous velocity values \( u_i \) are split into a mean value \( U_i \) and a fluctuation quantity \( u'_i \):

\[ u_i = U_i + u'_i \]  

(3.7)

The mean values are defined by \( U_i = \frac{1}{\tau} \int_{t_0}^{t_0+\tau} u_i \, dt \), and the fluctuations:

\[ \int_{t_0}^{t_0+\tau} u'_i \, dt = 0 \]

For the time averaging the following conditions must be valid:

\( \tau \gg \) time scale of the turbulent motion

\( \tau < \) time scale of the unsteady men flow.

Averaging the Navier-Stokes equations leads to the Reynolds-averaged equations:

\[ \nabla \cdot \vec{U} = 0 \]  

(3.8)

\[
\begin{align*}
\frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} &= \frac{1}{\rho} \cdot \frac{\partial P^*}{\partial x_i} + \nu \left[ \frac{\partial (U_i + \partial U_j)}{\partial x_j} \right]
\end{align*}
\]

(3.9)

Symmetrical tensor describing the influence of turbulent fluctuations on the mean value.

This system of equations is not closed because the correlations of the velocity fluctuations are unknown. Since the correlations have the form of the stress tensor, they are called Reynolds stresses or turbulent stresses. The turbulent stress tensor \( \rho \cdot u_i u_j \) is symmetric,
and brings 6 unknown components—the turbulent stresses—into the equation set. Similar to the momentum equations, the other transport equations have also to be averaged [3].

Here comes the role of the turbulent models, which serve as a tool for the calculation of the turbulent stresses, and which describe the influence of turbulence on the mean flow (it does not describe the turbulent vortexes in detail!).

The basis of a major part of all turbulence models, is the turbulent viscosity assumption of Boussinesq, which introduces the turbulent viscosity $\nu_t$ to describe the turbulent correlation $u_i u_j$ analogue to the formulation for the viscous stresses $\tau_{ij}$ (3.3):

$$\overline{u_i u_j} = -\nu_t \left( \frac{\partial U_j}{\partial x_j} + \frac{\partial U_i}{\partial x_i} \right) + \frac{1}{3} \overline{u_i u_j} \delta_{ij}$$  (3.10)

where $k = \frac{1}{2} \overline{u_i u_i}$ is the turbulent kinetic energy.

Introducing the law of Boussinesq into the RANS equations, we obtain:

$$\frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho} \cdot \frac{\partial P^*}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \nu - \nu_t \left( \frac{\partial U_j}{\partial x_j} + \frac{\partial U_i}{\partial x_i} \right) \right)$$  (3.11)

where $P^* = P + \frac{2}{3} \cdot \rho \cdot k$

Beyond the turbulence models, there are those that use the turbulent viscosity assumption of Boussinesq and those who do not [3]. The models discussed in this work, all use this principle. Within this category, we can distinguish between turbulence models using one or two equations.

The turbulence models treated in FLUENT are all 2-equation models except the Spalart-Allmaras model which is a 1-equation model and the RSM-model that uses seven equations. Furthermore, the possibility of simulating a laminar flow without using any turbulent model is provided (LAMINAR), as well as an option for simulations of inviscid flows (INVISCID), which can be reasonably used for high Reynolds number flows, where the friction forces are negligible.
3.1.1. The Spalart-Allmaras model

The Spalart-Allmaras model is a relatively simple one-equation model that solves a modelled transport equation for the turbulent viscosity [3]. This embodies a class of one-equation models in which it is not necessary to calculate a length scale related to the local shear layer thickness. This model is conceived especially for boundary layers\(^1\) subjected to adverse pressure gradients. In its original form, the Spalart-Allmaras model is a low-Reynolds-number model, requiring the viscous-affected region of the boundary layer to be properly solved. It could be important to notice, that the near-wall gradients of the transported variable in the model are much smaller than the gradients of the transported variables in the \(k-\varepsilon\) models. Likewise, it will be difficult to rely on the Spalart-Allmaras model to predict the decay of homogeneous, isotropic turbulence. Furthermore, this model can be criticized for its inability to rapidly accommodate changes in length scale, such as might be necessary when the flow changes abruptly from a wall-bounded to a free-shear flow.

The model was developed starting with a transport equation for the turbulent viscosity \(\nu_t\). Advanced versions of the Spalart-Allmaras model deal with the modified form of the kinematic turbulent viscosity, \(\tilde{\nu}_t\). In this case, the transported variable is identical to the turbulent kinematic viscosity except in the near-wall (viscous-affected) region [3]. The transport equation for \(\tilde{\nu}_t\) is:

\[
\frac{\partial \tilde{\nu}_t}{\partial t} + U_i \frac{\partial \tilde{\nu}_t}{\partial x_i} = G_{\nu} + \frac{1}{\sigma_{\nu}} \left[ \frac{\partial}{\partial x_i} \left( \nu + \tilde{\nu}_t \right) \frac{\partial \tilde{\nu}_t}{\partial x_i} \right] + C_{b2} \left( \frac{\partial \tilde{\nu}_t}{\partial x_i} \right)^2 - Y_{\nu} \tag{3.12}
\]

where \(G_{\nu}\) is the production of turbulent viscosity and \(Y_{\nu}\) is the destruction of turbulent viscosity that occurs in the near-wall region due to wall blocking and viscous damping. The terms \(\sigma_{\nu}\) and \(C_{b2}\) are constants. The model-formulations for \(G_{\nu}, Y_{\nu}\) can be found in respective literature (e.g. Coussirat, [3]; Hellster [4]).

\(^1\) For more information about the boundary layer theory, please refer to corresponding literature (e.g. Batchelor)
The turbulent viscosity $\nu_t$ is computed from:

$$\nu_t = \tilde{v}_t \cdot f_{v1}$$

where the viscous damping function $f_{v1}$ is given by $f_{v1} = \frac{\chi^3}{\chi^3 + C_{v1}^3}$ and

$$\chi = \frac{\tilde{v}_t}{v}, \text{ where } C_{v1} \text{ is a constant.}$$

### 3.1.2. The $k$-$\varepsilon$ model

The standard $k$-$\varepsilon$ model is a very well tested model [3,6]. There are several known flow situations, for which this model provides insufficient results. These are for example strongly swirling flows, flows with strong streamline curvature, or flows with strong positive pressure gradients; they are all cases that do not follow the Boussinesq approach. For most of these problems, there are correction terms that can be implemented in the model. The $k$-$\varepsilon$ model is only valid for completely turbulent flows. It cannot predict flows, where viscous effects play an important role, as it is the case for example in near wall regions [3]. The $k$-$\varepsilon$ model describes the transport and dissipation of turbulent kinetic energy. The model bases on the one-equation model of Prandtl & Kolmogorov, where the characteristic flow velocity $U_i$ can be described by the kinetic energy of the fluctuation movement [3]. The characteristic length scale $L_i$ is proportional to Prandtl's mixing length $l_m$:

$$U_i = \sqrt{k}, \text{ where } k = \frac{1}{2} \sum_i U_i^2$$

In the Prandtl-Kolmogorov model, the turbulent viscosity is defined by:

$$\nu_t = \sqrt{kL_i}, \text{ where } L_i = c_d \cdot l_m$$

The aim of the $k$-$\varepsilon$ model is to eliminate the mixing length dependence of the turbulent viscosity. Therefore a $k$-transport-equation is deduced from the Navier-Stokes equation. Instead of using the empiric mixing length for the definition of the characteristic length scale, the dissipation rate $\varepsilon$ of $k$ is introduced as follows:

$$\nu_t = C_\mu \cdot \frac{k^2}{\varepsilon}, \text{ with } L_i = C_\mu \cdot \frac{\sqrt{k^2}}{\varepsilon} \text{ (C_\mu is a constant)}$$
The transport equations for $k$ and $\varepsilon$ are:

**k- transport equation:**

$$\frac{\partial k}{\partial t} + U_i \frac{\partial k}{\partial x_i} = \frac{\partial}{\partial x_i} \left( \frac{\nu_t}{\sigma_k} \frac{\partial k}{\partial x_i} \right) + \nu_t \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \frac{\partial U_j}{\partial x_i} - \varepsilon \tag{3.13}$$

**\varepsilon- transport equation:**

$$\frac{\partial \varepsilon}{\partial t} + U_i \frac{\partial \varepsilon}{\partial x_i} = \frac{\partial}{\partial x_i} \left( \frac{\nu_t}{\sigma_k} \frac{\partial \varepsilon}{\partial x_i} \right) + C_{\varepsilon} \frac{\varepsilon}{k} \nu_t \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \frac{\partial U_j}{\partial x_i} - C_{\varepsilon} \frac{\varepsilon^2}{k} \tag{3.14}$$

All model constants ($C_{\mu}, C_{\varepsilon}, C_{\varepsilon}, \sigma_k$) have been optimised for different types of flows [3].

At this time, several notices upon the boundary layer theory have to be figured out. At the wall ($y = 0$), the wall adhesion condition is valid. In very proximate wall regions, cross movements are tending to zero. A thin zone, called viscous sublayer, or boundary layer $\delta$ (laminar) is created. While viscous effects on the energy containing turbulent motions are negligible throughout most of the flow, the no-slip condition at solid interfaces always ensures that, in the immediate vicinity of a wall, viscous effects are influential [3]. Although the thickness of this viscous-affected zone is usually some orders of magnitude smaller than the overall width of the flow, its effects extend over the whole flow field since, typically, 50% of the velocity change from the wall to the freestream occurs in this region.

Near-wall zones are the main source of vorticity and turbulence because in these regions exist high gradients of variables and the momentum transport is more vigorous. Also, turbulence mixing is suppressed by the proximate boundary, causing a high reduction of the transport across this layer. To take into account the non-isotropic nature of turbulence in near-wall regions, a special wall treatment has to be defined [3]. The k- $\varepsilon$ model does not solve the regions affected by viscous effects. Instead, empirical functions (“wall functions”) are used. Following values are defined:

- the stress velocity
  
  $$u_c = \frac{\varepsilon_w}{\sqrt{\rho}}$$

- the normalized wall velocity
  
  $$u^+ = \frac{U}{U_\tau}$$
• the normalized wall distance

\[ y^+ = \frac{y \cdot U_i}{v} \]

The values for \( u^+ \) and \( y^+ \) are normalized with the respectively highest values for \( u \) and \( y \). The logarithmic wall-function can be written as follows:

\[
u^+ = \begin{cases} 
\frac{1}{\kappa} \ln(1 + \kappa \cdot y^+) + C & \text{for } 10 < y^+ < 150 \\
\kappa \cdot y^+ & \text{for } y^+ < 10 
\end{cases}
\]

(3.15)

Using the k-\( \varepsilon \) models, the logarithmic law is applied when a “standard wall-function” is selected for the approximation of the near-wall region (when \( 10 < y^+ < 150 \) without adverse pressure gradients. Modified wall functions called non-equilibrium wall functions are implemented in the standard model for the computation of boundary layers affected by adverse pressure gradients.

![Diagram of Nomenclature of zero and adverse pressure gradient boundary layers.](image)

**Fig. 3.1.** Nomenclature of zero and adverse pressure gradient boundary layers.

Other near-wall treatment should be used, when \( y^+ < 10 \), (e.g. using in the near wall region a solver based in a simplified Navier-Stokes equation). Note that the Spalart-Allmaras and the k-\( \omega \) models automatically applies the appropriate near-wall treatment.
Apart from the standard k-ε model, FLUENT provides two further variants of the k-ε model: the RNG k-ε model and the realizable k-ε model.

The **RNG** k-ε model was derived using a rigorous statistical technique (called renormalization group theory) [3]. It is similar in form to the standard k-ε model, but includes the following refinements:

1. The RNG model has an additional term in its ε equation that significantly improves the accuracy for rapidly strained flows.
2. The effect of swirl on turbulence is included in the RNG model, enhancing accuracy for swirling flows.
3. The RNG theory provides an analytical formula for turbulent Prandtl numbers, while the standard k-ε model uses user-specified, constant values.
4. While the standard k-ε model is a high Reynolds-number model, the RNG theory provides an analytically-derived differential formula for effective viscosity that accounts for low Reynolds-number effects. Effective use of this feature does, however, depend on an appropriate treatment of the near-wall region [3].

The **realizable** k-ε model is a relatively recent development and differs from the standard k-ε model in two important ways:

1. The realizable k-ε model contains a new formulation for the turbulent viscosity.
2. A new transport equation for the dissipation rate ε has been derived from an exact equation for the transport of the mean-square vorticity fluctuation.

The term “realizable” means that the model satisfies certain mathematical constraints on the Reynolds stresses, consistent with the physics of turbulent flows. Neither the standard k-ε model nor the RNG k-ε models are realizable [3].

As in other k-ε models, the eddy viscosity is computed from:

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

The difference between the realizable k-ε model and the standard and RNG – models is that $C_{\mu}$ is no longer a constant. In Fluent it is computed from a given formula (see source [3]), as a function of the mean strain and rotation rates, the angular velocity of the system rotation, and the turbulence fields k and ε.
The realizable model is likely to provide superior performance for flows involving rotation, boundary layers under strong adverse pressure gradients [4]. For flows with complex secondary flow features, the realizable $k$-$\varepsilon$ model also shows best results.

### 3.1.3. The $k$-$\omega$ model

The $k$-$\omega$ model is a semi-empirical model based on model transport equations for the turbulent kinetic energy ($k$) and the specific dissipation rate ($\omega$), which can also be thought of as the ratio of $k$ and $\varepsilon$.

Two versions of the $k$-$\omega$ model are implemented in Fluent. Both, the standard and the shear stress transport (SST) $k$-$\omega$ model have similar forms, with transport equations for $k$ and $\omega$. The major differences are a modified turbulent viscosity formulation for the SST model and a gradual change at the standard $k$-$\omega$ model in the inner region of the boundary layer to a high-Reynolds-number version of the $k$-$\varepsilon$ model in the outer part of the boundary layer [4]. Only the standard $k$-$\omega$ model will be discussed here.

The turbulence kinetic energy, $k$, and the specific dissipation rate, $\omega$, are obtained from the following transport equations:

\[
\frac{\partial}{\partial t} (\rho \cdot k) + \frac{\partial}{\partial x_j} (\rho \cdot k \cdot u_j) = \frac{\partial}{\partial x_j} \left( \Gamma_k \frac{\partial k}{\partial x_j} \right) + G_k - \nu_k + S_k \tag{3.16}
\]

and

\[
\frac{\partial}{\partial t} (\rho \cdot \omega) + \frac{\partial}{\partial x_j} (\rho \cdot \omega \cdot u_j) = \frac{\partial}{\partial x_j} \left( \Gamma_\omega \frac{\partial \omega}{\partial x_j} \right) + G_\omega - \nu_\omega + S_\omega \tag{3.17}
\]

In these equations, $G_k$ represents the generation of turbulent kinetic energy due to mean velocity gradients. $G_\omega$ represents the generation of $\omega$. $\Gamma_k$ and $\Gamma_\omega$ represent the effective diffusivity of $k$ and $\omega$, respectively. $Y_k$ and $Y_\omega$ represent the dissipation of $k$ and $\omega$ due to turbulence. All of the above terms are calculated as described below. $S_k$ and $S_\omega$ are source terms [4].

The effective diffusivities for the $k$-$\omega$ model are given by:

\[
\Gamma_k = \mu + \frac{\mu_t}{\sigma_k} \quad \text{and} \quad \Gamma_\omega = \mu + \frac{\mu_t}{\sigma_\omega}
\]

where $\sigma_k$ and $\sigma_\omega$ are the turbulent Prandtl numbers for $k$ and $\omega$. The turbulent viscosity, $\mu_t$, is computed by combining $k$ and $\omega$ as follows:
\[ \mu_i = \alpha^* \frac{\rho \cdot k}{\omega} \]

The coefficient \( \alpha^* \) damps the turbulent viscosity causing a low-Reynolds-number correction and is given by:

\[ \alpha^* = \alpha^* \left( \frac{\alpha_0^* + \text{Re}_i / R_k}{1 + \text{Re}_i / \text{Re}_*^0} \right), \quad \text{where } R_k \text{ and } \alpha_0^* \text{ are constants and } \text{Re}_i = \frac{\rho \cdot k}{\mu \cdot \omega} \]

In the high-Reynolds number form of the \( k-\omega \) model, in the outer part of the boundary layer, \( \alpha^* = \alpha^*_e = 1 \).

The production of turbulent kinetic energy (k), is defined as:

\[ G_k = -\rho \frac{\partial u_i}{\partial x_j} \frac{\hat{\partial} u_j}{\hat{\partial} x_i} \]

To evaluate \( G_k \) in a manner consistent with the Boussinesq hypothesis, we can write:

\[ G_k = \mu_i \cdot S^2, \]

where \( S \) is the modulus of the mean rate-of-strain tensor, defined in the same way as for the \( k-\varepsilon \) model.

The production of \( \omega \) is given by

\[ G_\omega = \alpha \frac{\omega}{k} G_k, \]

where

\[ \alpha = \frac{\alpha^*}{\alpha^*} \left( \frac{\alpha_0^* + \text{Re}_i / R_\omega}{1 + \text{Re}_i / \text{Re}_*^0} \right), \quad \text{Re}_\omega \text{ and } \alpha_0^* \text{ are constants and } \text{Re}_i = \frac{\rho \cdot k}{\mu \cdot \omega} \]

Again, in the outer part of the boundary layer, \( \alpha = \alpha_\omega = 1 \).

The dissipation of \( k \) and \( \omega \) can be written respectively as:

\[ Y_k = \rho \beta^* f_p k \omega \quad Y_\omega = \rho \beta^* f_p \omega^2 \]

The parameters used for the calculation of \( Y_k \) and \( Y_\omega \) are mainly depending on the strain rate tensor and will not be discussed in detail here.
The SST k-ω model presents a modified turbulent viscosity definition to account for the transport of the principal turbulent shear stress. Other modifications include the addition of a cross-diffusion term in the ω equation and a blending function to ensure that the model equations behave appropriately in both the near-wall and far-field zones [4].

The wall boundary conditions for the k equation in the k-ω models are treated in the same way as the k equation is treated when enhanced wall treatments are used with the k-ε models. This means that all boundary conditions for wall-function meshes will correspond to the wall function approach, while for the fine meshes, the appropriate low Reynolds-number boundary layer conditions will be applied. For further details on the wall treatment with the standard or the SST k-ω model, especially the way they are treated in Fluent, we suggest to use literature references and Fluent’s User’s Guide.

### 3.2. Cavitating flows

The knowledge of the behaviour of multiphase fluids is of crucial importance in the optimisation process of many engineering devices. Some examples are: the spray mixing or injection processes, cavitation phenomena in hydraulic turbomachinery, flows in pipelines, etc. This research work is focused on the case of a fluid composed of two phases. The phases are defined as separate areas containing different fluids. These areas can be interpenetrating or separated from each other, establishing a surface stress between them. This difference leads to different model theories.

Cavitation is a phenomenon that appears in low pressure zones (e.g., hydraulic machinery) causing significant degradation in the performance, inducing reduced flow rates, lower pressure increases in pumps, load asymmetry and vibrations, and noise. Further consequences like erosion on the material surfaces can lead to a irreparable damage of turbomachinery. It is exactly this phenomenon that will be dealt with, as a major goal of this project, to streaming processes around different geometries.

Liquid fluids can lift up tensile stresses when nuclei for the inception of cavitation are missing. Real liquid fluids always contain nuclei for the inception of cavitation. When the vaporisation pressure $p_v$ is reached, the liquid begins to gasify. A gasification process without heating, only caused by low pressure, is called cavitation. When cavitation bubbles arriving in higher-pressure zones collapse, they induce near the geometry-surface microscopic erosion-processes which lead to surface damages at a bigger scale. The vaporisation pressure of liquid fluids strongly depends on temperature, $p_v = p_v(T)$. 
There are many levels of modelling that may be used in multi-phase computations. In general, one may distinguish between methods that employ an Eulerian framework for both phases and those that employ Eulerian for the gas-phase and Lagrangian for the liquid-phase. In the Eulerian-Eulerian framework, the simplest approach is to employ a single continuity equation for both phases, with the fluid density being described as a continuous function varying between the vapour and liquid phases [14]. At a more detailed level of modelling, separate continuity equations for the liquid and vapour phases are employed along with appropriate mass transfer terms to represent the phase-change phenomena [14]. The gas-liquid interface is, however, assumed to be in dynamic equilibrium and, consequently, mixture momentum equations are used. This is also the case in the full cavitation model by Singhal et al., 2002 which has been implemented in Fluent, although the continuity equations are computed only for the mixture.

The crucial requirement of multiphase algorithms is the ability to accurately and efficiently span both incompressible and compressible flow regimes. For single phase applications, time-marching techniques have long been established as the method of choice for high-speed compressible flows, while artificial compressibility or preconditioning techniques have enabled the extension of these methods to the incompressible and low-speed compressible regimes [14]. Preconditioning methods essentially maintain proper conditioning of the controlling time-scales of the time-marching system by introducing appropriate pseudo-time derivatives. Indeed, it is widely recognized that the careful selection of these derivatives is paramount for ensuring efficiency and accuracy over a wide range of Mach numbers, Reynolds numbers and Strouhal numbers [14], [25].

Several researchers have previously reported preconditioning formulations for multi-phase mixtures. Merkle et al. 2001 employed a two-species formulation, using mass fraction as the dependent variable [14], [25]. Singhal et al., 2002 and Kunz et al., 2001 also employed a multi-species formulation, but used volume fraction as the dependent variable [5], [14]. All of these formulations assumed constant densities for both liquid and vapour phases and did not account for compressibility effects in the two-phase mixture region. Ahuja et al. 2001 has developed a multi-phase algorithm, including compressibility effects in the component phases [14]. Some authors (e.g. Kunz and Venkatesvaran et al., 2001 [14], [25]) find that all approaches are, in fact, nearly the same with only minor differences between them.

In this work, we are confronted to isothermal two-phase flows, in which the densities of the fluids are assumed to be functions of the pressure, but not the temperature. Under this assumption, the energy equation is not solved and only the continuity and momentum equations are considered (see also paragraph 3.1). It should be noted that the system is still compressible because the pressure dependence of the densities gives rise to finite acoustic speeds [14]. The isothermal compressible model serves as a useful intermediate
step between the incompressible model and the fully compressible system. The development of the fully compressible model is currently underway.

To fulfill the main goal of this work, it is necessary to know if the isothermal compressible model is generally valid for the class of cavitation problems considered here. The primary interest lies in applications where sheet- and cloud cavitation appear. These flow fields are characterized by large density ratios between the liquid and vapour states. In addition, the flows are fully turbulent at the Reynolds numbers considered within this work. Most problems also exhibit large-scale unsteadiness because of cavity re-entrant jets and cavity pinching [14] (see paragraph 4.3.).

The cavitation model implemented in Fluent is a mixture model following the Euler-Euler approach, as described above. It has been developed from the so called “full cavitation model” by Singhal et al. (2002). The phases are treated as interpenetrating continua. The model solves the continuity equation for the mixture, the momentum equation for the mixture, and the volume fraction equation for the secondary phase. Singhal proposes an isothermal compressible model for cavitation. The compressibility of the phases remains “artificial”. A shortcut through Singhal’s model and the mixture model in Fluent is given in paragraph 3.3.

Furthermore, two-phase flows can be generally characterized by the Stokes number, which is a measure of the degree of influence of the bubble-movement on the stream of the continuous phase [1]:

\[
Sto = \frac{\tau_{dyn}}{\tau_{str}} ; \quad \tau_{dyn} = \frac{\rho_b \cdot d_b^2}{18 \cdot \eta} ; \quad \tau_{str} = \frac{D}{U}
\]

where \( \tau_{dyn} \) is the response time relative to the velocity change, \( \tau_{str} \) the characteristic term of the stream (D: characteristic length, U: velocity). The response time \( \tau_{dyn} \) can be calculated from the momentum equation, considering only the drag force acting on the bubbles:

\[
m \cdot \frac{d\vec{u}_b}{dt} = 3\pi \cdot \eta \cdot d_b \cdot (\vec{u}_b - \vec{u}_n) = c_D \cdot \frac{\pi}{4} d_b^2 \cdot \frac{\rho_b}{2} \cdot (\vec{u}_n - \vec{u}_b)^2,
\]

(3.18)

when

\[
\frac{d\vec{u}_b}{dt} = \frac{24 \cdot \eta \cdot \pi \cdot d_b^2 \cdot \rho_c \cdot (\vec{u}_n - \vec{u}_b)^2}{\rho_c (\vec{u}_n - \vec{u}_b) \cdot d_b} = \frac{6 \cdot 3 \cdot \eta \cdot \pi \cdot d_b^3}{6 \cdot m} (\vec{u}_n - \vec{u}_b), \quad \frac{m}{\pi d_b^3} = \rho_b
\]
\[
\frac{d\bar{u}_b}{dt} = \frac{18 \cdot \eta}{\rho_b} (\bar{u}_c - \bar{u}_b) = \frac{1}{\tau_{dy}} (\bar{u}_c - \bar{u}_b)
\]  

(3.19)

with: \( c_D = \frac{24}{Re} \); \( Re = \frac{\rho_c (\bar{u}_c - \bar{u}_b) \cdot d_b}{\eta} \)

Regarding the Stokes number, we can differentiate between following cases:

Sto << 1 the bubbles are following passively the continuous flow

Sto \approx 1 \text{ important interaction between the phases}

Sto >> 1 the bubbles are not influenced by the continuous phase

In the case of a cavitating flow with largely subsonic velocities and bubble diameters between 10\( \cdot 10^{-6} \) m and 10 mm, it can be assumed that Sto << 1 [1]. All the studied cases within this work will be regarded as flows with very low Stokes number, for which the mixture model (Euler-Euler approach) for a homogenous flow is recommended. This leads to several simplifications in modelling two-phase flows. The homogeneous repartition of the bubbles within the continuous phase can be assumed, the phases are interpenetrating.
3.3. The Cavitation Model

Numerical simulation of cavitating flows possesses unique challenges, both in modelling of the physical features and in developing robust numerical methodology. The major difficulty arises due to the large density changes associated with phase change. For example, the ratio of liquid to vapour density for water at room temperature is over 40000. Furthermore, the location, extent and type of cavitation are strongly dependent on the pressure field, which in turn is influenced by the flow geometry and conditions [5].

The full cavitation model is an extension of the VOF-Model (Volume of Fluid). The cavitation model relies on the vapour mass fraction concept which introduces the vapour mass fraction $f_b$. The basic approach of the full cavitation model consists in using the standard Navier-Stokes equation set for variable fluid density and a conventional turbulence model [5]. The fluid mixture density $\rho$ is a function of vapour mass fraction $f_b$, which is computed by solving a transport equation coupled with the mass and momentum conservation equations. The density, $f_b$ relationship of the substitute fluid is:

$$\frac{1}{\rho} = \frac{f_b}{\rho_b} + \frac{1-f_b}{\rho_c} \quad (3.20)$$

and the vapour volume fraction $\alpha_b$ can be deduced from $f_b$ as:

$$\alpha_b = f_b \cdot \frac{\rho}{\rho_b} \quad (3.21)$$

Following relation can be established between the volume fractions of both phases:

$$\alpha_c + \alpha_b = 1 \quad \text{or} \quad \alpha_c = (1 - \alpha_b) \quad (3.22)$$

The density of the substitute fluid is defined as:

$$\rho = \rho_b \cdot \alpha_b + \rho_c \cdot (1 - \alpha_b) \quad (3.23)$$

where $\alpha_c$ and $\alpha_b$ are the volume fractions of the continuous and the vapour phases respectively.

The vapour mass fraction $f_b$ is governed by the following transport equation:

$$\frac{\partial}{\partial t} (\rho \cdot f_b) + u_i \frac{\partial}{\partial x_i} (\rho \cdot f_b) = \nabla \cdot (\Gamma \nabla f_b) + R_v - R_c \quad (3.24)$$
The source terms $R_e$ and $R_c$ denote vapour generation (evaporation) and condensation rates, and can be functions of flow parameters like pressure or characteristic flow velocity, or fluid properties like the liquid and gaseous phase densities, saturation pressure and liquid-vapour surface tension.

The present model focuses on the use of simple rational formulations for phase change rates ($R_e$ and $R_c$). To do this, the bubble dynamics consideration is firstly required. As specified in the introduction, the liquid water phase contains plenty of nuclei for the inception of cavitation. Thus, the primary focus is on proper account of bubble growth and collapse. In a liquid with zero velocity slip between fluid and bubbles, the bubble dynamics equation can be derived from the generalized Rayleigh-Plesset equation [5]. Neglecting the viscous damping term and the surface tension term, the equation can be written as:

\[
\Omega_b \frac{D^2 \Omega_b}{Dt^2} + \frac{3}{2} \left( \frac{D \Omega_b}{Dt} \right)^2 = \left( \frac{p_v - p}{\rho_c} \right)
\]

(3.25)

where $\frac{D}{Dt} = \partial_t + u_i \frac{\partial}{\partial \xi_i}$

(3.26)

and $\Omega_b$ is the bubble radius.

This equation provides a physical approach to introduce the effects of bubble dynamics into the cavitation model. Combining the Rayleigh-Plesset equation with the two-phase continuity equations (see e.g. [5]), one finally obtains for the source terms:

\[
R_e - R_c = \left[ \frac{4}{3} \pi \cdot n \cdot \alpha_b^2 \right]^{1/2} \cdot \frac{\rho_b \cdot \rho_c}{\rho} \cdot \left[ \frac{2}{3} \left( \frac{p_d - p}{\rho_c} \right) \right]^{1/2}
\]

(3.27)

This is the simplified equation for vapour transport, the right side representing the generation rate of the bubbles (source term). Though the bubble collapse process is expected to be different from that of the bubble growth, as a first approximation, equation (3.27) is also used to model the collapse, when $p > p_v$, by using the absolute value of the pressure difference and treating the right side as a sink term [5]. The local far-field pressure $p$ is taken to be the same as the cell centre pressure. The bubble pressure $p_b$ is equal to the saturation vapour pressure $p_v$ in the absence of dissolved gas [5].

In equation (3.27) all terms except “$n$” are either constants or dependent variables. In the absence of a general model for estimation of the number density, the phase change rate expression can be rewritten in terms of bubble radius, which is determined by the balance
between aerodynamic drag and surface tension forces. The correlation used in the nuclear industry can be used:

\[
\Omega_b = \frac{0.061 \, We \, \sigma_s}{2 \, \rho_c \, u_{rel}^2}
\]

For bubbly flow regime, \( u_{rel} \) is generally fairly small (5-10\%) of liquid velocity. By using various limiting arguments, for example \( \Omega_b \to 0 \) as \( \alpha \to 0 \), and the fact that per unit volume phase change rates should be proportional to the volume fractions of the donor phase, the following expressions for vapour generation \( \Gamma \) and collapse rates are obtained in terms of the vapour mass fraction \( f \). In the bubble flow regime, the phase change rate is proportional to \( u_{rel}^2 \) \[5\]. However, in most practical two-phase flow conditions, the dependence on velocity can be assumed to be linear \[5\]. The characteristic velocity \( u_{ch} \) reflects the effect of the local relative velocity between liquid and vapour. It can be assumed, that the local turbulent velocity fluctuations in most turbulent flows are of the same order that \( u_i \) (1-10\% of the mean velocity). Therefore, as a first pragmatic approximation, \( u_i \) can be expressed as the square root of local turbulent kinetic energy \( \sqrt{k} \) \[5\]. With the empiric coefficients \( C_e \) and \( C_c \), one arrives to the final formulation of the full cavitation model:

\[
R_e = C_e \frac{\sqrt{k}}{\rho_s \rho_c} \left[ \frac{2}{3} \frac{p_v - p}{\rho_c} \right]^{1/2} (1 - f) \quad \text{and} \quad R_c = C_c \frac{\sqrt{k}}{\rho_s \rho_c} \left[ \frac{2}{3} \frac{p_v - p}{\rho_c} \right]^{1/2} f \quad \text{(3.28)}
\]

Within this formulation, the effect of non-condensable gases has not been taken into account. For further specification, and for information about the determination of the empirical constants \( C_e \) and \( C_c \), please refer to the source paper.

Several experimental investigations have shown significant effect of turbulence on cavitating flows (e.g. Arndt \[12\]). Also, Singhal (see references of source paper) reported a numerical model, using a probability density function approach for accounting the effects of turbulent pressure fluctuations \[5\]. This approach required:

(a) estimation of the local values of the pressure fluctuations as:

\[
p_{\text{urb}} = 0.39 \, \rho \, k
\]

(b) computations of time-averaged phase change rates by integration of instantaneous rates in conjunction with assumed probability density function for pressure variation with time; in the present model, this treatment has been simplified by simply raising the phase-change threshold pressure value as:
\[ p_v = \left( p_{sat} + \frac{p_{turh}}{2} \right) \]

Unlike this approach by Singhal assuming single-phase, variable fluid density flows, the cavitation model implemented in Fluent has a more complicated configuration due to its capability to account for the effects of the slip velocities between phases. This capability has not been taken into account in our simulations for the reasons that have been specified at the beginning of this chapter. This could be done by switching off the “slip velocity” button in the mixture model panel. Another assumption made by Fluent, this one having much more impact upon the accuracy of the simulation results and affecting the physics of the model basements, is the incompressibility of the liquid phase: only the secondary phase is treated as compressible for the formulation of the density of the substitute fluid.

The presence of nuclei of inception in real fluids is taken into account by introducing the volume fraction of “non-condensable gases” \( \alpha_g \), which can have significant effects on both the physical realism and the convergence characteristics of the solution, even in a very small amount [20]. As a consequence, equation 3.22. is replaced by:

\[ \alpha_c + \alpha_b + \alpha_g = 1 \]  \hspace{1cm} (3.29)

The volume fraction of non-condensable gases is constant and can be set in the mixture model panel in Fluent. The dynamic viscosity of the mixture is computed from:

\[ \eta = \alpha_c \cdot \eta_c + \alpha_b \cdot \eta_b \]  \hspace{1cm} (3.30)
4. Numerical results

Several steps were necessary for the simulation of complex flows-fields. To gain insight in the capabilities of the models that have been used (turbulence and cavitation), several cases were modelled. To check the capabilities of the turbulence models, first, the unsteady flow around a circular cylinder was simulated at different Reynolds-numbers. Then the cavitation model was used in several examples, being the initial case a circular sharp edged orifice with steady turbulent flow, and finally the turbulent, unsteady, cavitating flow around a hydrofoil was simulated. For all cases, there are reliable experimental investigations, as well as some numerical research that have been taken into consideration for comparison and evaluation of the present simulation results.

The complexity of cavitating flows grows when the phenomenon becomes unsteady. This happens for example in a cavitating flow around a hydrofoil. For the reasons mentioned above, and in order to acquire more knowledge about unsteady flow fields, a classical example has been chosen for a preliminary study of a non-cavitating flow: the circular cylinder.

4.1. Circular Cylinder

The flow past a circular cylinder can be studied separately at low Reynolds numbers and at high Reynolds numbers. A complete understanding of a flow past a circular cylinder is particularly elusive because the transition from laminar to turbulent flow occurs in a distinct succession over a huge range of Reynolds numbers, and each transition state is sensitive to extremely small disturbances [6]. The simulation will be done mainly at high Reynolds numbers. Moreover, the simulation with FLUENT shows a big spread of results, especially in the drag crisis region, at Reynolds $5.3 \times 10^5$, where the laminar boundary layer becomes turbulent.

The flow around a circular cylinder falls into three distinct flow regimes: subcritical, supercritical, and transcritical. The subcritical flow indicates purely laminar boundary layer separation. In this regime, regular vortex shedding at a Strouhal number of about 0.2 is observed over a range of Reynolds numbers from roughly 200 to 100000. The supercritical regime, from Reynolds numbers of roughly 100000 to 4 million, is characterized by both a dramatic rise in the Strouhal number and a loss of organized vortex shedding altogether [6]. It
is somewhere in this regime that transition to a turbulent boundary layer begins to occur on the body at or near the point of separation. In the transcritical regime, above a Reynolds number of roughly 4 million, periodic vortex shedding re-establishes but at a higher Strouhal number of 0.26 – 0.30. The cylinder now experiences fully turbulent boundary layer separation.

The critical Reynolds number varies greatly with the surface roughness, the intensity of existing fluctuations (degree of steadiness) within the outer irrotational flow [17]. For example, the critical Reynolds number becomes lower if either the roughness of the wall surface or the intensity of fluctuations in the free stream is increased. In this work, we will consider the geometries having smooth surfaces, which means, that the roughness shape fluctuations are smaller than the thickness of the boundary layer.

Neglecting the volume forces, the general Navier-Stokes equations become (eq. 3.6):

\[
\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \nu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right]
\]  

(4.1)

Within the near-wall region (y=0), we can assume that \(u_y = u_z = 0\). For an unsteady flow, the NS-equations are reduced to:

\[
\frac{1}{\rho} \frac{\partial p}{\partial x} = \nu \left( \frac{\partial^2 u}{\partial y^2} \right)_{y=0}
\]

(4.2)

Upstream of the highest point of the cylinder the streamlines of the outer flow converge, resulting in an increase of the free stream velocity \(u_\infty\) and a consequent fall of pressure with \(x\). Downstream of the highest point the streamlines diverge, resulting in a decrease of \(u_\infty\) and a rise in pressure. Therefore, the shape of the boundary layer will strongly depend on the pressure gradient, as shows figure 4.1.1.

In an accelerating stream \(dp/dx < 0\), and therefore:

\[
\left( \frac{\partial^2 u}{\partial y^2} \right)_{wall} < 0 \text{ (decelerating)}, \quad \left( \frac{\partial^2 u}{\partial y^2} \right)_{wall} > 0 \text{ (accelerating)}
\]  

(4.3)

In contrast to an accelerating external flow, where the velocity profile decreases with the \(y\)-coordinate from a positive value to zero, a point of inflection characterises the velocity profile in a decelerating flow, where the curvature (velocity second derivatives) is zero. The shape of the velocity profiles in the lower picture (fig.4.1.1.) suggests that a decelerating pressure gradient tends to increase the sickness of the boundary layer.
Besides, the existence of this point of inflection implies a slowing down of the region next to the wall, a consequence of the uphill pressure gradient. Under a strong enough adverse pressure gradient, the flow next to the wall reverses direction, resulting in a region of backward flow. The reversed flow meets the toward flow at a certain wall point, where $\frac{\partial u}{\partial y} = 0$. Here the fluid near the surface is transported out into the mainstream: the flow separates from the wall.

In general, analytical solutions of viscous flows can be found only in two limiting cases, namely $Re \ll 1$ and $Re \gg 1$. In the case of the circular cylinder, it is interesting to study the flow at a Reynolds number increased beyond 40, at which point the wake behind the cylinder becomes unstable [17]. Photographs show that the wake develops a slow oscillation in which the velocity is periodic in time and downstream distance, with the amplitude of the oscillation increasing downstream. The oscillating wake rolls up into two staggered rows of vortices with opposite sense of rotation. Von Karman investigated the phenomenon as a problem of superposition of irrotational vortices. He concluded that a non-staggered row of vortices is unstable, and a staggered row is stable only if the ratio of lateral distance between the vortices to their longitudinal distance is 0.28. Such a staggered row of vortices behind a blunt body is called von Karman vortex-street. Figure 4.1.2. shows this phenomenon for different Reynolds numbers. The vortices move downstream at a speed smaller than the upstream velocity $u_\infty$. This means that the vortex pattern slowly follows the cylinder if it is pulled through a stationary fluid. In the range $40 < Re < 80$, the vortex street does not interact with
the pair of attached vortices. As Re is increased beyond 80 the vortex street forms closer to the cylinder, and the attached eddies (whose downstream length has now grown to be about twice the diameter of the cylinder) themselves begin to oscillate. Finally the attached eddies periodically break off alternately from the two sides of the cylinder. While an eddy on one side is shed, that on the other side forms, resulting in an unsteady flow near the cylinder. Vortices of opposite circulation around the cylinder change sign, resulting in an oscillating “lift” or lateral force.

The passage of regular vortices causes velocity measurements in the wake to have a dominant periodicity. The frequency $f$ is expressed as a non-dimensional parameter known as the Strouhal number, defined as:

$$St = \frac{f \cdot d}{u_w}$$

Experiments show that for a circular cylinder the value of St remains close to 0.21 for a large range of Reynolds numbers [17].

![Fig. 4.1.2. Streaklines in the wake behind a cylinder for different Reynolds numbers Re (Re = 32, 55, 65, 73, 102, 161).](image)
Below \( Re = 200 \), the vortices in the wake are laminar and continue to be so for very large distances downstream. Above 200, the vortex street becomes unstable and irregular, and the flow within the vortices themselves becomes chaotic. However, the flow in the wake continues to have a strong frequency component corresponding to a Strouhal number of \( St = 0.21 \). Above a very high Reynolds number, say 5000, the periodicity in the wake becomes imperceptible, and the wake may be described as completely turbulent.

At high Reynolds numbers the frictional effects upstream of separation are confined near the surface of the cylinder, and the boundary layer approximation becomes valid as far downstream as the point of separation. For \( Re < 5.3 \times 10^5 \), the boundary layer remains laminar, although the wake may be completely turbulent. The laminar boundary layer separates at approximately 82° from the toward stagnation point. The pressure in the wake downstream of the point of separation is nearly constant and lower than the upstream pressure. As the drag in this range is primarily due to the asymmetry in the pressure distribution caused by separation, and as the point of separation remains fairly stationary in this range, the drag coefficient also stays constant at \( c_D \approx 1.2 \).

Important changes take place beyond the critical Reynolds number of \( Re_{cr} = 5.3 \times 10^5 \). In the range \( 5.3 \times 10^5 < Re < 3 \times 10^6 \), the laminar boundary layer becomes unstable and undergoes transition to turbulence. Because of its greater energy, a turbulent boundary layer is able to overcome a larger adverse pressure gradient. In the case of a circular cylinder, the turbulent boundary layer separates at 125° from the toward stagnation point, resulting in a thinner wake and a pressure distribution more similar to that of potential flows. The next figure compares the pressure distributions around the cylinder for two values of \( Re \), one with a laminar and the other with a turbulent boundary layer. It is apparent that the pressures within the wake are higher when the boundary layer is turbulent, resulting in a sudden drop in the drag coefficient from 1.2 to 0.33 at the point of transition. For values of \( Re > 3 \times 10^6 \), the separation point slowly moves upstream as the Reynolds number is increased, resulting in an increase of the drag coefficient. It must be noted that the critical Reynolds number at which the boundary layer undergoes transition is strongly affected by two factors, namely the intensity of fluctuations existing in the approaching stream and the roughness of the surface, an increase in either of which decreases \( Re_{cr} \).

### 4.1.1. Preliminary discussion and numerical implementation

At Reynolds numbers of roughly 200 or less, many researchers have successfully computed the Strouhal number and mean drag over a circular cylinder [6]. At higher Reynolds numbers, however, two-dimensional numerical methods cannot predict the lift and drag forces accurately, due to the increasingly prominent three-dimensionality of the real flow field.
Additional forces acting along the cylinder-axis are the main error sources regarding the 3D-effects. Furthermore, according to latest researches, the difficulties of three-dimensional modelling of flows around circular cylinders could arise due to non accurate physical models and mathematical formulations concerning the conservation of helicity, which could block the energy cascade and alter the Kolmogorov $k^{-5/3}$ scaling (e.g. see reference [8]).

Nonetheless, it is still important to understand and to characterize the capabilities and limitations of existing two-dimensional numerical methods at higher Reynolds numbers, since these methods may yield a deeper insight into the physics at a relatively low cost.

Any attempt to numerically model circular cylinder flow is complicated not only by the fact that the flow above a Reynolds number of around 180 is three-dimensional, raising doubts about the applicability of two-dimensional simulations. Additionally, transition occurs off-body in the wake or shear layer at Reynolds numbers between roughly 200 and the supercritical regime [6]. Without performing very expensive direct numerical simulations (DNS), this behaviour is not captured by numerical methods that solve the Navier-Stokes equations on typical grids used for hydrodynamic analysis. This deficiency may or may not be important at lower Reynolds numbers, depending on how far behind the cylinder transition occurs and what feature of the flow is of interest [6].

But it certainly has an adverse effect at higher Reynolds numbers for which transition occurs at or near the separation point on the cylinder. For Reynolds numbers at and above the supercritical regime, Reynolds-averaging with the use of a turbulence model is one way to introduce the important effect of turbulence into a numerical simulation. However, without an accurate built-in transition model, it is difficult, if not impossible, to model the important effects of transition, particularly when it occurs on or near the body. The turbulent models used here do not fulfil optimal conditions to predict these complex flow characteristics. It is not surprising, then, that most numerical studies of the flow around a circular cylinder have focused primarily on low Reynolds number flows less than about 1000.

In this work, an investigation is made into the ability to numerically predict the hydrodynamic properties of a circular cylinder across a range of Reynolds numbers from 3000 to 1 million, and thus confirm previous simulation researches whose results we partially described above.

The unsteady flow field is computed with the Reynolds-averaged Navier Stokes (RANS) flow solver Fluent v6.1, employing a variety of turbulence models, as well as the laminar Navier-Stokes equation to prove accuracy. The computations are two-dimensional and time-accurate. Water was used as an incompressible fluid in all computations. Body forces have been neglected, viscous and pressure forces having been assumed to be much larger. Second order discretisation schemes have been set for all terms to compute the flow as well as for the unsteady solver. The SIMPLE (semi-implicit pressure linked equation) velocity -
pressure correction algorithm has been enabled for all computations. Four different hybrid meshes have been used for comparison: a fine mesh (mesh 01, ≈ 50000 nodes), an intermediate mesh (mesh 02, ≈ 32000 nodes), a coarser mesh (mesh 03, ≈ 14000 nodes) and a very coarse mesh (mesh 04, ≈ 10000 nodes). The smallest cell size in the direction from the cylinder wall is $10^{-5}$ m for the fine mesh. The computational domain is shown in figure 4.1.3. The intermediate mesh (mesh 02) around the cylinder is shown in figure 4.1.4.

![Fig. 4.1.3. Computational domain for the circular cylinder.](image)

The reference pressure is the ambient pressure, $P_{outlet} = 1.01325$ bar. The inlet velocity using a constant profile was calculated from the suited Reynolds number:

$$\text{Re} = \frac{u_\infty \cdot d \cdot \rho}{\eta},$$

where $d$ is the diameter of the cylinder, $\rho$ the water density, and $\eta$ the dynamic viscosity of water at ambient temperature, and $u_\infty$ the constant flow velocity.
### 4.1.2. Mesh sensitivity control and numerical results

The drag and the lift forces ($F_D$ and $F_L$ respectively) are the x and y-components of the resulting force upon the cylinder. They are very important for the study and the analysis of the results of the flow over a circular cylinder. They are calculated as follows:

\[
F_D = \frac{1}{2} \cdot c_D \cdot \rho \cdot u_\infty \cdot A \quad \text{and} \quad F_L = \frac{1}{2} \cdot c_L \cdot \rho \cdot u_\infty \cdot A,
\]

**Fig. 4.1.4.** Grid distribution for the circular cylinder, mesh 02.
where \( A \) is the specific area of the submerged body, \( c_L \) the lift coefficient and \( c_D \) the drag coefficient. For a circular cylinder, \( A \) can be written as:

\[
A = d \cdot b,
\]

where \( b \) is the length of the cylinder and \( d \) its diameter.

In a 2D case, it can be assumed that the cylinder length is infinite, or \( b \gg d \) (although, 3D effects along the axis seem to play an important role). Therefore, for the computation of the drag and lift coefficients and especially the plot of the drag and lift convergence, the computation area was made by unit of length, i.e., \( b = 1 \text{m} \), with a value of the area resulting in \( A = 0.02 \text{m}^2 \). A typical plot of lift and drag convergence is shown in fig.4.1.5. The convergence criterium is the constant sinusoidal periodicity of the drag and lift fluctuations, without neglecting of course, the drop of the residual curves.

As it can be seen on the graph (fig.4.1.5.), the drag has a double frequency compared to the lift: whereas one vortex appears upward, two vortices simultaneously shed in the wake behind the cylinder. The time step of the calculations defined in Fluent has been chosen to be between 1/10 and 1/20 of the vortex shedding. Also, the lift coefficient oscillates between negative and positive values symmetrically around zero, while the drag is always positive. Obtaining these convergence curves, the average drag value could be read off the graphic and compared to the experimental data. The time step of one full oscillation could be averaged from at least ten cycles of the drag and be converted into frequency for the calculation of the Strouhal number.
The most difficult task regarding the simulation of a flow past a circular cylinder was the setting of the time step. The velocity of the flow field could be calculated for each Reynolds number. The corresponding time step could be obtained from the Strouhal number, which was taken from literature (e.g. Zdravkovich, 2003) and set to be equal to 0.2 for all computations. The resulting time step \(1/f\) corresponds to one cycle of the lift force. In order to visualise the oscillating phenomenon with Fluent, the time step \(1/f^*\) to be set for simulation was chosen to be a fraction of the time step obtained from the Strouhal number (e.g. \(f^* = 20f\)).

Nevertheless, this adjustment required some experience since finding an adequate fraction of the frequency in order to obtain clear convergence curves as shown in figure 4.1.5. was a difficult task. Depending on the Reynolds number and on the turbulent model, the chosen time step could be too large and in some cases even too small! The time step definition was not only problematic for the simulations of the flow past a circular cylinder, but as we will see, also for the computation of cavitating flows. It did often occur that a change in amplitude and frequency of the lift coefficient led to convergence after changing the time step.

A mesh sensitivity analysis has been done using the Spalart-Allmaras and the SST k-\(\omega\) turbulent model at Reynolds 5.0e+04 and at Reynolds 1.0e+05. Asymptotic stability has been obtained because the mesh 01 and the mesh 02 offer similar results for the drag coefficient in all cases, as it can be verified in the figures from appendices A.1 and A.2. However, this asymptoticity does not guarantee good numerical results. Choosing an adequate time step for example, also influences the accuracy of the results.

Surprisingly, it can also be observed, that the mesh 04 offers best results compared to the experimental data. It is widely recognized that it is very difficult to obtain good results near the drag crisis region, and a big spreading of numerical results have been reported, see e.g. Cox, 1997) [6] (see more details in appendix A.7-A.8). The illusive accuracy of the drag coefficient obtained with the mesh 04 is probably due to pure coincidence, since the same computation using the mesh 03 gives very unrealistic findings.

The Spalart-Allmaras model has also been used for the mesh sensitivity control (see A.3, A.4). It is known from previous studies (see e.g. Coussirat, 2003 [3]) that this turbulent model is very sensitive for this kind of analysis because it revealed a very strong mesh dependence, so that the k-\(\omega\) SST model should be rather considered as reference.

Evidently, all cases simulated for refinement have been executed with identical computational set-ups and parameter settings. The Standard k-\(\varepsilon\) model has not been used for grid adaptation, as computations using the mesh 01 and the mesh 02 require a different boundary-layer approach compared to the coarser meshes. It has been assumed that the
present approach is valid for all Reynolds numbers, so that the mesh 02 could guarantee accuracy for all simulations.

At Reynolds 5.0e+04 a big difference in the results using the meshes 01 and 02 (see appendix A.2) is due to an imperfect convergence of the drag coefficient: (the average drag could be clearly determined, but the drag oscillations are not perfectly sinusoidal,). This example perfectly shows the impact of the time step definition on the drag and lift convergence and thus on the obtained Strouhal number.

<table>
<thead>
<tr>
<th>Defined Parameters</th>
<th>Simulation results</th>
</tr>
</thead>
<tbody>
<tr>
<td>Re</td>
<td>Velocity (u)</td>
</tr>
<tr>
<td>100</td>
<td>5.02E-03</td>
</tr>
<tr>
<td>200</td>
<td>1.00E-02</td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td>5.0E+04</td>
<td>2.51E+00</td>
</tr>
<tr>
<td>1.0E+05</td>
<td>5.02E+00</td>
</tr>
<tr>
<td>1.4E+06</td>
<td>7.02E+00</td>
</tr>
<tr>
<td>1.0E+06</td>
<td>5.02E+01</td>
</tr>
</tbody>
</table>

Fig.4.1.6. Velocity magnitude plot for: a) Re = 5.0e+04,  b) Re = 1.0e+05,  c) Re = 1.0e+06; (mesh 02), see graphics in appendices A1-A8.

The computations were done in two stages. First a steady model was used to find a steady solution and after this one converged, the unsteady module was started with initial values corresponding to the steady solution. Using the 2\textsuperscript{nd} order scheme for the unsteady solutions led in most cases to divergence, so that only 1\textsuperscript{st} order schemes have been used.

Several computations have been done at low Reynolds number using a solver for steady flows (without turbulent models). At Reynolds 2300, the flow becomes turbulent. As show the annexes A.5 and A.6, the results become worse with increasing Reynolds number, especially the drag values. This is due to the effects of turbulence that cannot be captured with this model.

Further, near the drag crisis region, the turbulent models discussed in the third chapter of this work have been used for simulation. The table in fig.4.1.6. gives an overview of the solved cases. From the results, no clear conclusions can be made concerning the most adequate turbulence model to be used at these Reynolds numbers. It can still be confirmed that all used turbulent models show difficulties to accurately predict the important effects of transition (refer also to chapter 3 and the preliminary discussion within this paragraph). Regarding the different wakes generated at the different Reynolds numbers, some plots show good accuracy with visualisation experiments and other numerical simulations. In fig.4.1.7. the
vorticity plot at $Re = 100$ can be compared with the corresponding picture in fig.4.1.2. (even though this graphic shows the streaklines). As the computational domain is shorter than the experimental domain from fig.4.1.2., the regular vortices that begin to form at about 30-35 cylinder diameters in the wake cannot be seen in the simulation.

In fig.4.1.8. the vorticity magnitude at $Re = 1.0e+05$ is plotted and compared with the results obtained by Cox, 1997 [6].

The plots of velocity magnitude at different Reynolds numbers in fig.4.1.9. show a good concordance with the theoretical approach discussed at the beginning of this chapter.
Fig. 4.1.9. Velocity magnitude plot for: a) $Re = 5.0e+04$, b) $Re = 1.0e+05$, c) $Re = 1.0e+06$; (mesh 02).
4.2. Sharp-edged orifice

In this part of the work, cavitation characteristics were computationally determined for sharp edged orifices. Cavitation bubbles form due to a very low static pressure that occurs in this case near the sharp inlet corner. This low static pressure is predicted by incompressible potential flow theory, which indicates that flow around a sharp corner, (e.g. a corner with infinitesimally small radius of curvature), will have infinite negative pressure [18]. This physically impossible result is a direct consequence of the constant density restriction, which can be encountered in some cases solved in Fluent without using the cavitation model. In real cavitating flows, the density of the fluid (water) decreases with decreasing pressure, most likely leading to a change in phase. The sharper the corner and the higher the velocity, the more likely cavitation is to occur. High velocities near the orifice entrance generate those zones of very low pressure right after the constriction, which reduces the flow rate (chocking type phenomenon, [5]) and can lead to surface damage downstream of the orifice.

4.2.1. Experimental set-up

In the case of a sharp inlet, where the flow separates at the corner, the flow experiences a vena contracta. A drawing in figure 4.2.1. shows a sharp entrance flow. After the vena contracta, the flow re-establishes and a boundary layer starts growing until exit from the orifice.

![Characteristics of flow in a sharp edged orifice.](image)
The resulting velocity profile will depend on the Reynolds number (based on the orifice diameter) and development distance (i.e. x-position within the orifice). If the inlet pressure is high enough and the cavitating edge sufficiently sharp, the cavity inside the nozzle grows and reaches its outlet. As this occurs, the downstream ambient air finds a way to flow into the orifice, resulting in a so called hydraulic flip (e.g. Vahedi et al, 2004) [18].

Nurick, 1972 [9], has published extensive experimental data for cavitation in a sharp-edged circular orifice. In his paper, the author develops a cavitation model to establish accuracy to justify the experimental results. Using the continuity equation, the contraction coefficient $c_c$ can be defined as:

$$c_c = \frac{A_c}{A_1}$$

(4.4)

where $A_c$ represents the effective flow area through the contraction and $A_1$ represents the geometrical area of the orifice. The value of the contraction coefficient varies with the orifice geometry and cavitation characteristics. For a very rounded entrance, the flow will not separate and the coefficient of contraction will be unity. For a short orifice with a sharp entrance, as it will be used in the present case, an empirical value of $c_c = 0.62$ was taken [5],[9].

Another relevant integral property of the flow is the coefficient of discharge, $c_d$. The coefficient of discharge represents the efficiency of the orifice between inlet and outlet and thus it is a

Fig. 4.2.2. Cavitation characteristics in Nurick’s experiments with a lucite orifice, L/d = 5.
measure of whatever losses occur in the orifice. Nurick defines the coefficient of discharge as:

\[ c_d = c_e \cdot \sqrt{\sigma} = \frac{m}{A_i \cdot \sqrt{2 \cdot \rho_e \cdot (P_0 - P_1)}}, \]  

(4.5)

where \( \sigma \) is the cavitation number defined as:

\[ \sigma = \frac{P_0 - P_v}{P_0 - P_1} \]  

(4.6)

These definitions assume the losses to occur between the vena contracta and the orifice exit, when the Bernoulli formulation is used.

Nurick plotted the coefficient of discharge versus the cavitation number on log-log axes in order to verify the square root dependence of \( c_d \) on \( \sigma \). He observed that the data from the cavitating region lay on a straight line with a slope of one-half, where \( c_e \) is the value of the Y-intercept. At some point, the value of the cavitation number is high enough so that the orifice no longer cavitates. The higher values of \( \sigma \) occur when the difference between the upstream and downstream pressure is small. At high values of \( \sigma \) the coefficient of discharge stays fairly constant. This variation occurs because the coefficient of discharge is no longer a function of \( \sigma \), but depends on the Reynolds number instead. Thus, the point of inflection of Nurick’s graphic indicates the inception of cavitation (see also fig.4.2.5.).

4.2.2. Numerical implementation and boundary conditions

Within the present work, it has been intended to reproduce Nurick’s correlation, predicting different discharge coefficients and comparing with the experimental results. The main objective was to check if the cavitation model used would predict inception of cavitation.

Geometrical parameters of the circular orifice are \( D/d=2.88 \) and \( L/d=5 \), where \( D,d, \) and \( L \) are respectively the inlet diameter, the orifice diameter and the orifice length. All sizes correspond to the lucite orifice used by Nurick to visualize his results. Experiments and simulation have been done with a fixed exit pressure, \( P_b=0.95 \) bar. The pressure taps that Nurick placed at \( \frac{1}{4} \) and \( \frac{1}{2} d \) downstream of the inlet of the orifice to measure the cavity pressure have also been simulated by defining 2 points at these locations and plotting the corresponding local static pressure. The taps have been defined at 0.095mm in radial direction from the wall, where the diameter of the orifice is 7.62mm (in the case of the coarse mesh this corresponds to 50% of the distance from the wall of the first mesh point). The upstream static pressure, \( P_1 \), was
varied between 9.6e+04 pa and 5.0e+07 pa to generate the flow at different coefficients of discharge (fig. 4.2.5.).

The flow is 2-D but axisymmetric. The inlet boundary condition is the specified upstream pressure with an inlet velocity assumed to be zero. Upper boundaries defining the contours of the orifice are no-slip walls. At the pressure outlet boundary, the fixed pressure of $P_b = 0.95$ bar has been set for all cases. Figure 1 shows a close-up of the coarse grid (fig. 4.2.3.a) and an outline of the computational domain with all boundary condition types (fig.4.2.3.b). The smallest cell size in the near-wall region is for the coarse grid a 2.5% of the orifice diameter. All cases have been calculated with double precision. The solving strategy used was the steady, SIMPLE algorithm. Second order discretisation schemes have been used for the calculation of the flow terms.

All of the described applications have been generated using the Standard $k-\varepsilon$ model with Standard Wall Functions. A plot of the $y+$ values has shown that applying this model is accurate, since $25 < y+ < 60$ along the orifice wall for the intermediate grid.

The cavitation model (Multiphase Model – Mixture) was configured with a vaporization pressure of $P_{vap} = 3540$ Pa, a percentage of non condensable gas of $1.5e-05$ and no slip velocity between the phases. Water-vapour with a density of $0.02558$ kg/m$^3$ and a viscosity of $1.26e-06$ kg/(m·s) has been defined as the secondary phase.
4.2.3. Mesh refinement analysis

To guarantee mesh independence results, different meshes have been tested for the simulation. Best results have been obtained with only one mesh, used by Singhal with a total of 3700 nodes (intermediate grid). First refinement has been done with a second mesh of 7400 nodes (fine mesh). Finer meshes did not provide acceptable results (compare with Fluent v6.2 at the end of this chapter). For comparison, a third mesh of 1900 nodes has been successfully tested. The refinement analysis has been done using the same computational parameter set-up and conditions for all meshes. The inlet pressure has been set at 5.0E+05 Pa.

![Pressure profile graph](image)

Fig. 4.2.4. Mesh refinement analysis for $P_{\text{inlet}} = 5.0E+05$ Pa using 3 different grids.

Figure 4.2.4. shows the plot of static pressure along the orifice wall and provides evidence about the exactitude of the results obtained with the simulations of the flow through a sharp edged orifice. The red curve represents the pressure plot obtained with the 3700 node grid. The fine grid (7400 nodes) is represented by the blue curve, while the black curve shows the pressure characteristics along the wall for a grid with 1900 nodes. As the graphic shows, the differences between both curves are minimal at the region where the cavity is located (a 0.1 to 3% of error), so that the intermediate grid seems to offer a good approach. The unpredictable hydraulic flip as well as the constant position of the reattachment zone at the orifice wall documented with the vector velocity plot (see next paragraph) may be a result of a too coarse grid. The error percentage at $C = 0.012$ where reattachment should occur at lower
inlet pressures, is about 11% and corresponds to the maximum discrepancy of the static pressure curves between the fine and the intermediate mesh.

Varying the inlet pressure led to a drop of the $y^+$ values near the orifice wall. This would have implied the use of enhanced wall treatment ($k$-$\varepsilon$ model). This could not be achieved because the condition for mesh refinement which requires a strict similarity at parameter definitions would have been infringed (this is only valid if $k$-$e$ model is used, for S-A and $k$-$\omega$ models this problem does not appear, see also chapters 3.1.1. – 3.1.3.). Finer meshes could not be used for the same reason. Because of unknown reasons, the simulation of the mentioned cases led to insufficient results when a finer mesh was used. As a consequence of our mesh sensitivity analysis, the intermediate mesh has been used for all simulations, assuming that further refinement would provide similar findings.

### 4.2.4. Simulation results

The inlet pressures and the corresponding discharge coefficients and cavitation numbers calculated as described before (eq. 4.5 and 4.6) are listed below [Fig. 4.2.5.].

<table>
<thead>
<tr>
<th>sigma</th>
<th>Inlet static pressure [$\times 10^5$ Pa]</th>
<th>discharge coeff.</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.96</td>
<td>1.90</td>
<td>0.890</td>
</tr>
<tr>
<td>1.07</td>
<td>2.02</td>
<td>0.940</td>
</tr>
<tr>
<td>1.70</td>
<td>2.11</td>
<td>0.930</td>
</tr>
<tr>
<td>1.77</td>
<td>2.13</td>
<td>0.827</td>
</tr>
<tr>
<td>1.76</td>
<td>2.15</td>
<td>0.823</td>
</tr>
<tr>
<td>1.72</td>
<td>2.23</td>
<td>0.813</td>
</tr>
<tr>
<td>1.59</td>
<td>2.50</td>
<td>0.782</td>
</tr>
<tr>
<td>1.45</td>
<td>3.00</td>
<td>0.747</td>
</tr>
<tr>
<td>1.33</td>
<td>3.75</td>
<td>0.715</td>
</tr>
<tr>
<td>1.23</td>
<td>5.00</td>
<td>0.688</td>
</tr>
<tr>
<td>1.10</td>
<td>10.00</td>
<td>0.650</td>
</tr>
<tr>
<td>1.02</td>
<td>50.00</td>
<td>0.626</td>
</tr>
<tr>
<td>1.01</td>
<td>100.00</td>
<td>0.623</td>
</tr>
<tr>
<td>1.00</td>
<td>500.00</td>
<td>0.620</td>
</tr>
</tbody>
</table>

![Fig. 4.2.5. Reproduction of Nurick’s correlation with Fluent.](image)

For each case, the inlet pressure has been increased according to the table in Fig. 4.2.5. The inception of cavitation was verified by plotting the contours of static pressure and asserting whether the minimal value exceeded or not the vapour pressure. This was reached at $\sigma = 1.76$, which fits very well with Nurick’s results (fig. 4.2.6.).
Nurick’s correlation can be plotted following the data of the table: from $\sigma = 1.00$ to $\sigma = 1.76$ the flow cavitates ($P_{\text{stat,min}} < P_{vap}$) and is described by equation 4.5. When $\sigma < 1.76$, the cavity pressure should remain constant and equal to the vapour pressure until hydraulic flip occurs. For an orifice with $L/d = 5$, Nurick observed that hydraulic flip occurred slightly after the inception of cavitation. It was not possible to simulate the hydraulic flip, even perfect sharpness of the orifice being accomplished. Hydraulic flip cannot be predicted without the use of a VOF model (see also Vahedi et al., 2004 [18]). Also, deficiencies in the software implementation make the cavity pressure decrease much beneath the vapour pressure, which cannot correspond to reality. Fluent programmers argue that this is a difficult numerical task. In the latest version, the default minimal pressure corresponds to 50% of the vaporization pressure. Fluent assures that the default value in the Fluent 6.2 version will be 85% of $p_{vap}$.

For $\sigma > 1.76$, equation 4.5. cannot be applied anymore; the cavity pressure must substitute the vapour pressure and the losses described by the discharge coefficient can be assumed to be constant when $\sigma$ varies. Unfortunately, Nurick plotted his results only for $L/d = 10$ and $L/d = 6$. Since the difference between both graphics are minimal, we assumed the same for an orifice with $L/d = 5$.

![Fig. 4.2.6. Static pressure and velocity magnitude vectors at $\sigma = 1.76$.](image)

---

1 Fluent Inc. 2004, personal communication.
Cavitation occurs after the local static pressure has reached the vapour pressure of water at ambient temperature. Nurick observed the appearance of a fuzzy region near the inlet at $P_1 = 1.67 \times 10^5$ Pa (corresponding to $\sigma = 2.27$ if we assume that $P_{vap} = 3540$ Pa also for Nurick’s experiments) for which the cavity pressure measured with the tap did not reach the vapour pressure yet, see Fig. 4.2.2. In the present simulation, the contours of volume fraction also reveal the presence of vapour within the continuous water phase before the vapour pressure was reached, see Fig. 4.2.7. This can also be considered as a good result, although the fuzzy region has been observed at higher inlet pressures ($\sigma = 1.82$) than in Nurick’s experiments. Thus, Fluent predicts inception of cavitation before the static pressure reaches the vapour pressure. This phenomenon is real, and describes what Nurick calls “fuzz”, occurring when the flow is two-phase. Besides the differences between experiments and the present simulation, regarding the inlet pressure at which “fuzz” can be observed, another divergence is to be noticed.

Fig. 4.2.7. Volume fraction of the vapour phase $\alpha$, at $\sigma = 1.82$.

Nurick describes a separation region that lengthens to about four orifice diameters before reattachment occurs, and assumes it to be the consequence of the fuzzy region of the orifice inlet. Numerical results obtained at $\sigma = 1.82$ show a separated flow region but the reattachment occurs at less than $\frac{1}{2}$ orifice diameter. It seems important to say, that according to the obtained simulation results, the position along the x-axis at which reattachment occurs varies very few when the inlet pressure is increased. For very high inlet pressures where the cavity length exceeds two orifice diameters, reattachment occurs already at a distance of about $\frac{1}{2}$ orifice diameter (fig. 4.2.8.). The cavity length is about two orifice diameters and is
much shorter than it was observed by Nurick, who predicts hydraulic flip at $\sigma = 1.7$. Figure 4.2.8. shows a plot of the vapour volume fraction at $\sigma = 1.0$ and the corresponding vector plot of the velocity magnitude: reattachment occurs already at $\frac{1}{2}$ orifice diameter. This result can be explained by the fact that the plot shows the velocity field of the mixture.

Finally, a series of simulations have been done to reproduce the relation between the cavity pressure and the upstream pressure. A comparison between the cavity pressure (measured at the two taps) obtained by Nurick as a function of the upstream pressure and the same data obtained in the present simulation is shown in fig. 4.2.9.

The linear reduction of the cavity pressure obtained by Nurick for the tap at $\frac{1}{4}$ diameter gives evidence of the fact that the vena contracta is at or near that location. This does not happen in the simulation, which lets us deduce that the vena contracta is much shorter in numerical calculations than the experiments show. The curve obtained by Nurick for the tap placed at $\frac{1}{4}$ orifice diameter corresponds to the simulation plot of the static pressure at $x = \frac{1}{2}$ orifice diameter. The plateau region in Nurick's curve obtained for the first tap corresponds to the appearance of fuzz, and thus what he called the inception of cavitation.

The location of the taps was shown in figure 4.2.3. Further increasing the upstream pressure, leads to a drop of the static pressure within the cavity beyond the vapour pressure. A cavity forms and grows as shown in figure 4.2.10. with increasing $\sigma$. At $\sigma = 1.02$ hydraulic flip is not reached.

\begin{figure}
\centering
\includegraphics[width=\textwidth]{fig428.png}
\caption{a) Volume fraction of the vapor phase $\alpha_v$ and b) vector velocity plot at $\sigma = 1.02$.}
\end{figure}
The solutions show robust convergence characteristics, with residual curves dropping more than ten orders of magnitude. Figure 4.2.11. shows typical convergence plots for the orifice cases: while an initial plateau represents initial condition errors being convected out, the following abrupt curve drop shows the solutions converging rapidly.

**Fig. 4.2.9.** Cavity pressure characteristics as a function of upstream pressure, comparison Nurick and present calculation.

**Fig. 4.2.10.** Volume fraction of the vapor phase $\alpha_v$ at

\[ \sigma = 1.64; \sigma = 1.55; \sigma = 1.52; \sigma = 1.50; \sigma = 1.32; \sigma = 1.02. \]
The poor results obtained for high pressure differences between inlet and outlet when using fine meshes are probably due to software deficiencies. Other meshes with finer closure have been tested. Only the intermediate mesh offered acceptable results with good convergence characteristics for these conditions. The pressure correction factors for the AMG solver have been set at 0.4. Wall function options have been adapted to the obtained y+ minimum and maximum values for each grid. Under-relaxation factors have been set very low as well as a maximum of iterations per time step to ensure a more stable iteration process. Other turbulent models than the standard k-ε model have also been tested with these meshes without success (Spalart-Allmaras, RNG k-ε, SST k-ω).

At the present time, calculations have been done with the new version of Fluent. Good results have been obtained: very fine meshes could be tested using all above mentioned turbulence models. The earlier version was numerically very unstable, and allowed computations using the cavitation model only coupled with the k-ε turbulence model. A mesh refinement has been done using five different meshes. The results obtained for this mesh sensitivity analysis can be seen in figure 4.2.12. The better quality of the results at this mesh refinement, and particularly the numerical stability of Fluent v6.2 encourages to perform computations of cavitating flows using this version.

The unsteady turbulent cavitating flow past a hydrofoil described in the next chapter has been simulated using Fluent 6.1. The latest version will have to be used for computations in future work considerations.

Fig. 4.2.11. Convergence characteristics for $P_{\text{inlet}} = 5.0\times10^5$ pa
Fig. 4.2.12. Mesh sensitivity control using 5 different meshes ("skw_0.5" coarsest and "skw_4.0" finest, standard k-ω model, Fluent v6.2
4.3. Hydrofoil

The turbulent wake generated by streamlined bodies like hydrofoils has been investigated numerically as well as experimentally in the past. But the details of flow near the trailing edge are still poorly understood [15]. The unsteady turbulent wake behind a cavitating hydrofoil is even more complex. The formation of individual bubbles and subsequent development of attached cavities, bubble clouds, etc., makes the unsteady structures in the wake of a cavitating hydrofoil fundamentally different from those in the wake of a non-cavitating hydrofoil. A detailed description and analysis of cavitation phenomena with different cavitation number and angles of attack can be found in Arndt et al., 2000, and Song and Qin, 2001.

It is known that cavitation inception occurs at composite parameter ($\sigma/2\alpha$) of about 8.5, where $\sigma$ is the cavitation number and $\alpha$ the angle of attack [15]. It has been reported by Song and Qin, that when $\sigma/2\alpha$ is slightly lower than 8.5 (between 6 and 8), a bubble cavity will first appear near the leading edge on the suction side when the lift coefficient is maximum, at the location with minimum pressure [15]. This bubble, which is also an eddy, slides down along the suction surface and the lift decreases to a minimum when it arrives at the tail of the foil. The lift starts to increase again as the eddy moves away in the wake. A new bubble cavity occurs when the lift increases to a maximum again and the process repeats itself. This

Fig. 4.3.1. Pressure and vorticity fields of a) sheet cavitation [$\sigma = 1.5$], b) sheet/cloud cavitation [$\sigma = 1.3$].

Source: Kubota, 1992, $\alpha = 8^\circ$ for all cases.
phenomenon is called **bubble cavitation**. Bubble cavitation can change to a less stable **bubble/cloud cavitation**. A cloud cavity may follow a small bubble at the instant of maximum lift. When the cloud cavity collapses near the trailing edge, the lift drops to a minimum, the drop being this time characterized by intense fluctuations [15].

Further reducing $\sigma/2\alpha$ (between 4 and 6) results in the generation of sheet or more unstable **sheet/cloud cavitation** near the leading edge of the suction side (fig.4.3.1.). This is the most dynamic flow regime related to partial cavitation. This cavity grows and breaks off to form a cloud cavity at about 1/3 chord length due to the re-entrant jet [15], [19]. The cloud cavity collapses at near ¾ chord to generate a strong shock wave [15]. Song and Qin estimates the average Strouhal number to be about 1.5 accentuating the fact that this is an unsteady and "inherently turbulent phenomenon with significant degree of randomness". Curiously, Berntsen calculates a Strouhal number based on the maximum foil thickness as reference length to be about 0.2 for similar conditions.

Figure 4.3.2. shows the contours of void fraction for a NACA0015 hydrofoil at $\sigma = 1.0$, $\alpha = 8^\circ$ and lift plot, Kawakami, 2004.

In most cases, according to Song and Qin, the cloud cavity breaks up and collapses into more than one piece. When $\sigma/2\alpha$ falls below 4, the flow pattern is changed significantly. The maximum cavity length exceeds the chord length and the cavity oscillates between partial cavity and **super cavitation** [15].
Within the present work, we intended to predict bubble, bubble/cloud and sheet/cloud cavitation using Fluent 6.1. Although most of the documented investigations used LES (Large Eddy Simulation) or/and user defined cavitation models for the simulation of this type of cavitation phenomena for a flow around a hydrofoil, we mostly tested Fluent’ s ability to solve the case(s) described below using RANS equations and the implemented multi-phase model. As for the case of a sharp-edged orifice and despite of a very rigorous procedure, the results have been good only for single settings and under very restricted conditions. We first intended to follow the investigation of Kubota (1992) and base our research on his results for a hydrofoil under different angles of attack and a Reynolds number of 3e+05. Further, another case has been simulated, the calculations being set up to resemble the simulations taken from the paper of Berntsen et al. (2001). Here, a Reynolds number of 1.2e+06 based on chord length was used; the flow was at σ/2α of 5, with an angle of attack of 8º.

Kubota already found important results concerning bubble, bubble/cloud and sheet/cloud cavitation. Like Arndt, Song, Qin, etc. he suggested that vortex cavitation is often observed downstream of attached cavitation. It is caused by vorticity shed into the flow field just downstream of the cavity. Such vortex cavitation generates a large cavitation cloud under certain conditions [10]. The vortex cavitation impinges on the body where its subsequent collapse results in erosion. Kubota describes the cavitation phenomenon more precisely in his formulation of bubble two-phase flow cavity model.

The cases of Kubota 1992 and Berntsen, et al. 2001 were selected to extensively check the capabilities of the cavitation model implemented in Fluent. These benchmarks were selected because both experimental measurements and complete information concerning the parameters used in the numerical computations are available.

4.3.1. Simulation based on the case of Kubota (1992)

In the case of Kubota, we have restricted our investigation to the case where the flow is at σ =1.2. This corresponds to a composite parameter σ/2α of about 4.3. A suitable mesh has been generated for the NACA0015 hydrofoil, see Fig 4.3.3. The computation was performed at α = 0º, 8º and 20º. The Reynolds number Re, based on the uniform flow velocity and chord length of the hydrofoil, was 3e+05. The computation at α=0º was performed only for non-cavitating conditions to evaluate numerical accuracy obtained in the flow pattern prediction.
4.3.1.1. Computational domain and boundary conditions

The grid is a hybrid grid type with a total of about 30700 nodes. The smallest cell size in the direction from the wall is 0.005mm, which corresponds to approximately $1/350 \times \text{Re}^{-1/2}$. The results in the first grid point yield $y^+ < 3$.

The distance from the leading edge to the inlet boundary is 16 to 30C, where C is the chord length of the hydrofoil (C = 0.1m). The distance between trailing edge and the outlet boundary of the computational domain is 20C. Figure 4.3.3. shows a close-up of the 2D grid of the NACA 0015 – hydrofoil used for this simulation. The parabolic-shaped inlet boundary is a velocity inlet with constant velocity profiles. The outflow boundary condition is set with constant pressure (reference pressure). This configuration permits the flow to be treated as unconfined. The angle of attack was simulated by defining the inlet velocity by its components corresponding to the appropriate Reynolds number. The Reynolds number $\text{Re}$, pressure coefficient $c_p$ and cavitation number $\sigma$ are defined as follows:

$$
\text{Re} = \frac{u_\infty \cdot C \cdot \rho_c}{\eta_c}, \quad c_p = \frac{p - p_{ref}}{\frac{1}{2} \rho_c \cdot u_\infty^2}, \quad \sigma = \frac{p_{ref} - p_{vap}}{\frac{1}{2} \rho_c \cdot u_\infty^2}
$$

where $u_\infty$ is the uniform flow velocity, $\rho_c$ is the density of liquid water, $\eta_c$ the dynamic viscosity of water, $p_{ref}$ the reference pressure or operating pressure and $p_{vap}$ the water vapour pressure. Using these definitions and the constant parameters mentioned above ($\sigma$, $\text{Re}$, C, $\rho_c$, $\eta_c$) the reference pressure could be calculated. The vapour pressure was 2400 Pa.

Fig. 4.3.3. Grid distribution for the hydrofoil, the Kubota case reproduction.
4.3.1.2. Simulation results and discussion

The computation at $\alpha = 0^\circ$ was performed only for non-cavitating conditions to evaluate numerical accuracy. Figure 4.3.5.a) shows the pressure distribution (pressure coefficient) on the foil surface. Since the NACA0015 hydrofoil has a symmetric shape, pressure distributions on the suction and the pressure side are equal.

Kubota observed a little numerical difference near the trailing edge (not plotted here) due to the unsteadiness of the flow field. This did not result in the present calculation, which constitutes a better approach to the experimental findings. As shown in figure 4.3.4., the pressure field is perfectly symmetric.

The computed result for the pressure coefficient at $\alpha = 8^\circ$ agrees also very well with the experiment and is similar to the result obtained by Kubota. Kubota found for $\alpha = 0^\circ$ at the trailing edge slightly different pressure coefficient distribution for the two sides of the hydrofoil, which is physically not correct because of the symmetric flow conditions. The present simulation also shows better results for this purpose. Figure 4.3.5.b) shows the time-averaged pressure distribution on the foil surface and trailing edge at $\alpha = 8^\circ$. The result agrees fairly well with the experimental data, especially at the trailing edge of the hydrofoil, where the computed $c_p$ by Kubota was almost 0.3 higher than the experimental one.

The lift coefficient $c_L$ computed with the present simulation was about 0.780 ($c_L$, experiment = 0.946), while the lift coefficient predicted by Kubota was only about 58% of that obtained by
experiment \((c_{L, \text{Kubota}} = 0.5505)\). As a reference, Berntsen (2001) obtained \(c_{L, \text{Berntsen}} = 0.740\) [11]. The upper curve describes the pressure coefficient on the suction side of the hydrofoil, where the curve drops far into negative values, whereas the lower curve corresponding to the plot for the pressure side indicates that the local pressure is always bigger than the defined reference pressure. The negative \(c_p\) values on the suction side near the leading edge of the hydrofoil points out the location of the cavitation inception. Fig. 4.3.6 shows a plot of the absolute pressure around the hydrofoil. The negative pressure is physically impossible and must be understood as the inception of cavitation. In the wake of the hydrofoil, the formation of unsteady vortices can be observed, although their frequency and circulation characteristics do not seem to be realistic.

This is highlighted by a comparison between the plot of vorticity magnitude of the present computation and similar experimental data or numerical results, e.g. fig 4.3.1.a) obtained using LES. The Strouhal number based on the maximum foil thickness as reference length was set to 0.2 as suggested by Kubota, 1992. The time step used was varied between 1.0e-04 and 1.0e-06. This gives 50-50000 time steps per cycle. More than \(10^6\) iterations have been performed in isolated cases to obtain lift convergence. First, a very tapered shaped trailing edge was suspected to be a possible cause of the steady character of the flow in the wake.

A second mesh has been generated for verification, consisting of a hydrofoil having a well rounded trailing edge. No changes within the velocity or within the pressure field have been noticed. This second mesh has been further used for the simulation of the case of Berntsen, Kjeldsen and Arndt, 2001, mentioned before, and represented in figure 4.3.10.
At this point, as cavitation conditions were manifest, the same case was tried to be solved activating the cavitation model in Fluent. Firstly no adequate results have been obtained. All simulations have been done using the Spalart-Allmaras turbulence model before starting the computation using the multiphase model in order to obtain better initial values. The value of y+ was for all cases smaller than 4, so that enhanced wall treatment could be applied while using the k-ε turbulence model. For the same cavitation number, different flows have been calculated varying those parameters which mostly influence the cavitation phenomenon or facilitates its computation: operating pressure, Reynolds number, time step, discretisation factors, angle of attack, etc.

The discretisation factors have been chosen very small at the beginning of each computation. The pressure correction parameter in the Fluent AMG solver has been set up to 0.4 as recommended for “complex” cases. Second order schemes have been changed to first order schemes in isolated cases to test stability. For all simulations, the double precision mode has been used. Changing the angle of attack to \( \alpha = 20^\circ \) did not offer better results. As generally observed, the cases were solved very well as long as the cavitation model was not activated. Also in the case \( \alpha \) is set to \( 20^\circ \), the static pressure plot (similar to figure 4.3.6. for \( \alpha = 8^\circ \)) shows evidence of the cavitation phenomena at the leading edge of the NACA0015 hydrofoil. The activation of the cavitation model led to divergence/ no clear convergence of the solver.

To assure that our procedure was adequate, a tutorial case of Fluent has been reproduced. This 2nd case computes also a symmetric hydrofoil at \( 8^\circ \) angle of attack at the same Reynolds and the same cavitation number as in the case of Kubota. The k-ε RNG turbulent model is suggested here, as well as a vapour pressure of 80000 Pa. These parameters for

Fig. 4.3.6. Contours of absolute static pressure around the NACA 0015 hydrofoil, \( \alpha = 8^\circ \), \( Re = 3 \times 10^5 \).
the case of Kubota were set and the Reynolds number was increased to $6.0 \times 10^5$. To obtain the same cavitation number as Kubota ($\sigma = 1.2$), the operating pressure was set equal to the ambient pressure. The case surprisingly converged clearly, showing characteristics of sheet cavitation.

However, no oscillations of the cavity have been detected showing evidence of the cloud cavitation that should be caused by the sheet cavity due to re-entrant jet.

The contours of the vapour volume fraction are shown in figure 4.3.7., as well as plotted versus chord length for the suction side (blue curve) and the pressure side (red curve). A cavity length of about $3/5$ chord length can be deduced from the graph. The vector plot of the

![Fig. 4.3.7. Contours of vapour volume fraction around the NACA 0015 hydrofoil and cavity length, $\sigma = 1.2$, $\alpha = 8^\circ$, $Re = 6 \times 10^5$.](image1)

![Fig. 4.3.8. Vector plot of the velocity magnitude of the NACA 0015 hydrofoil, $\alpha = 8^\circ$; a) leading edge, b) trailing edge.](image2)
velocity magnitude at the leading edge and the trailing edge of the hydrofoil are plotted in figure. 4.3.8. At the rear of the hydrofoil, a detachment of the flow building a vortex is to be observed. The detachment occurs just behind the cavity, at 2/3 chord length. The circulation of the vortex is correctly positive and represents the so called re-entrant jet.

Finally, the cavity length, l, is normalized with respect to chord length, c, and plotted versus $\sigma/2\alpha$. In figure 4.3.9, the numerical results found for $Re = 6.0e+05$ are compared to experimental and numerical results by Berntsen et al., 2001. For the case of Kubota, $\sigma/2\alpha$ is 4.3. The result fits fairly well with the experimental data, but is not as good as the results obtained by Berntsen et al., 2001, for similar flow conditions. In addition, Berntsen was modelling with Fluent 5, and thus did not benefit from the physically more correct model used here, which predicts bubble collapse.

![Fig. 4.3.9. Cavity length plotted as a function of $\sigma/2\alpha$.](image)

The same case has been then reproduced following the same steps and changing only the Reynolds number to $Re = 3 \times 10^5$. No clear convergence has been obtained. A detailed tabular listing of all the simulations done in the present work concerning the reproduction of the case of Kubota can be found in the annexes A.9-A.11.

As it can be seen in the graphs of the annexes, the cases solved using a Reynolds number of $3.0e+05$ do not converge clearly. The density of water-vapour has been set back to the values suggested by Kubota [$\rho(vap) = 0.02558 \text{ kg/m}^3$, $\eta(vap) = 9.0e-06 \text{ kg/(ms)}$], while they have been slightly higher for the case where $Re = 6.0e+05$. In both cases, the volume fraction of non-condensable gas has been set for all cases to $1.5e-05$. However, following Berntsen,
increasing the nuclei density in the numerical model may increase the dynamics of the system, thus adding computational instability. In his work, he increased the nuclei density to a number 20 times higher than the value suggested by Kubota, 1992. In future works, it is suggested to vary this parameter in order to find more accurate results, although it is not clear how far water impurities affecting the cavitating flow field are taken into account in the present cavitation model (see also e.g. Kawakami, 2004 [20]). A further suggestion is also a further mesh refinement. Some authors having researched on similar cases report that the computations can be started with any initial condition, so that it is not believed that varying parameters which affect these settings would yield better solutions.

The vapour pressure should neither be an error source as long as the cavitation number is kept constant by changing respectively the inlet / operating pressure. Reproducing the case of Berntsen et al., 2001 all acknowledgments from the presently elaborated case have been taken into account, and other parameters have been changed to eliminate suspicions. This case is presented in the following.

4.3.2. Simulation based on the case of Berntsen et al., 2001

In order to have more reference, the case of Berntsen et al. has been simulated. The mentioned work clearly defines almost all flow parameters, as well as geometry set-up and computational configuration, so that we could identify possible error sources more easily. The simulation by Berntsen, et al. was done with Fluent 5.

Some changes have been operated, where the most important was the definition of the angle of attack, which no further was given by the corresponding velocity vector components, as it did in our previous simulations (Kubota, 1992), but with a geometrically modified chord orientation against the x-axis (horizontal).

4.3.2.1. Computational domain and boundary conditions

The hydrofoil is also a NACA0015. The grid is a hybrid C-grid type with a total of about 22000 cells. The smallest cell size in the direction from the wall is 0.01 which corresponds perfectly to the source case. The results in the first grid point are at $y^+ < 5$, which implies the use of the $k$-$\varepsilon$ model with enhanced wall treatment. The inlet is set to 12m/s with constant profile, the velocity being computed from a Reynolds number equal to $1.2e+06$ based on chord length. The unsteady SIMPLE algorithm was used, as well as second order discretisation schemes. Internally, all numbers are stored and calculated with double precision. The time step used was 0.0001 and the water vapour pressure was set to 2400Pa. The same effect could also be
achieved by having a constant reference pressure and adjusting the vapour pressure. The cavitation number was adjusted by changing the reference pressure, $P_{\text{ref}}$, in the solver, as it was done for all previous simulations. The composite parameter was $\sigma/2\alpha = 5$ at 8 degrees AoA. A close-up of the grid and the computational domain including boundary conditions are drafted in figure 4.3.10.

### 4.3.2.2. Simulation results and discussion

An instantaneous static pressure contour plot and velocity vector plot at the same instant in time as obtained by Berntsen et al. is shown in figure 4.3.11.a. In the simulation by Berntsen et al., we can observe (fig 4.3.11.b), the cavity length is identified from both, the static pressure plot and the velocity vector plot, this one showing clearly the re-entrant jet and the closure of the attached cavity.

![Grid close-up surrounding the 2D hydrofoil](image1)

![Outline of the computational domain with boundary condition type](image2)

**Fig. 4.3.10.** a) Grid close-up surrounding the 2D hydrofoil, and b) outline of the computational domain with boundary condition type

The very strong vortex structures downstream of the cavity give very low pressure regions that effectively transport the bubbles shed from the foil. The authors can well predict the flow field, especially the cavity length and the characteristics of cloud cavitation. The dynamics of the cavitating system is still not very accurately computed.

Unfortunately we could not obtain comparable results. As for the case of Kubota, the cases showed good results as long as the cavitation model was not used. The Appendix A.12 shows detailed information about the methodology used for simulating this case as well as set-up parameters. While using the cavitation model when the local static pressure was lower than the defined vaporisation pressure, divergence of the numerical simulation has always been observed.
This case shows clearly a probable deficiency of the cavitation model or its implementation in Fluent 6.1. The results obtained by Berntsen et al. may not be very realistic, but at least according to the authors, the computations converge. Concerning their unrealistic results, the authors argue, that the cavitation model “suffers from several simplifications”, where the most critical would be the fact that both phases should be treated as compressible, condition which is not fulfilled for the cavitation model implemented in Fluent.

Fig. 4.3.11. Results by Berntsen, Kjeldsen and Arndt, 2001:
a) constant pressure contour plot and
b) corresponding velocity plot; $\alpha / 2 = 5$ at 8 degrees AoA.

At this point, a theoretical research on cavitation models should be pursued. The implementation of Singhal’s cavitation model should be revised. The new version of Fluent has been tested with the cases of Nurick, and confirmed our doubts about the reliability of the numerical problems on Fluent 6.1. The much higher numerical stability of Fluent 6.2 treating cavitating flows encourages the application of the cases treating a cavitating hydrofoil discussed in this chapter. This will be part of future work considerations.
Conclusions

Within this work unsteady turbulent cavitating flows have been simulated. Due to the complex character of such flow fields, a preliminary study of CFD and its possibilities was indispensable. Also, an important bibliographic work has been done concerning previous theoretical studies of cavitating flows and the different existing cavitation models.

Several cases of steady/unsteady turbulent or/and cavitating flows with the commercial CFD code (Fluent v6.1) have been studied. The case of an unsteady and turbulent, non-cavitating flow around a 2D circular cylinder was simulated as a first step using different turbulence models at Reynolds numbers around the critical drag-crisis region. A mesh refinement control helped choosing an adequate mesh for the computations. Compared with experimental data, the results were quite divergent, but similar to previous numerical researches of other authors (e.g. Cox, Zdravkovich). The non-accurate prediction of the transition of the viscous underlayer from laminar to turbulent in the drag crisis region is mainly due to 3D effects. Also, a deeper knowledge concerning the selection of a suitable step size and its influence on the numerical results was obtained.

Further, the cavitation phenomenon has been studied in several applications. First, the full cavitation model implemented in Fluent has been tested comparing findings with corresponding experimental data. The first case was a steady, cavitating flow through a sharp-edged orifice by Nurick, 1976. A previous mesh sensitivity study has been successfully done. The inception of cavitation has been correctly predicted at $\sigma = 1.76$.

Finally, the unsteady turbulent flow around a 2D NACA 0015 hydrofoil has been simulated using the cavitation model. This work was based on the publications by Kubota and Berntsen et al., 2001 Results revealed that depending on the case, the cavitation model offers useful results, but only in a qualitative way. Accurate fittings with experiments are obtained only in few cases. A systematic and documented simulation process argues against any user's error. A software problem was mainly suspected, but also some problematic presumptions within the cavitation model.

Numerical simulations of cavitating flows are very challenging since localized large variations of density are present within a predominantly incompressible liquid medium. In its most general form, the gas pockets causing these density variations can be highly compressible with large temperature variations. However, even in the simpler case where the gas bubble may be treated as incompressible, large density changes are still present at the liquid/gas interface.
In summary, an interesting step for the understanding of the cavitation phenomena and its coupling with vortex shedding has been done. The acquired physical background and the experience in simulating complex flows builds an important basis for the accomplishment of this task. Nevertheless, some further progress has to be done. In order to hopefully obtain this progress, the steps described before have to be followed.

**Future work**

Many of the results presented here must be further explored. Firstly, it has to be determined if the cavitation model implementation in Fluent 6.1 is not a possible error source, because some inconsistencies in the obtained results point out that suspicion. A new version of the software is presently being tested, where several changes have been done regarding the cavitation model. The case of Nurick is being used for validation. Very promising first results have been obtained at the simulation of these cases. Different meshes have been used, as well as different turbulent models. A very important progress has been done concerning the numerical stability of the software. As a further procedure, the latest version of Fluent has to be tested using the more complex cases of Kubota and Berntsen et al., treating of an unsteady turbulent cavitating flow around a hydrofoil. Also, other numerical codes will be tried out (e.g. CFX-5.7), using the cases of Kubota and Berntsen et al. for simulation.

Since at the present time RANS models are the unique real possibility in complex and turbulent flow modelling, depending on the obtained results, a question of major importance remains open: “Are RANS models the adequate tool to be used for the study of such complex technical problems?” Probably and in most cases related to the cavitation phenomena, at least big vortex structures have to be mathematically computed. The work of many authors (e.g. the aforementioned works of Arndt, Kunz, Qin, Song) yield better results using the LES method for the simulation of turbulent cavitating flows, but LES cannot be applied in more complex cases due to high computer requirements.

Finally, after having obtained a deeper knowledge of the selected cavitation model, the major goal which consists in the prediction of erosive effects of cavitation in components of fluid flow machines has to be followed. The first step to reach this goal is the code validation in 3-D airfoils using experiments from Escaler, 2001.
Acknowledgements

The achievement of this work has been done thanks to the bilateral work between the Departament de Mecànica de Fluids of the Universidad Politècnica de Catalunya, Bartcelona, Spain and the Institut für Strömungsmechanik und Hydraulische Strömungsmaschinen of the University of Stuttgart, Germany. The academic exchange programme Erasmus was the organisation which has made this interchange possible.

First and foremost, I would like to thank my supervisor, Mr. Miguel Coussirat for his invaluable direction and insight, his patience, and genuine care and interest in this work and my progress. I would also like to thank my professors and advisors in Germany, Dr. Albert Ruprecht and Prof. Eberhard Göde who offered me the possibility to achieve my master thesis in Barcelona and gave me a strong support for my work, helpful advice and encouraging directives.

I am also very grateful to Prof. Eduard Egusquiza who made it possible for me to accomplish my work at the Departament de Mecànica de Fluids in Barcelona.

I would also like to express my gratitude to all members of the Departament de Mecànica de Fluids as well as to the staff of the Institut für Strömungsmechanik und Hydraulische Strömungsmaschinen who offered me occasional support and guidance.

Lastly, I would like to thank my parents for their dedication and continual support; they very much encouraged and helped me in the completion of this work.
Annexes

A.1 Mesh refinement – drag coefficient, $k$-$\omega$ SST.

A.2 Mesh refinement – Strouhal number, $k$-$\omega$ SST.

A.3 Mesh refinement – drag coefficient, Spalart-Allmaras.

A.4 Mesh refinement – Strouhal number, Spalart-Allmaras.

A.5 Drag coefficient for a circular cylinder at different Reynolds numbers (50000 nodes grid = mesh 01), experimental data and numerical results.

A.6 Strouhal number for a circular cylinder at different Reynolds numbers (50000 nodes grid = mesh 01), experimental data and numerical results.

A.7 Drag coefficient for a circular cylinder, experimental data and numerical results, (J.S.Cox et al., 1997).

A.8 Strouhal number for a circular cylinder, experimental data and numerical results, (J.S.Cox et al., 1997).

A.9 Data table, systematic numerical research, case of Kubota.

A.10 Data table, systematic numerical research, case of Kubota and adaptation from the Fluent tutorial case.

A.11 Data table, systematic numerical research, case of Kubota at lower Reynolds number.

A.12 Data table, systematic numerical research, case of Berntsen et al.
Mesh refinement - Strouhal number - k-ω SST
Drag coefficient for a circular cylinder, J.S. Cox et al. (1997)
<table>
<thead>
<tr>
<th>CASE</th>
<th>SOLVER</th>
<th>CAV. MODEL</th>
<th>TURB. MODEL</th>
<th>INLET</th>
<th>REYNOLDS</th>
<th>OUTLET</th>
<th>OP. PRESSURE</th>
<th>SIGMA</th>
</tr>
</thead>
<tbody>
<tr>
<td>case Kubata (Re =6x05)</td>
<td>A1</td>
<td>UNSTEADY</td>
<td>OFF</td>
<td>SA strain vorticity based prod</td>
<td>v = 6 m/s</td>
<td>Re =6x05</td>
<td>p(gauge) = 24177.3 pa</td>
<td>p(op) = 0</td>
</tr>
</tbody>
</table>

**=> k-eps enhanced wall treat. With int. sol from A1 (10000 more iter.)**

<table>
<thead>
<tr>
<th>CASE</th>
<th>PHASES</th>
<th>VAP. PRESS.</th>
<th>Y+</th>
<th>UNDER-RELAX.</th>
<th>DISCRETIZATION</th>
<th>TIME STEP</th>
<th>CONV.</th>
<th>CAV.</th>
</tr>
</thead>
<tbody>
<tr>
<td>case Kubata (Re =6x05)</td>
<td>A1</td>
<td>-</td>
<td>[2400]</td>
<td>y+&lt;1.0</td>
<td>SIMPLE, 2nd order</td>
<td>1.00E-05</td>
<td>YES</td>
<td>potat &lt; 0</td>
</tr>
</tbody>
</table>

**=> k-eps enhanced wall treat. With int. sol from A1 (10000 more iter.)**

<table>
<thead>
<tr>
<th>CASE</th>
<th>PHASES</th>
<th>VAP. PRESS.</th>
<th>Y+</th>
<th>UNDER-RELAX.</th>
<th>DISCRETIZATION</th>
<th>TIME STEP</th>
<th>CONV.</th>
<th>CAV.</th>
</tr>
</thead>
<tbody>
<tr>
<td>case Kubata (Re =6x05)</td>
<td>A1</td>
<td>-</td>
<td>[2400]</td>
<td>p(0.2); mole(1); mole(2); mol(visc(0));1.4</td>
<td>SIMPLE, 2nd order</td>
<td>1.00E-05</td>
<td>YES</td>
<td></td>
</tr>
</tbody>
</table>

**=> started from A2 with cavt.**

<table>
<thead>
<tr>
<th>CASE</th>
<th>PHASES</th>
<th>VAP. PRESS.</th>
<th>Y+</th>
<th>UNDER-RELAX.</th>
<th>DISCRETIZATION</th>
<th>TIME STEP</th>
<th>CONV.</th>
<th>CAV.</th>
</tr>
</thead>
<tbody>
<tr>
<td>case Kubata (Re =6x05)</td>
<td>A1</td>
<td>-</td>
<td>[2400]</td>
<td>0</td>
<td>SIMPLE, 2nd order</td>
<td>1.00E-05</td>
<td>NO CLEAR</td>
<td></td>
</tr>
</tbody>
</table>

**=> increase time step (from A2)**

<table>
<thead>
<tr>
<th>CASE</th>
<th>PHASES</th>
<th>VAP. PRESS.</th>
<th>Y+</th>
<th>UNDER-RELAX.</th>
<th>DISCRETIZATION</th>
<th>TIME STEP</th>
<th>CONV.</th>
<th>CAV.</th>
</tr>
</thead>
<tbody>
<tr>
<td>case Kubata (Re =6x05)</td>
<td>A1</td>
<td>-</td>
<td>20000</td>
<td>-</td>
<td>SIMPLE, 2nd order</td>
<td>1.00E-05</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**=> set p(op) = 0 and p(op) = 24177 pa (from A2)**

<table>
<thead>
<tr>
<th>CASE</th>
<th>PHASES</th>
<th>VAP. PRESS.</th>
<th>Y+</th>
<th>UNDER-RELAX.</th>
<th>DISCRETIZATION</th>
<th>TIME STEP</th>
<th>CONV.</th>
<th>CAV.</th>
</tr>
</thead>
<tbody>
<tr>
<td>case Kubata (Re =6x05)</td>
<td>A1</td>
<td>-</td>
<td>20000</td>
<td>-</td>
<td>SIMPLE, 2nd order</td>
<td>1.00E-05</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**=> change vapor pressure (same cavt. number) (from A2)**

<table>
<thead>
<tr>
<th>CASE</th>
<th>PHASES</th>
<th>VAP. PRESS.</th>
<th>Y+</th>
<th>UNDER-RELAX.</th>
<th>DISCRETIZATION</th>
<th>TIME STEP</th>
<th>CONV.</th>
<th>CAV.</th>
</tr>
</thead>
<tbody>
<tr>
<td>case Kubata (Re =6x05)</td>
<td>A1</td>
<td>-</td>
<td>20000</td>
<td>-</td>
<td>SIMPLE, 2nd order</td>
<td>1.00E-05</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**=> case A2' started from A2**

<table>
<thead>
<tr>
<th>CASE</th>
<th>PHASES</th>
<th>VAP. PRESS.</th>
<th>Y+</th>
<th>UNDER-RELAX.</th>
<th>DISCRETIZATION</th>
<th>TIME STEP</th>
<th>CONV.</th>
<th>CAV.</th>
</tr>
</thead>
<tbody>
<tr>
<td>case Kubata (Re =6x05)</td>
<td>A1</td>
<td>-</td>
<td>20000</td>
<td>-</td>
<td>SIMPLE, 2nd order</td>
<td>1.00E-05</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**=> case A2' started from A2**

<table>
<thead>
<tr>
<th>CASE</th>
<th>PHASES</th>
<th>VAP. PRESS.</th>
<th>Y+</th>
<th>UNDER-RELAX.</th>
<th>DISCRETIZATION</th>
<th>TIME STEP</th>
<th>CONV.</th>
<th>CAV.</th>
</tr>
</thead>
<tbody>
<tr>
<td>case Kubata (Re =6x05)</td>
<td>A1</td>
<td>-</td>
<td>20000</td>
<td>-</td>
<td>SIMPLE, 2nd order</td>
<td>1.00E-05</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
### Numerical Simulation of Unsteady Turbulent Cavitating Flows

<table>
<thead>
<tr>
<th>CASE</th>
<th>SOLVER</th>
<th>CAV. MODEL</th>
<th>TURB. MODEL</th>
<th>INLET</th>
<th>REYNOLDS</th>
<th>OUTLET</th>
<th>OP. PRESSURE</th>
<th>SIGMA</th>
</tr>
</thead>
<tbody>
<tr>
<td>B1</td>
<td>STEADY</td>
<td>OFF</td>
<td>k-epsilon RNG non-equ. wall fact.</td>
<td>$\nu = 6 \times 10^{-3}$</td>
<td>Re = 6x10^5</td>
<td>$\rho_{(\text{gauge})} = 0$</td>
<td>$p_{(\text{gauge})} = 101325$</td>
<td>$\sigma = 1.19$</td>
</tr>
<tr>
<td>B2</td>
<td>UNSTEADY</td>
<td>ON</td>
<td>$k$-epsilon RNG non-equ. wall fact.</td>
<td>$\nu = 6 \times 10^{-3}$</td>
<td>Re = 6x10^5</td>
<td>$\rho_{(\text{gauge})} = 0$</td>
<td>$p_{(\text{gauge})} = 101325$</td>
<td>$\sigma = 1.19$</td>
</tr>
<tr>
<td>B3</td>
<td>-</td>
<td>-</td>
<td>$k$-epsilon RNG non-equ. wall fact.</td>
<td>$\nu = 6 \times 10^{-3}$</td>
<td>Re = 6x10^5</td>
<td>$\rho_{(\text{gauge})} = 0$</td>
<td>$p_{(\text{gauge})} = 101325$</td>
<td>$\sigma = 1.19$</td>
</tr>
<tr>
<td>B4</td>
<td>STEADY</td>
<td>OFF</td>
<td>k-epsilon RNG non-equ. wall fact.</td>
<td>$\nu = 6 \times 10^{-3}$</td>
<td>Re = 6x10^5</td>
<td>$\rho_{(\text{gauge})} = 0$</td>
<td>$p_{(\text{gauge})} = 101325$</td>
<td>$\sigma = 1.19$</td>
</tr>
<tr>
<td>B5</td>
<td>UNSTEADY</td>
<td>ON</td>
<td>$k$-epsilon RNG non-equ. wall fact.</td>
<td>$\nu = 6 \times 10^{-3}$</td>
<td>Re = 6x10^5</td>
<td>$\rho_{(\text{gauge})} = 0$</td>
<td>$p_{(\text{gauge})} = 101325$</td>
<td>$\sigma = 1.19$</td>
</tr>
<tr>
<td>B6</td>
<td>-</td>
<td>-</td>
<td>$k$-epsilon RNG non-equ. wall fact.</td>
<td>$\nu = 6 \times 10^{-3}$</td>
<td>Re = 6x10^5</td>
<td>$\rho_{(\text{gauge})} = 0$</td>
<td>$p_{(\text{gauge})} = 101325$</td>
<td>$\sigma = 1.19$</td>
</tr>
<tr>
<td>B7</td>
<td>-</td>
<td>-</td>
<td>$k$-epsilon RNG non-equ. wall fact.</td>
<td>$\nu = 6 \times 10^{-3}$</td>
<td>Re = 6x10^5</td>
<td>$\rho_{(\text{gauge})} = 0$</td>
<td>$p_{(\text{gauge})} = 101325$</td>
<td>$\sigma = 1.19$</td>
</tr>
</tbody>
</table>

### Parameters: case Kubota (RE = 6,600+65)
- **CASE**: B1 to B7
- **SOLVER**: STEADY, UNSTEADY
- **CAV. MODEL**: OFF
- **TURB. MODEL**: k-epsilon RNG non-equ. wall fact.
- **INLET**: $\nu = 6 \times 10^{-3}$
- **REYNOLDS**: Re = 6x10^5
- **OUTLET**: $\rho_{(\text{gauge})} = 0$
- **OP. PRESSURE**: $p_{(\text{gauge})} = 101325$
- **SIGMA**: $\sigma = 1.19$

**CASE A.10**

<table>
<thead>
<tr>
<th>CASE</th>
<th>PHASES</th>
<th>VAP. PRESS.</th>
<th>$y+$</th>
<th>UNDER-RELAX</th>
<th>DISCRETIZATION</th>
<th>TIME STEP</th>
<th>CONV.</th>
<th>CAV.</th>
</tr>
</thead>
<tbody>
<tr>
<td>B1</td>
<td>0</td>
<td>100000</td>
<td>$y+$</td>
<td>3</td>
<td>p($\nu$) = 0, $\rho_{\text{turb}}(\text{fg})$, $\text{c}<em>{\text{turb}}(\text{fg})$, $\text{m}</em>{\text{turb}}(\text{fg})$, $\text{c}<em>{\text{turb}}(\text{fg})$, $\text{m}</em>{\text{turb}}(\text{fg})$</td>
<td>1.00E-04</td>
<td>YES</td>
<td>pstat &lt; 0</td>
</tr>
<tr>
<td>B2</td>
<td>0</td>
<td>100000</td>
<td>$y+$</td>
<td>3</td>
<td>p($\nu$) = 0, $\rho_{\text{turb}}(\text{fg})$, $\text{c}<em>{\text{turb}}(\text{fg})$, $\text{m}</em>{\text{turb}}(\text{fg})$, $\text{c}<em>{\text{turb}}(\text{fg})$, $\text{m}</em>{\text{turb}}(\text{fg})$</td>
<td>1.00E-04</td>
<td>YES</td>
<td>pstat &lt; 0</td>
</tr>
<tr>
<td>B3</td>
<td>0</td>
<td>100000</td>
<td>$y+$</td>
<td>3</td>
<td>p($\nu$) = 0, $\rho_{\text{turb}}(\text{fg})$, $\text{c}<em>{\text{turb}}(\text{fg})$, $\text{m}</em>{\text{turb}}(\text{fg})$, $\text{c}<em>{\text{turb}}(\text{fg})$, $\text{m}</em>{\text{turb}}(\text{fg})$</td>
<td>1.00E-04</td>
<td>YES</td>
<td>pstat &lt; 0</td>
</tr>
<tr>
<td>B4</td>
<td>0</td>
<td>100000</td>
<td>$y+$</td>
<td>3</td>
<td>p($\nu$) = 0, $\rho_{\text{turb}}(\text{fg})$, $\text{c}<em>{\text{turb}}(\text{fg})$, $\text{m}</em>{\text{turb}}(\text{fg})$, $\text{c}<em>{\text{turb}}(\text{fg})$, $\text{m}</em>{\text{turb}}(\text{fg})$</td>
<td>1.00E-04</td>
<td>YES</td>
<td>pstat &lt; 0</td>
</tr>
<tr>
<td>B5</td>
<td>0</td>
<td>100000</td>
<td>$y+$</td>
<td>3</td>
<td>p($\nu$) = 0, $\rho_{\text{turb}}(\text{fg})$, $\text{c}<em>{\text{turb}}(\text{fg})$, $\text{m}</em>{\text{turb}}(\text{fg})$, $\text{c}<em>{\text{turb}}(\text{fg})$, $\text{m}</em>{\text{turb}}(\text{fg})$</td>
<td>1.00E-04</td>
<td>YES</td>
<td>pstat &lt; 0</td>
</tr>
<tr>
<td>B6</td>
<td>0</td>
<td>100000</td>
<td>$y+$</td>
<td>3</td>
<td>p($\nu$) = 0, $\rho_{\text{turb}}(\text{fg})$, $\text{c}<em>{\text{turb}}(\text{fg})$, $\text{m}</em>{\text{turb}}(\text{fg})$, $\text{c}<em>{\text{turb}}(\text{fg})$, $\text{m}</em>{\text{turb}}(\text{fg})$</td>
<td>1.00E-04</td>
<td>YES</td>
<td>pstat &lt; 0</td>
</tr>
<tr>
<td>B7</td>
<td>0</td>
<td>100000</td>
<td>$y+$</td>
<td>3</td>
<td>p($\nu$) = 0, $\rho_{\text{turb}}(\text{fg})$, $\text{c}<em>{\text{turb}}(\text{fg})$, $\text{m}</em>{\text{turb}}(\text{fg})$, $\text{c}<em>{\text{turb}}(\text{fg})$, $\text{m}</em>{\text{turb}}(\text{fg})$</td>
<td>1.00E-04</td>
<td>YES</td>
<td>pstat &lt; 0</td>
</tr>
<tr>
<td>CASE</td>
<td>SOLVER</td>
<td>CAV. MODEL</td>
<td>TURB. MODEL</td>
<td>INLET</td>
<td>REYNOLDS</td>
<td>OUTLET</td>
<td>OP. PRESSURE</td>
<td>SIGMA</td>
</tr>
<tr>
<td>-------</td>
<td>--------</td>
<td>------------</td>
<td>-------------</td>
<td>-------</td>
<td>-----------</td>
<td>--------</td>
<td>--------------</td>
<td>-------</td>
</tr>
<tr>
<td>B10</td>
<td>STEADY</td>
<td>OFF</td>
<td>k-eps RNG</td>
<td>3m/s</td>
<td>Re = 3e+05</td>
<td></td>
<td>p(peak) = 0</td>
<td>sigma = 1.2</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>enh. wall</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>treat.*</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>press. grad.</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>effects</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>turb. intensity = 5%</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>turb. visc. ratio = 2</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

- => started from B10 with unsteady and cav. Model
- => continue from B11 with 2nd order schemes
- => continue from B12 increasing the number of max. iterat. per time step
- => continue from B13 with some number of max. iterat. But again 1st order schemes

<table>
<thead>
<tr>
<th>CASE</th>
<th>VAP. PRESS.</th>
<th>PHASES</th>
<th>Y+</th>
<th>UNDER-RELAX</th>
<th>DISCRETIZATION</th>
<th>TIME STEP</th>
<th>CONV.</th>
<th>CAV.</th>
</tr>
</thead>
</table>
| B12   | rho=0.002568 kg/m³
|       | e=2.4e-06 (kg/m³) | [00000] | y+<3 | p(0.3) rho(1), R(f) |
|       |               |          |     | SIMPLE, 1st order | p(standard) | - | YES | sigma = 1.2 |

- => started from B10 with unsteady and cav. Model
- => continue from B11 with 2nd order schemes
- => continue from B12 increasing the number of max. iterat. per time step
- => continue from B13 with some number of max. iterat. But again 1st order schemes

<table>
<thead>
<tr>
<th>CASE</th>
<th>VAP. PRESS.</th>
<th>PHASES</th>
<th>Y+</th>
<th>UNDER-RELAX</th>
<th>DISCRETIZATION</th>
<th>TIME STEP</th>
<th>CONV.</th>
<th>CAV.</th>
</tr>
</thead>
</table>
| B11   | rho=0.002568 kg/m³
|       | e=2.4e-06 (kg/m³) | 80000 | 0 | p(0.3) rho(1), R(f) |
|       |               |          |     | SIMPLE, 1st order | p(standard), vol. fraction | 1.00E-06 not clear | |

- => started from B10 with unsteady and cav. Model
- => continue from B11 with 2nd order schemes
- => continue from B12 increasing the number of max. iterat. per time step
- => continue from B13 with some number of max. iterat. But again 1st order schemes

<table>
<thead>
<tr>
<th>CASE</th>
<th>VAP. PRESS.</th>
<th>PHASES</th>
<th>Y+</th>
<th>UNDER-RELAX</th>
<th>DISCRETIZATION</th>
<th>TIME STEP</th>
<th>CONV.</th>
<th>CAV.</th>
</tr>
</thead>
<tbody>
<tr>
<td>B12</td>
<td>&quot;</td>
<td>&quot;</td>
<td>0</td>
<td>&quot;</td>
<td>SIMPLE, 2nd order</td>
<td>1.00E-06</td>
<td>&quot;</td>
<td>&quot;</td>
</tr>
</tbody>
</table>

- => started from B10 with unsteady and cav. Model
- => continue from B11 with 2nd order schemes
- => continue from B12 increasing the number of max. iterat. per time step
- => continue from B13 with some number of max. iterat. But again 1st order schemes

<table>
<thead>
<tr>
<th>CASE</th>
<th>VAP. PRESS.</th>
<th>PHASES</th>
<th>Y+</th>
<th>UNDER-RELAX</th>
<th>DISCRETIZATION</th>
<th>TIME STEP</th>
<th>CONV.</th>
<th>CAV.</th>
</tr>
</thead>
<tbody>
<tr>
<td>B13</td>
<td>&quot;</td>
<td>&quot;</td>
<td>0</td>
<td>&quot;</td>
<td>SIMPLE, 1st order</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
</tr>
</tbody>
</table>

- => started from B10 with unsteady and cav. Model
- => continue from B11 with 2nd order schemes
- => continue from B12 increasing the number of max. iterat. per time step
- => continue from B13 with some number of max. iterat. But again 1st order schemes

<table>
<thead>
<tr>
<th>CASE</th>
<th>VAP. PRESS.</th>
<th>PHASES</th>
<th>Y+</th>
<th>UNDER-RELAX</th>
<th>DISCRETIZATION</th>
<th>TIME STEP</th>
<th>CONV.</th>
<th>CAV.</th>
</tr>
</thead>
<tbody>
<tr>
<td>B14</td>
<td>&quot;</td>
<td>&quot;</td>
<td>0</td>
<td>&quot;</td>
<td>p(standard), SIMPLE, 1st order</td>
<td>&quot;</td>
<td>&quot;</td>
<td>&quot;</td>
</tr>
</tbody>
</table>

- => started from B10 with unsteady and cav. Model
- => continue from B11 with 2nd order schemes
- => continue from B12 increasing the number of max. iterat. per time step
- => continue from B13 with some number of max. iterat. But again 1st order schemes
<table>
<thead>
<tr>
<th>Case</th>
<th>Solver</th>
<th>CAV Model</th>
<th>Turbulent Model</th>
<th>inlet</th>
<th>Reynolds</th>
<th>Outlet</th>
<th>OP Pressure</th>
<th>Sigma</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case of Bontems et al., simulated from steady solution after 600 sec. (core)</td>
<td>C1</td>
<td>UNSTADY</td>
<td>OFF</td>
<td>BA</td>
<td>$v = 12m/s$</td>
<td>Re=1,2e+9</td>
<td>$p_{inlet} = 91200$ Pa</td>
<td>$p_{outlet} = 0$</td>
</tr>
<tr>
<td>=&gt; C1 with zero enthalpy</td>
<td>C2</td>
<td>-</td>
<td>-</td>
<td>SA intermittency based prof.</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>=&gt; started from C1, 5 st. steps</td>
<td>C3</td>
<td>-</td>
<td>-</td>
<td>SA intermittency based prof.</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>=&gt; started from C3 with 5 steps</td>
<td>C4</td>
<td>-</td>
<td>OR</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>=&gt; with mesh (Mesh 0106), simulated from steady solution after 600 sec. (core)</td>
<td>P1</td>
<td>UNSTADY</td>
<td>OFF</td>
<td>BA</td>
<td>$v = 12m/s$</td>
<td>Re=1,2e+9</td>
<td>$p_{inlet} = 91200$ Pa</td>
<td>$\sigma = 1.4$</td>
</tr>
<tr>
<td>=&gt; P1 with radiation (k-eps model)</td>
<td>P2</td>
<td>UNSTADY</td>
<td>OR</td>
<td>fixed k-eps with Wall treatment and press gain 0.95</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>change op. Pressure to gauge pressure</td>
<td>X1</td>
<td>STEADY</td>
<td>OFF</td>
<td>BA</td>
<td>$v = 12m/s$</td>
<td>-</td>
<td>$p_{gaugel} = 0$</td>
<td>$p_{gauge} = 101200$</td>
</tr>
<tr>
<td>=&gt; started from 01, unsteady</td>
<td>X2</td>
<td>UNSTADY</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>=&gt; started from 02, unsteady</td>
<td>X3</td>
<td>OR</td>
<td>k-eps, SST</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
</tbody>
</table>

| Case | Pres | GFP, Pres. | $\gamma$ | UNDER RELAX | DISCRETIZATION | TIME STEP | CORR | CAV |
|------|------|-----------|-------|------------|----------------|------------|------|
| Case of Bontems et al., simulated from steady solution after 600 sec. (core) | C1 | - | [2400] | 0.5 | SIMPLE, 2nd order | 1,000 Sa | undefined | 0 |
| => C1 with zero enthalpy | C2 | - | [2400] | 0.5 | SIMPLE, 2nd order | 1,000 Sa | undefined | 0 |
| => started from C1, 5 st. steps | C3 | - | [2400] | 0.5 | SIMPLE, 2nd order | 1,000 Sa | undefined | 0 |
| => started from C3 with 5 steps | C4 | - | [2400] | 0.5 | SIMPLE, 2nd order | 1,000 Sa | undefined | 0 |
| => with mesh (Mesh 0106), simulated from steady solution after 600 sec. (core) | P1 | - | [2400] | 0.5 | SIMPLE, 2nd order | 1,000 Sa | undefined | 0 |
| => P1 with radiation (k-eps model) | P2 | - | [2400] | 0.5 | SIMPLE, 2nd order | 1,000 Sa | undefined | 0 |
| change op. Pressure to gauge pressure | X1 | - | [2400] | 0.5 | SIMPLE, 2nd order | 1,000 Sa | undefined | 0 |
| => started from 01, unsteady | X2 | - | [2400] | 0.5 | SIMPLE, 2nd order | 1,000 Sa | undefined | 0 |
| => started from 02, unsteady | X3 | - | [2400] | 0.5 | SIMPLE, 2nd order | 1,000 Sa | undefined | 0 |

A.12
Bibliography


Complementary works


