Temperature distribution of shallow water FLNG cooling water outfalls

Remco op het Veld

Delft University of Technology
National University of Singapore
TEMPERATURE DISTRIBUTION OF SHALLOW WATER FLNG COOLING WATER OUTFALLS

Remco (H.T.H.) op het Veld

Thesis committee: Prof. dr. J. D. Pietrzak, TU Delft
                  Dr. ir. M. Zijlema, TU Delft
                  Ir. T. J. Zitman, TU Delft
                  Dr. ir. J. Kreeft, Shell
                  Ir. M. Klabbers, Shell
                  Dr. S. K. Ooi, NUS

An electronic version of this thesis is available at http://repository.tudelft.nl/.

Cover image: “Shell prelude FLNG “, http://www.shell.com, Photographic Services, Shell International Ltd.
Acknowledgements

This MSc. thesis is the final result of the double degree Master of Science program at the faculty of Civil Engineering at the Delft University of Technology and the National University of Singapore. This thesis has been performed under the guidance of the section of Environmental Fluid Mechanics of the TU Delft in cooperation with the Civil Structures and Offshore Engineering group and the Fluid Flow and Reactor Engineering group of Shell.

I would like to thank my daily supervisors at Shell for their commitment and interest in progress and results. I have experienced a nice atmosphere, with many interesting talks during coffee, lunch and dinner. Martijn Klabbers and Arjan Maijenburg, thank you for the nice collaboration, your time and very constructive feedback. Jasper Kreeft, I really appreciate your effort and time to introduce me to OpenFOAM and your insightful thoughts and comments.

In addition, I would like to thank my thesis committee, with special attention to prof. dr. Julie Pietrzak, for the useful feedback and constructive meetings. Further, Tjerk Zitman and Marcel Zijlema, who were able to address different points of view to the thesis results and context, which certainly improved my thesis report.

Furthermore, I am grateful to my friends and other graduation colleagues at Shell to make the last eight month enjoyable. Last, but certainly not least, special thanks to my parents, my sister and Britt for their encouragements, interest and unconditional support.

Remco op het Veld
December 10, 2014
Abstract

FLNG cooling water outfalls can be characterized as high momentum buoyant jets, relatively close to the surface. The buoyant jet trajectories, spreading rates and surface distribution very much depend on the jet outflow characteristics. A study with the CFD software OpenFOAM has been carried out into the mixing and transport processes of buoyant jets. It is shown that OpenFOAM can be used to model the three dimensional trajectories of buoyant jets and their far-field buoyant plume distribution.

Two dominant mixing processes are the result of the high initial jet momentum. First, the high momentum jet results in large turbulent jet entrainment rates. Second, the relatively shallow high momentum jets result in large horizontal surface currents. These currents horizontally advect the buoyant plume into the far-field and result in steep vertical velocity gradients, which induce vertical mixing of the buoyant plume.

The momentum length scale, $L_M$ proves to be an important parameter to characterize buoyant jets. The momentum length scale represents a distance along the jet trajectory, where buoyancy effects become dominant over initial jet momentum. For these jet characteristics it is found that the dimensionless surface temperature rise follows a logistic distribution function to the momentum length scale, after the point of surface impingement. The relative surface temperature results become constant for increasing values of the momentum length scale. This is the result of increased mixing by the jet turbulent entrainment and steep vertical velocity gradient.

From the logistic distribution an empirical relation is found which can accurately predict the surface temperature rise as a function of the outfall velocity, outfall diameter, outfall temperature, outfall angle and distance from the jet orifice. The outfall depth appears to have no significant influence on the surface temperature rise for the conditions used in this study.

The empirical equation proves to give reliable results for distances larger than 1.5 times the value of the momentum length scale and jet submergence smaller than 7.5 times the jet diameter. The robustness of the equation is also tested for extreme value outfall scenarios. The equation overestimates the temperature rise for small outflow diameters, combined with high initial jet temperatures. For other considered extreme values, the equation proves to give reliable results.

Moreover, it is also demonstrated that the standard $k - \epsilon$ turbulence closure can be successfully used to model the buoyant jet centerline velocities, jet trajectories, spreading rates and centerline dilution rates of a round turbulent buoyant jet.
Contents

1 Preface ................................................. 1
  1.1 Context ........................................ 1
  1.2 Shallow water FLNG ................................ 1
  1.3 Environmental regulations for discharging cooling water ............. 2

2 Introduction ........................................... 5
  2.1 Literature review ................................ 6
  2.2 Research objective ................................ 9
  2.3 Research methodology ................................ 10
  2.4 Thesis outline ...................................... 11

3 Theoretical background of jets and plumes ................................ 13
  3.1 Introduction ....................................... 13
  3.2 Length scales ...................................... 14
  3.3 Pure jet physics .................................... 15
  3.4 Buoyant jet physics .................................. 18
  3.5 Water depth classification ............................. 21
  3.6 Turbulent entrainment ................................ 22
  3.7 Jet changing parameters ............................... 22

4 Governing equations of fluid dynamics .................................. 23
  4.1 Introduction ....................................... 23
  4.2 Total derivative ..................................... 23
  4.3 Conservation laws of fluid dynamics ......................... 24
  4.4 Reynolds averaging ................................... 26
  4.5 Equation of state .................................... 27
  4.6 Turbulent Prandtl number ............................. 28
  4.7 Pressure ............................................. 29

5 Methods .................................................. 31
  5.1 OpenFOAM .......................................... 31
  5.2 Computational mesh ................................... 35
  5.3 Validation ........................................... 36
  5.4 Outfall parameter sensitivity study .......................... 38
  5.5 Dimensional analysis .................................. 38

6 Results - Model performance ........................................ 43
  6.1 Grid convergence test .................................. 43
  6.2 Qualitative model results ................................ 44
  6.3 Validation ............................................ 46
Chapter 1

Preface

1.1 Context

Natural Gas is an increasingly important part of the global energy market. The Floating Liquefied Natural Gas (FLNG) facility is a relatively new technological concept. This innovation avoids the need for export pipelines, onshore liquefaction plants, export jetties and other necessary infrastructure for producing LNG. The gas will be processed entirely offshore. The extracted gas from the gas field is liquefied by cooling it down to a temperature of -162 °C. This will shrink its volume by a factor 600. Due to this volume reduction, LNG carriers can transport the LNG to markets worldwide. At an onshore facility the LNG will be re-gassified for industry or personal use. Prelude, Shells first FLNG facility, will be located in the Browse Basin, 230 kilometer of the North West side coast of Australia. The estimated operational lifetime of the facility is 25 years. During this entire period, Prelude will be located above the same gas field, where water depths of over 200 meters occur. The scheduled starting of production will take place in 2017, where it will be capable of producing 3.6 million tonnes of LNG per year.

![Figure 1.1: Artist impression of Shell Prelude Floating Liquefied Natural Gas (FLNG) facility, source: www.shell.com](www.shell.com)

1.2 Shallow water FLNG

Following the Shell Prelude FLNG near the coast of Australia, the FLNG concept is being studied for other geographic locations with relatively shallow water. For FLNG processes, which include cooling the
gas, large volumes of water are used which are extracted from and discharged in the surrounding waters. For Prelude, 150 meters deep water intake risers are to be deployed to provide cooling water for the LNG processing. However, for shallow water FLNG facilities, the cooling water intake will be located inside the hull.

For shallow water moorings, i.e. 30 meter water depth and less, near-field mixing (within 100m of the outfall system) is not well understood. It is important to understand this near-field mixing in order to ensure compliance with environmental regulations and to reduce recirculation risks. Previous studies commissioned by Shell regarding FLNG outfall systems do not provide a satisfactory insight into the near- and far-field mixing of the discharged water around the FLNG.

For shallow water FLNG facilities two problems arise from the cooling water discharge. First, the discharged cooling water should not exceed a given temperature rise outside a defined mixing zone. Second, there is a risk of cooling water recirculation. This could lead to a decreased efficiency of the cooling system if warmed up ambient water would get into the cooling water intake, see figure 1.2 and figure 1.3.

![Figure 1.2: Illustration of the buoyant jet surface spreading, side view](image1.png)

![Figure 1.3: Illustration of the buoyant jet surface spreading, looking at the stern](image2.png)

### 1.3 Environmental regulations for discharging cooling water

Thermal discharges should be designed to ensure that ambient water temperatures do not exceed a given temperature rise outside an established mixing zone. The environmental restrictions are given by the World Bank. The World Bank defines the mixing zone as a zone where initial dilution of a discharge takes place and where exceedence of the water quality standards is allowed [IFC, 2008a]. The mixing zone involves a limited amount of volume, where the initial mixing and dilution of a discharge occurs. How the mixing zone is specified is project specific and where no standard exists, the maximum temperature increase needs to be established through an environmental assessment process. These procedures are time consuming and can be subject to personal interpretation.
In addition to the edge of mixing zone criterion, the World Bank provides a more specific thermal criterion. The criterion states “The cooling water discharge depth should be selected to maximize mixing and cooling of the thermal plume to ensure that the temperature is within 3 degrees Celsius of ambient seawater temperature at the edge of the defined mixing zone or within 100 meters of the discharge point” [IFC, 2008b]. For projects which include a cooling water outfall, the 100 meter criteria is used most of the times in studying the environmental criteria. However, local regulatory agencies and governments can change the local criteria. This introduces some uncertainty for future cooling water discharge criteria. Shell is looking for a tool with which they can quickly assess the temperature distribution of the outfall. This tool should be able to perform a flexible and quick environmental assessment of FLNG cooling water outfalls, regardless of the location and local environmental restrictions. To develop such a tool, the mixing and transport processes of FLNG outfalls need to be understood. Consequently, the research in this thesis focusses on the development and the spreading of the buoyant jet, together with the related mixing and transport processes.
Chapter 2

Introduction

Turbulent jets and plumes are found in a wide variety of natural phenomena and engineering applications [Lee and Chu, 2003]. From the rising smoke plumes from chimneys and factories, fire plumes to volcanic eruptions. Fire plumes have been intensively investigated and this has led to the improvement of building designs and sprinkler systems. The jets of interest for this thesis interact with surrounding waters. These are often encountered for supercritical high velocity flows in many hydraulic applications, for example scouring sluices. Applications in coastal areas are mostly found in pre-treated sewage discharges and cooling water outfalls of power stations and other industries. Figure 2.1 shows a buoyant sewage outfall in Florida.

Thermal or chemical pollutants are often discharged in the ambient environments with different volumes, outflow velocities and density differences. Many of these outfalls have turbulent jet characteristics. Floating Liquefied Natural Gas (FLNG) cooling water outfalls can be characterized as turbulent buoyant jets. The understanding of turbulent jet characteristics and the mixing processes are essential to water quality control and environmental impact and risk assessment.

![Sewage outfall, Broward County, Florida](http://www.marinephotobank.org)

An excellent summary of the theory of jets and plumes can be found in Fischer [1979] and Lee and Chu [2003]. Following the definition of Fischer [1979], a pure jet is typically considered as a flow from an orifice with a pure momentum source. The high velocity injection introduces turbulence and mixing. A plume is a pure source of buoyancy, caused by the density differences between the ambient fluid and the plume. For a plume, local accelerations occur due to the buoyancy flux. These small scale accelerations introduce turbulent mixing. A buoyant jet is a flow consisting of both a source of
momentum and buoyancy. The turbulent mixing for both momentum and buoyancy effects need to be taken into account to determine mixing and jet trajectories. Figure 2.2 includes many of the processes which influence the jet trajectory and behavior. The figure from Jirka and Harleman [1979], includes the jet geometry ($D$), outflow density ($\rho$), outflow angle ($\theta$), effect of turbulent entrainment ($v_e$), jet width ($b$), velocity profiles ($u$), concentration profiles ($c$) and the ambient density ($\rho_a$). All these characteristics which influence the jet will be discussed in this thesis. In the following parts of this thesis, the jet angle will be given by ($\alpha$).

![Figure 2.2: Inclined buoyant jet in stagnant ambient environment, [Jirka and Harleman, 1979]](image)

2.1 Literature review

2.1.1 Historical perspective

Jets have been studied extensively in the past, both experimentally and analytically. Albertson et al. [1950] carried out laboratory experiments on a simple momentum jet in a stagnant ambient fluid. The experimental results demonstrated that the jet velocity and concentration profiles are self-similar and can be well approximated by a Gaussian distribution after the zone of flow establishment. Furthermore, a linear increase of jet width with distance was found in the experimental study. Since the study of Albertson et al. [1950] many other researchers focused their research on other type of jets.

Some of the earlier work on buoyant jets has been carried out by Fan [1967]. He studied the problem of turbulent round and plane buoyant jets in a stagnant ambient fluid. Buoyant jets introduce an initial density difference. The density difference influences the jet trajectory and dilution rates. Fan [1967] found entrainment coefficients for both round and planar jets by performing various experimental studies. In addition, different spreading rates were found. Entrainment of ambient fluid is one of the most important mixing processes found in turbulent jets. The turbulent fluid motions, together with pressure gradients entrain ambient fluid into the jet, see figure 2.3. The entrainment hypothesis relates the inflow of ambient fluid to the local outflow properties of the jet [Fan, 1967].

Abraham [1963] extended the concept of the entrainment hypothesis based on the theory of integral analysis. With the analytical integral analysis, based on balances, various jet paths and dilutions were predicted. Experimental studies on horizontal and vertical buoyant jets were carried out. The integral
Figure 2.3: Turbulent entrainment of ambient fluid

Figure 2.4: Jets with different momentum length scales, $L_M$

The dimensional analysis was extended by Sobey et al. [1988]. He described the stability region for buoyant jets by dimensionless parameters and the effect of bottom and surface attachment due to the Coanda effect. The Coanda effect, is the attachment of jets to nearby walls, induced by the jet entrainment demand. Criteria for stable buoyant jets and surface attachment are proposed based on the experiments. In addition, also Shimada et al. [2004] investigated this Coanda effect and proposed a criteria for attachment in terms of the densimetric Froude number, this results in $Fr > 5.3 H/D$, where $H$ is the water depth and $D$, the jet diameter. The densimetric Froude number gives an indication of whether the jet source characteristics are more jet-like or plume-like. The strength of the jet is directly related to the densimetric Froude number.

In addition to the integral model, other mathematical models have been developed to predict jet mixing. Lee and Cheung [1990] and Chu [1996] have introduced buoyant jets models based on a Lagrangian description of the jet motions. This approach allowed one to calculate the jet width and trajectories in a three-dimensional domain. Kuang and Lee [2006] and Cederwall [1971] have performed studies on vertical buoyant jets. For these cases, the jets and plumes have a perpendicular approach towards the surface. At the point where surface interaction takes place, the jet becomes a buoyant surface jet, which
Cederwall [1971] examined buoyant jets for multi-port diffusors. A two dimensional approach is proposed for merging three dimensional jets by using an equivalent slot jet for multiple momentum jets. Merging jets result in an increased diffusion and also the trajectory of merging jets is influenced. Later, an improved integral model for interacting and merging jets was proposed by Yannopoulos [2010].

2.1.2 Recent research

From 1990 new techniques have been developed to study jets. Doneker and Jirka [1990] developed a system, where many flow classifications are included. Mathematical and theoretical based criteria using length scales are used. Together with experimental data and field measurements the most appropriate flow classifications are used for a particular jet configuration. The results of Doneker and Jirka [1990] formed the basis of the development of the first version of CORMIX [CORMIX-UserGuide, 1996]. The Cornell Mixing Zone Expert System is a software system used for the analysis and prediction of waste- and cooling water discharges. The software is developed by the U.S. Environmental Protection Agency (U.S. EPA) and the Cornell University. CORMIX is built from a flow classification system. This classification systems consists of a database that can distinguish many flow patterns of jets and outfalls. The system uses theoretical established length scale analyses and empirical knowledge to identify the most appropriate flow classifications for the considered jet [CORMIX-UserGuide, 1996]. CORMIX does not need a computational mesh. It uses a sequence of relatively simple simulations, which predict the trajectories and dilution of the jet in the given flow conditions. The outputs of a simulation calculating mass, flow and energy balances are used as new initial conditions for the next simulation. CORMIX can only perform calculations in steady-state situations together with relatively easy bathymetry [CORMIX-UserGuide, 1996].

In addition to the improving modeling techniques, also new measurement methods were applied. The study performed by Chu [1996] includes measurements with tracer concentrations. By means of laser-induced fluorescence (LIF) and laser doppler anemometry (LDA) clear images of concentration fields of a jet under various conditions could be examined. This technique is used to improve the existing mathematical models based on the Lagrangian approach and increase the understanding of buoyant jet mixing and entrainment processes. The LIF and LDA techniques resulted in more reliable data for model testing and validation. Lee and Chu [2003] used this data to improve the integral models and the Lagrangian approach. Jirka [2004] developed a rigorous and general jet integral model to predict jet centerline velocities and diffusion. The jet integral model has been tested for a wide range of conditions with the available high-quality data. The findings of this study have been used to develop the jet integral model CorJet, a near-field jet integral model for single- and multi-port discharges in unbounded ambient environment. CorJet became a part of the CORMIX software.

With improving computational power and new techniques, many jets and waste water studies are examined by means of a numerical study. Mortensen et al. [2013] performed a comparison study of the CFD software OpenFOAM to the CORMIX model. He found that the results of OpenFOAM gave a better prediction to flow spreading and dilution rates than the results of CORMIX.
Computational Fluid Dynamics software solves all the equations of motion for every location in the computational domain. A large amount of CFD software packages are available. Both open-source and commercial packages are used for engineering purposes. Many different flow and bathymetry configuration can be used. If a CFD package is used, the user should determine the computational method and corresponding grid sizes. This depends on the accuracy the user needs for their computations. Three computational methods can be distinguished; DNS, LES and RANS. With Direct Numerical Simulation (DNS) the Navier-Stokes equations are directly solved without the use of a turbulence model. For this thesis it would be too expensive in computational time and the obtained accuracy of the solutions do add value to the thesis objective. Large Eddy Simulations (LES) is based on a filter approach. The small scale turbulence are eliminated from the Navier-Stokes equations. This reduces the computational cost of the simulation. The turbulent motions larger than the defined scale will be calculated exactly. Turbulence scales smaller than the filter size can be modeled by various subgrid-scale models [Versteeg and Malalasekera, 2007]. The last model approach is called the Reynolds Averaged Navier-Stokes (RANS) approach. The approach is based on time averaging. The turbulent fluctuations are no longer modeled, but an average is computed. The RANS model includes a large variety of turbulence closures. Each closure is different in computational cost, accuracy and application. Figure 2.6 shows the results of turbulent jet modeled with DNS, LES and RANS.

Mainly LES and RANS studies have been performed in the area of jets and their performance depends on the turbulence closure used. Aziz et al. [2008] evaluated the accuracy of different \( k-\epsilon \) closure models to the decay of centerline velocities, jet growth, velocity profiles and kinetic energy profiles. He found that the standard \( k-\epsilon \) scheme performed equally well and in some cases better than other turbulence schemes for round turbulent jets.

### 2.2 Research objective

The effect of the surface interaction and the far-field spreading is important for environmental restrictions regarding waste- and cooling water. There is not yet an easy assessment method to examine buoyant temperature plumes at the surface, governed by the outflow properties of the jet. Outflow depth, velocity, temperature, angle and diameter have not yet been directly related to the surface temperature rise. The turbulent jets, induced by a FLNG outfall, contain a large amount of water and heat. These outfalls have to comply with the environmental regulations.

The objective of this study is to obtain an understanding of the buoyant jet characteristics of a FLNG cooling water outfall in order to keep the environmental impact to a minimum. An empirical relation...
needs to be developed to predict the temperature rise for varying outfall designs. The development of this relation is related to the understanding of the diffusion processes and propagation mechanisms of the thermal plume from the outfall system. This results in the following formulation of the research objective:

*Understanding of the jet development and mixing processes related to buoyant jets and the development of an empirical relation for the ambient temperature rise around FLNG cooling water outfalls.*

**Research questions**

To meet the final objective, the following research questions need to be answered.

- What are the physical processes of both pure and buoyant jets?
- What are the governing design- and environmental parameters which influence buoyant jet trajectories and diffusion?
- Which modeling approach is required to capture the relevant physics of jets?
- Are the results from the chosen model approach in line with experimental data?
- What is the relation between backwater temperature rise and outfall design parameters?

**2.3 Research methodology**

In this section the approach of the research is explained. First, an introduction to the relevant physics of jets is given. This part explains all the relevant physical processes involved in both pure jets and buoyant jets. This theoretical background provides a basis for the study of the environmental effects of FLNG outfalls.

The technique used to study the turbulent jets is a numerical model. The numerical open-source CFD software OpenFOAM is chosen to model the outfall problem. This allows for three-dimensional computations of the jet trajectory. This numerical model needs to be able to compute correctly the jet characteristics and diffusion processes. After the setup of the model, the performance of the model is compared to the results of various experimental datasets. If the numerical model proves to give reliable results, the simulations of different FLNG outfall designs can start.

With the numerical model, various outfall designs are simulated. These simulations include a parameter sensitivity study on the surface temperature rise. The results of this study lead to a set of outfall parameters, which significantly impact the ambient temperature rise. The parameters resulting from this study are used to develop an empirical relation of the surface temperature rise. A dimensional analysis is then used to find the relations between the surface temperature rise and the initial source characteristics of the outfall. The physical meaning of the relation is explored and the empirical relation is tested for its accuracy and robustness.

**2.3.1 Research scope**

The research in this thesis focuses on the environmental restrictions prescribed for cooling water outfalls. Therefore, the domain of the study will be restricted to the area around the FLNG, which is influenced by the jet induced by the FLNG outfall. This results in a square box domain behind the stern of the FLNG. The illustration in figure 2.7 represents the area around the FLNG cooling water outfall considered for this thesis. The hull of the facility is not taken into account. In addition, no tides and waves will be considered. The temperature in the domain is constant, with a fixed water level and only a single jet is considered.
2.4 Thesis outline

In chapter 3 the physical behavior and processes involved in the mixing of buoyant jets will be discussed. The purpose of this chapter is to develop an understanding of jet characteristics and their influence on the mixing and propagation in the ambient environment. Fundamental theoretical background and understanding of the phenomena related to thermal plumes are essential to use and interpret the model results.

In chapter 4 the governing equations are presented. This chapter provides insight into the assumptions and limitations involved in simulating the physical processes.

In chapter 5 the methods are described. Here, the chosen modeling software will be discussed. The chosen CFD software will be described together with the initial- and boundary conditions. In addition the setup of the model and setup of the computational mesh is provided.

In chapter 6 the model results will be compared to known physical phenomena and experimental data. If the model results prove to be in good agreement with the experimental data, it can be assumed that the model represents correctly the involved physical processes. This is done for both pure jets and buoyant jets. If successful, the chosen mesh and model setup can be used for the parameter sensitivity study. In addition, various outfall scenarios are studied. From these scenarios, the parameters which are the most important for the ambient temperature rise are selected. In addition, the approach of a dimensionless analysis is introduced.

In chapter 7 the results of the various outfall scenarios are given. The effect of the ambient temperature rise and the underlying physics of the results are discussed. The scenarios lead to a group of parameters important for studying the ambient temperature rise. These results are used to find relations between different dimensionless groups of parameters. The relationships found, are used to formulate an empirical relation of the surface temperature rise as a function of the outfall parameters. The empirical relation will also be tested for accuracy and robustness.

Finally, chapters 8 and 9 will discuss and describe the main conclusions of this thesis and will present recommendations for future research.
Chapter 3

Theoretical background of jets and plumes

3.1 Introduction

The jet initial conditions play an important role in the flow patterns near the jet orifice. The initial conditions are governed by the initial jet momentum, outflow velocity and initial density differences.

The jet mass flux, momentum flux and buoyancy flux influence the flow patterns of the jet. In equation 3.1.1 the three fluxes are given. The mass flux ($\rho q$) describes the amount of mass passing through a cross section per unit of time. The momentum flux ($\rho m$) gives the amount of momentum passing a jet cross section per unit of time. Finally, the buoyancy flux ($\rho b$) of the jet is defined as the buoyant or submerged weight of the fluid passing through a cross section per unit of time.

$$
\rho q = \int_A \rho u \, dA \quad , \quad \rho m = \int_A \rho u^2 \, dA \quad , \quad \rho b = \int_A g \Delta \rho u \, dA
$$

(3.1.1)

In equation 3.1.1, $q$ is called the specific mass flux, $m$ is the specific momentum flux, $b$ is the specific buoyancy flux, $u$ is the time averaged jet velocity, $A$ is the cross-sectional area of the jet and $\Delta \rho$ is the difference in density between the fluid in the jet and the surrounding water body. The generic formulation of fluxes in equation 3.1.1 are valid for the whole jet trajectory. The initial values of the volume flux, momentum flux and buoyancy flux for a round jet orifice are given by equation 3.1.2 - 3.1.4 [Fischer, 1979].

Volume Flux, $Q_0 = \frac{1}{4} \pi D^2 U_0$  

(3.1.2)

Momentum Flux, $M_0 = \frac{1}{4} \pi D^2 U_0^2$  

(3.1.3)

Buoyancy Flux, $B_0 = \frac{g \Delta \rho}{\rho} Q = g' Q$  

(3.1.4)
The reduced gravitational acceleration \( g \frac{\Delta \rho}{\rho} \) in the buoyancy flux term is of great importance for buoyant jets. The fluxes \( Q_0, M_0 \) and \( B_0 \) are the governing parameters for a round turbulent jet. The turbulence of the jet is related to the jet Reynolds number. For a round jet the Reynolds number is given by equation 3.1.5, with the outflow velocity \( u \), the jet diameter \( D \) and the kinematic viscosity \( \nu \). The jet is called fully turbulent if the \( Re > 4000 \). Only fully turbulent jets will be discussed in this thesis. Many engineering outfalls are in the fully turbulent range. This is also the case for FLNG outfall systems.

\[
Re = \frac{uD}{\nu}
\]  

(3.1.5)

### 3.2 Length scales

From the fluxes \( Q_0, M_0 \) and \( B_0 \) several jet characteristic length scales can be formed. The length scales describe the relative importance of discharge volume flux, momentum flux, buoyancy flux and ambient cross flow. The length scale, found by Fischer [1979], are often used in dimensional analyses and scaling of jet parameters. Relevant length scales found by Fischer [1979] are the characteristics volume length scale \( L_q \), the momentum length scale \( L_M \), and the buoyancy length scale \( L_b \).

\[
L_q = \frac{Q_0}{M_0^{1/2}},
\]  

(3.2.1)

\[
L_M = \frac{M_0^{3/4}}{B_0^{1/2}},
\]  

(3.2.2)

\[
L_b = \frac{B_0}{u_a}.
\]  

(3.2.3)

For round turbulent jets, the above equations result in equations 3.2.5 - 3.2.7. The length scales can be related to a distance \( z \) along the jet trajectory. This will provide an useful insight in the dominant processes at a certain location along the jet trajectory. From equation 3.2.5 it becomes clear that \( L_q \) is only depending on the jet geometry. For distances \( z \ll L_q \), the initial geometry has the largest effect on the jet flow. Where, \( z \gg L_q \), the volume flux \( Q \) becomes less important and the jet is mostly influenced by the initial momentum. For distances \( z \gg L_q \), the flow of called fully developed.

\[
\frac{M^{1/2}z}{Q} = \frac{z}{L_q}
\]  

(3.2.4)

The same approach can be used for the momentum length scale \( L_M \). The momentum length scale consists of both momentum- and buoyancy flux. The momentum length scale gives a measure for the distance, along jet trajectory, where buoyancy effects become more important than the jet momentum. For \( z \ll L_M \) this results in a flow that has mainly jet-like characteristics. For \( z \gg L_M \), the flow has mainly plume-like characteristics. An illustration of the momentum length scale is already given in figure 2.4. In this thesis, the momentum length scale \( L_M \) proves to be a very important parameter to characterize buoyant jets. The buoyancy length scale gives a measure of the strength of the buoyancy
over the ambient current.

\[ L_q = \frac{\sqrt{\pi} D_0}{2} \]  

(3.2.5)

\[ L_M = \frac{u_0 (\pi D_0^2 / 4)^{1/4}}{\sqrt{\Delta \rho / \rho_0} g} \]  

(3.2.6)

\[ L_b = \frac{u_0 \pi D_0^2 g_0'}{4 u_0^3} \]  

(3.2.7)

### 3.3 Pure jet physics

For pure jets, the outflow density is similar to the ambient density. Many laboratory experiments on a simple momentum jet in a stagnant ambient were carried out by Albertson et al. [1950]. In these studies, the development of a pure jet is divided in roughly two stages. The zone of flow establishment (ZFE) and the zone of established flow (ZEF).

#### 3.3.1 Zone of Flow Establishment

The zone of flow establishment (ZFE) is defined as the zone where the jet exits the nozzle up to the point where the jet is fully developed. When a jet exits the nozzle it will immediately begin to mix with the ambient fluid. For a round jet, a turbulent mixing layer surrounds the core of the jet. Within the ZFE, the velocity in the core is the same as the exit velocity from the nozzle. The mixing layer grows when more ambient fluid is entrained in the jet. At the same time the size of the core is decreasing. This is clearly visible in figure 3.1. The velocity and concentration in the core are not affected by the mixing processes. When the mixing layer grows up to the center of the jet, the jet is called fully developed. At this point the centerline velocity \( u_m \) becomes less than the initial jet velocity \( u_0 \).

At the point where the jet is fully developed, the radial velocity profile can be described by a Gaussian distribution. It is shown by experiments that the length of the ZFE is approximately equal to 6.2 \( D_0 \).
CHAPTER 3. THEORETICAL BACKGROUND OF JETS AND PLUMES

where $D_0$ is the diameter of the jet orifice, see figure 3.1. Lee and Chu [2003] give the following expressions of the axial velocity and concentration profile in the zone of flow establishment (ZFE) with the local velocity $u$, local concentration $c$, initial concentration $c_0$, the jet width $b_g$, the radius of the jet core $R$ and the ratio of velocity width over concentration width $\lambda$.

$$u = u_0, \quad c = c_0; \quad r \leq R,$$

$$u = u_0 \exp\left(-\frac{(r-R)^2}{b_g^2}\right), \quad c = c_0 \exp\left(-\frac{(r-R)^2}{\lambda^2 b_g^2}\right); \quad r \geq R.$$  

(3.3.2)

The ZFE is clearly visible in figure 3.2. This figure shows by various measurements that the centerline velocity reaches up to approximately six times the jet diameter. Also the decay of the centerline velocity in the ZEF is well described. In chapter 6.3 the results from the numerical model are compared to the same data sets.

![Figure 3.2: Profile of time averaged velocity in a round jet, [Lee and Chu, 2003]](image)

3.3.2 Zone of Established Flow

For distances larger than 6.2·$D$, the jet is called fully developed. The axial velocity and concentration profiles are found to be self-similar and Gaussian distributed [Albertson et al., 1950]. This is called the zone of established flow (ZEF). The self-similarity means that at any cross section, the time averaged velocity and concentration profile result in a Gaussian distribution. Equation 3.3.3 give the expression for the velocity- and concentration profiles in the ZEF [Lee and Chu, 2003]. The maximum centerline velocity and concentration are given by $u_m$ and $c_m$. Where $r$ is the radial distance from the centerline. Note that there is a difference in spreading of the velocity and concentration over the width by a factor $\lambda$ (figure 3.3). This is to account for the difference between the diffusion of mass and diffusion of momentum.

$$u = u_0 \exp\left(-\frac{r^2}{b_g^2}\right), \quad c = c_0 \exp\left(-\frac{r^2}{\lambda b_g^2}\right)$$

(3.3.3)

It is found that the turbulent round jet, without buoyancy or ambient currents, spreads linearly with a factor $\beta$. Albertson et al. [1950] and Fiedler [1996] suggested a jet spreading rate of $\beta_G = \frac{db}{dx} = 0.114$. 

16
where subscript $G$ refers to a Gaussian profile. By performing an analysis with the governing equations of momentum and continuity equations for a round jet, the following expressions with $\lambda$ and $\beta$ are obtained [Lee and Chu, 2003]:

$$b(x) = \beta x,$$

$$\frac{u_{\infty}}{u_0} = \frac{1}{\sqrt{2\beta}} \left( \frac{x}{D} \right)^{-1},$$

$$\frac{c_m}{c_0} = \frac{1 + \lambda^2}{2\sqrt{2\lambda^2\beta}} \left( \frac{x}{D} \right)^{-1}.$$  

Figure 3.4 shows the results of measured radial velocity profiles for multiple downstream jet locations. This normalized velocity profile proves to be self-similar and fits well to a Gaussian distribution. Chu [1996] showed by Laser-induced-Fluorescence techniques the radial concentration distribution in the jet cross-section. The measured concentration is normalized by the maximum concentration in the center of the jet. As noted before, the radial time averaged velocity and concentration distribution are not equal, however they are self-similar. Figure 3.5 shows the measurements from Chu [1996] where the data fits very well to a Gaussian distribution. In this figure $b_{gc} = \lambda b_g$, which is called the concentration half-width, where $C$ takes the value $e^{-1}C_m = 0.37C_m$. The Gaussian concentration profile $b_{gc}$ is in general wider than the velocity profile $b_g$ (figure 3.3). Measurements of Albertson et al. [1950] and Chu [1996] show that $\lambda = b_{gc}/b_g = 1.2$.

Figure 3.3: Gaussian distribution of velocity and concentration, [Chu, 1996]
3.4 Buoyant jet physics

The mixing zones for buoyant jets discharging continuously into an ambient body of water can be subdivided into a near-field and far-field region. For the near-field region, the initial jet characteristics, momentum flux, buoyancy flux and outfall geometry strongly influence the jet trajectory and mixing processes [Jirka et al., 1992]. This region also includes the buoyant jet surface interaction. In the near-field region of the jet, outfall design can highly influence the initial mixing characteristics. In the far-field region, the initial source characteristics become less important. As the buoyant plume travels further away, conditions in the ambient environment will control the jet trajectory through buoyant spreading and passive ambient diffusion [Jirka et al., 1992]. However, the near-field and far-field regions are connected. Influencing the near-field region by changing the outflow parameters has an effect on the distribution characteristics in the far-field.

In the description of the buoyancy flux, the reduced gravity is used. In a static ambient environment, two forces act on a small fluid element, first the downward force of gravity and second the upward pressure force introduced by the weight of the displaced ambient fluid. The net force is called the buoyancy force \( F_b = (\rho_a - \rho)g\delta V \).

Buoyant jet parameters

Buoyant jet parameters are used to indicate the characteristics of a specific buoyant jet. The value of these parameters can already indicate something about the initial jet characteristics and their behavior in the ambient environment.

The dimensionless densimetric Froude number gives an indication of whether the source characteristics are more jet-like or plume-like. With \( Fr_d \ll 1 \) the flow is acting as a pure plume, for \( Fr_d \gg 1 \) the flow is
acting like a pure jet. The strength of the jet is directly related to the densimetric Froude number. The jet will turn rapidly into a plume if $Fr_d$ is only slightly larger than 1. For $Fr_d > 1$ the initial momentum will force the jet into the direction of the jet orientation. Buoyancy becomes dominant and bend the jet in upward direction. The initial densimetric Froude number is given by:

$$Fr_d = \frac{u_0}{\sqrt{\frac{(\rho_0 - \rho_a)}{\rho_a}} g D}$$

(3.4.1)

Any jet flow which contains a continuous source of buoyancy will eventually behave like a plume at some distance from its source. This is the result of the buoyant forces that remain, while the momentum of the jet decreases. The point where buoyancy becomes dominant over the initial jet momentum is given by the momentum length scale $L_M$, already provided in equation 3.2.2. It is possible to rewrite the momentum length scale in terms of densimetric Froude number. The momentum length scale is a measure for the distance of which buoyancy becomes more important than the jet momentum. The unit of the momentum length scale $L_M$ is meter [m].

$$L_M = \frac{u_0 (\pi D_0^2 / 4)^{1/4}}{\sqrt{(\Delta \rho / \rho_a)} g} = \left(\frac{\pi}{4}\right)^{1/4} Fr_d D$$

(3.4.2)

Next to the densimetric Froude number, another dimensionless parameter is introduced. The jet Richardson number, which is defined by the ratio of $L_q / L_M$. The Richardson number is often used as it leads to simpler expressions and always has a value between 0 and 1. Next to that it provides a better physical insight, because the number can always be related to the ratio of the length scales of volume flux and momentum flux. The Richardson number can also be written in terms of the densimetric Froude number $Fr_d$. For a round jet this can be written as:

$$R_0 = \frac{L_q}{L_M} = \frac{QB^{1/2}}{M^{5/4}} = \left(\frac{\pi}{4}\right)^{1/4} \frac{1}{Fr_d}$$

(3.4.3)

### 3.4.1 Stage of Intermediate

In the stage of intermediate, both jet momentum and buoyancy govern the jet flow. For a horizontal buoyant jet, the outflow will cause the jet to move horizontally. At the same time, buoyancy forces induced by the density difference force the jet to move in upward direction. Figure 3.7 illustrates the region considered the stage of intermediate.

A positively buoyant jet will eventually interact with the water surface. The point where the buoyant jets meets the surface is called the point of surface impingement. The jet will deflect in a horizontal direction and will result in a stable layer at the surface. An unstable circulation layer will occur if the momentum of the jet is very high, combined with a low buoyancy. These unstable type of mixing processes do not have a clear change in density profile over depth. Figure 3.8 gives an example of a stable buoyant surface spreading.
3.4.2 Stage of plume

After the point of surface impingement, the jet will spread in horizontal direction (figure 3.7). The horizontal spreading is governed by advection, induced by the jet momentum, and buoyant spreading. The buoyant spreading is caused by the density difference between the plume and the ambient fluid. With large density differences, the plume can spread in the direction perpendicular to the jet direction. This spreading results in a decrease in the layer thickness. In the frontal zone, the plume mixes with the ambient fluid.

At a further distance from the jet orifice, the plume is highly influenced by the ambient parameters. At this point the mixing is governed by passive ambient diffusion. The existing turbulence in the environment will mix the plume in the vertical direction. An increase in plume thickness is observed, while at the same time density differences decrease. This will continue up to the point where mixing has occurred over the whole depth (figure 3.9). The strength and intensity of the passive ambient diffusion depends on the ambient conditions. These include currents, tides, waves, wind, bottom friction and so forth.
3.5 Water depth classification

Different classification systems of the water depth are used for buoyant jets and plumes. In this section the classification of Lee and Jirka [1981] is used. The water depth is classified in three different categories. All the categories are based on the available water column above the outlet, compared to the outlet diameter.

Deep water is defined as a water depth above the outlet much greater than twenty jet diameters \( (d/D) \gg 20 \). The velocity and density profiles outside the zone of established flow can be treated as self-similar or Gaussian. The buoyant jet will develop into a plume and buoyancy effects will be dominant. The approach of the buoyant plume to the surface is perpendicular, resulting in a radial spreading at the surface.

Shallow water jets are defined as jets, discharged from depths of 6 to 20 jet diameters \( (6 < d/D < 20) \) [Lee and Jirka, 1981]. If the jet does not have sufficient buoyancy or has a high momentum flux, there is a possibility of recirculating or unstable flow patterns. The velocity and density profiles outside the zone of established flow, can still be approximated by a Gaussian distribution. Both buoyancy and jet momentum affect the surface spreading of the buoyant plume.

For very shallow water \( (d/D) < 6 \), the jet can attach to the surface. This would result in momentum dominated surface flows. The jet trajectory can be influenced by the nearby presence of a boundary. Jets have the tendency to attach to boundaries if the nozzle is located near the bed or near the water surface. Jet attachment can significantly change the dilution and turbulent entrainment of the jet. Due to low pressure effects created by the jet entrainment, one side of the jet will attach to the boundary, this is called the Coanda effect. Shimada et al. [2004] found a relationship between the jet submergence \( d \), jet distance above bed \( H \) and the jet diameter \( D \).

1. Where \( d \) is sufficiently larger than \( H \) the jet will attach to the channel bed, \( (d/D) \leq 2.2H/D \)
2. Where \( H \) is sufficiently smaller than \( D \), the jet will attach to the water surface, \( (d/D) \leq 0.27H/D + 4.2 \)
3. Where \( H \) and \( D \) are roughly equal the deflection is not fixed, \( 0.27H/D + 4.2 \leq d/D \leq 2.2Z/D \)

For a jet close to the surface, the water level will be lowered due to the pressure difference. The Coanda effect is also examined by Jirka and Domeker [1991]. The criteria they proposed for attachment is \( H/L_m < 0.2 \), in terms of the densimetric Froude number this results in \( Fr_d > 5.3H/D \).
3.6 Turbulent entrainment

3.6.1 Buoyant jet

In the near-field region of the jet, the volume of fluid inside the jet is increasing while the jet moves along its axis. At a short distance from the jet orifice, large velocity and pressure gradients between the jet and nearby fluid exist. This introduces a turbulent shear layer and unstable flow patterns. The turbulent fluid motions, together with the pressure gradient entrain ambient fluid into the jet. This causes an increase in jet volume and decrease in concentration and temperature. The streamlines of the ambient fluid towards the jet are already illustrated in figure 2.3.

The basis of the entrainment hypothesis is to relate the inflow of ambient fluid to the local outflow properties of the jet. Fischer [1979] hypothesized that the velocity of inflow of an ambient fluid would be proportional to the maximum mean velocity in the jet. In other words, the rate of change in volume with distance along the jet is equal to the rate of inflow by entrainment. The entrainment hypothesis is given in equation 3.6.1. The hypothesis holds for many different jets, only the entrainment coefficient, \( \alpha \) is different for different jet configurations. \( Q_e \) represents the local entrainment flux into the jet, \( v_e \) the entrainment velocity, which is assumed to be proportional to the centerline velocity \( u_m \) (equation 3.6.2) and the entrainment coefficient \( \alpha \).

\[
\frac{dQ}{dx} = \frac{d}{dx} (\pi u_m b_g^2) = 2\pi (r \alpha u_m) = Q_e \quad (3.6.1)
\]

\[
v_e = \alpha u_m \quad (3.6.2)
\]

3.6.2 Buoyant plume

In the far-field region of the buoyant plume turbulence and entrainment takes place. The turbulence is often generated by shear stresses between layers of different flow velocities or between the interface of water to air, for example wind shear stress. The resulting turbulent flow in the upper part of the water column can entrain fluid from the layer beneath. Heavier fluids are lifted over lighter fluids as a consequence of the turbulent motions and vortices. This leads instabilities and introduces vertical mixing of the buoyant plume, which will result in a decrease of the temperature of the plume in time.

3.7 Jet changing parameters

3.7.1 Ambient currents

In the far-field region of the jet, the ambient conditions become more important. Currents can change the jet trajectory and transport the thermal plume over large distances. The most limiting current direction for considering the environmental regulations is a co-flowing current [Cederwall, 1971]. This is the case if the current direction and the jet outflow direction are similar. This restrictive case is true for a constant current over depth, without any obstacles. For a co-flowing current around a FLNG facility, the current pattern would be complicated. In the case of a FLNG facility large eddies occur behind the stern. These eddies influence the current pattern and predicting the current pattern becomes complicated. A jet which is subject to a cross-flowing current deflects to the direction of the current. The vortex entrainment, on the wake side of the jet, becomes the dominant entrainment process. This results in a larger diffusion and decrease in temperature for a buoyant jet.
Chapter 4

Governing equations of fluid dynamics

4.1 Introduction

The transport of heat is governed by the flow field. However, this movement of the flow is at the same influenced by the distribution of heat. To understand how the flow is influenced, one needs to understand how the heat transport relates to the flow conditions and the other way around. This section describes mathematically the involved physical processes, which are used in numerical models. A Cartesian frame of reference is used.

Some general assumptions for the fluid properties are used in the further elaboration of the governing equations.

1. Continuum; this assumes that the fluid completely fills the space it occupies. The fluid is described on a macroscopic scale.

2. Incompressible fluid for pressure ($\partial \rho / \partial p = 0$), $\rho$ is only a function of temperature, $\rho = \rho(T)$

3. Isotropic; material properties do not vary with direction.

4. Constant dynamic viscosity, $\nu = \mu = \nu(T)$

4.2 Total derivative

In this section a general mathematical description is provided for the rate of change of a flow with a physical quantity $q$. This physical quantity can for example be momentum, heat or concentration. An infinitesimal small fluid element is considered with its dimensions $\partial x, \partial y, \partial z$, with along its axes respectively the unit vectors $i, j$ and $k$. The vector velocity field is given by $V = iu + jv + kw$, [Bat, 1967].

The total derivative of the physical quantity $q$ can be written as:

$$\frac{Dq}{Dt} = \frac{\partial q}{\partial t} + \frac{\partial q}{\partial x} u + \frac{\partial q}{\partial y} v + \frac{\partial q}{\partial z} w \equiv \frac{\partial q}{\partial t} + \mathbf{V} \nabla q. \quad (4.2.1)$$

The Lagrangian derivative 4.2.1 is expressing a derivative following the motion of the fluid.
4.3 Conservation laws of fluid dynamics

The equations of motion for a homogeneous fluid are based on three physical conservation laws. The fundamental equation of the fluid motion are:

1. Conservation of mass (continuity)
2. Conservation of momentum (Newton’s second law)
3. Conservation of energy (first law of thermodynamics)

The resulting system of equations is known as the Navier-Stokes equations and will be introduced in this chapter.

Conservation of mass

Mass conservation is represented by the continuity equation. The mass conservation equation contains the density (\(\rho\)) as the conserved quantity as it is by definition mass per unit volume. If a fluid with a density \(\rho\) flows into a small fluid element \((dx, dy, dz)\) a mass balance of the element can be made [Stewart, 2008]. In a steady state process, the rate which mass enters the volume is equal to the rate at which mass leaves the system.

\[
\frac{\partial \rho}{\partial t} + \nabla (\rho u) = 0
\]  
(4.3.1)

The continuity equation:

\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u)}{\partial x} + \frac{\partial (\rho v)}{\partial y} + \frac{\partial (\rho w)}{\partial z} = 0.
\]  
(4.3.2)

Conservation of energy

For buoyant jets, the heat transfer from the jet to the ambient water is of great importance. For buoyancy driven flows the density changes due to temperature variations. The distribution of the heat influences the flow pattern. The density changes are considered in the gravitational term of the momentum equation, therefore all three conservation equations become coupled through the equation of state, which will be discussed in the next section. Energy in flows are present in many forms. Energy as kinetic energy, due to the mass and velocity of the fluid, thermal energy, chemical energy and potential energy [Andersson et al., 2012]. The advection-diffusion equation of temperature is given by:

\[
\rho C_p \frac{DT}{Dt} = k \left( \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} \right)
\]  
(4.3.3)

Where \(T\) is the temperature, \(\alpha_t\) the thermal diffusivity \((\alpha_t = k/\rho C_p)\) and \(C_p\) is the specific heat capacity \((J/(kg \cdot K))\). Where \(k\) is the thermal conductivity\((W/(m \cdot K))\), \(\rho\) the density \((kg/m^3)\). In this equations the dissipation function \(\Phi\), representing the work done against viscous forces, is neglected. This term is only important where viscous heating is not negligible [Mondal and Mukherjee, 2012].
Conservation of momentum

The rate of change of momentum of a fluid element equals the external forces on the element. For the x-direction it is written as:

\[
\frac{\partial \rho u}{\partial t} + \frac{\partial (\rho u^2)}{\partial x} + \frac{\partial (\rho uv)}{\partial y} + \frac{\partial (\rho uw)}{\partial z} = F \tag{4.3.4}
\]

Where \( F \) is the resultant of all forces acting on the fluid element. Four forces are important to consider.

- Pressure forces
- Friction forces
- Gravitational forces
- Coriolis force

Due to a pressure difference of two opposing faces, with a surface \((dydz)\), a resultant force is acting on the fluid element. The pressure balance in the x-direction is given by [Stewart, 2008] equation 4.3.5. This equation is similar for pressure differences in both y and z direction.

\[
F_x = -\frac{\partial p}{\partial x} dx dy dz \tag{4.3.5}
\]

Due to friction, stresses are introduced at the faces of the fluid element. In section 4.4 a derivation of these forces due to viscous stresses and their physical meaning will be discussed in more detail. The force due to viscous stresses in x-direction is given by:

\[
\left( \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} \right) dx dy dz \tag{4.3.6}
\]

The gravitational force is defined as the mass of the fluid element times the gravitational acceleration in a downward direction.

\[
F_g = -\rho g (dx dy dz)
\]

In most observations and calculations a reference frame fixed to the Earth’s surface is used. However, the Earth is rotating around its own axis. On large scales, this rotation deflects fluid to the right in the Northern Hemisphere and to the left in the Southern Hemisphere. To take this effect into account, while using a fixed reference frame, the Coriolis force is introduced.

The velocity of the Earth’s rotation is not the same at every point on Earth. At the equator the speed has a maximum and at the poles the speed is zero. However, the angular velocity of the Earth’s rotation, \( \Omega \) is constant. For small regions relative to the Earth’s radius, the \( f \)-plane approximation can be made [Pietrzak, 2013]. This implies that the Coriolis force is assumed to be constant with latitude \( \varphi \). The Coriolis force is than given by equation 4.3.7. For the scale considered in this thesis, the coriolis force will be small compared to the other forces.

\[
f = 2\Omega \sin \varphi \tag{4.3.7}
\]
CHAPTER 4. GOVERNING EQUATIONS OF FLUID DYNAMICS

All the previous mentioned forces are put into equation 4.3.4. If this equation is divided by the element volume \( dx\,dy\,dz \), the momentum equations for all three directions are found [Stewart, 2008]. These are known as the Navier Stokes equations:

\[
\begin{align*}
\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} &= -\frac{\partial p}{\partial x} + \frac{\partial^2 \pi}{\partial x^2} + \frac{\partial^2 \pi}{\partial y^2} + \frac{\partial^2 \pi}{\partial z^2} - \frac{\partial u' u'}{\partial x} - \frac{\partial u' v'}{\partial y} - \frac{\partial u' w'}{\partial z} \\
\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} &= -\frac{\partial p}{\partial y} + \frac{\partial^2 \pi}{\partial x^2} + \frac{\partial^2 \pi}{\partial y^2} + \frac{\partial^2 \pi}{\partial z^2} - \frac{\partial u' u'}{\partial x} - \frac{\partial u' v'}{\partial y} - \frac{\partial u' w'}{\partial z} \\
\frac{\partial w}{\partial t} + u \frac{\partial w}{\partial x} + v \frac{\partial w}{\partial y} + w \frac{\partial w}{\partial z} &= -\frac{\partial p}{\partial z} + \frac{\partial^2 \pi}{\partial x^2} + \frac{\partial^2 \pi}{\partial y^2} + \frac{\partial^2 \pi}{\partial z^2} - \frac{\partial u' u'}{\partial x} - \frac{\partial u' v'}{\partial y} - \frac{\partial u' w'}{\partial z}
\end{align*}
\] (4.3.8)

4.4 Reynolds averaging

Due to the high velocities gradients at the point where the jet enters the ambient fluid, turbulence is introduced in the water column. For a laminar flow \((Re < 2000)\), the Navier Stokes equations provide a stable solution. However, for fully turbulent flows \((Re > 4000)\) the non-linear convective terms in equation 4.3.8 become dominant over the viscous terms, causing instability [Andersson et al., 2012]. To be able to compute turbulence in a numerical model, Reynolds decomposition is introduced and the equations are solved for an average value and a fluctuating part (figure 4.1). For example the horizontal velocity becomes; \(u = \bar{\pi} + u'\), where \(\bar{\pi} = 0\). Averaging the velocity results in \(\bar{\pi} + u' = \pi\). Not only the velocities fluctuate due to turbulence, also the pressure results in small turbulent fluctuations over time. The same approach is used to solve these pressure fluctuations.

![Turbulent velocity fluctuation as a function of time](image)

**Figure 4.1: Turbulent velocity fluctuation as a function of time**

4.4.1 Reynolds averaged Navier-Stokes equations (RANS)

The decomposed velocities and pressures can be substituted into the three directional Navier-Stokes equations 4.3.8. The velocities and pressure contain an average and a fluctuating part. If then, the whole equations are averaged, a new equation for the mean flow is found. This procedure is called Reynolds averaging. The result obtained by averaging and substituting the mean and fluctuating velocities into the momentum equations are called the Reynolds Averaged Navier-Stokes (RANS) equations:

\[
\begin{align*}
\frac{\partial \bar{u}}{\partial t} + u \frac{\partial \bar{u}}{\partial x} + v \frac{\partial \bar{u}}{\partial y} + w \frac{\partial \bar{u}}{\partial z} &= -\frac{1}{\rho} \frac{\partial \bar{p}}{\partial x} + \nu \left[ \frac{\partial^2 \bar{\pi}}{\partial x^2} + \frac{\partial^2 \bar{\pi}}{\partial y^2} + \frac{\partial^2 \bar{\pi}}{\partial z^2} \right] - \frac{\partial u' u'}{\partial x} - \frac{\partial u' v'}{\partial y} - \frac{\partial u' w'}{\partial z} \\
\frac{\partial \bar{v}}{\partial t} + u \frac{\partial \bar{v}}{\partial x} + v \frac{\partial \bar{v}}{\partial y} + w \frac{\partial \bar{v}}{\partial z} &= -\frac{1}{\rho} \frac{\partial \bar{p}}{\partial y} + \nu \left[ \frac{\partial^2 \bar{\pi}}{\partial x^2} + \frac{\partial^2 \bar{\pi}}{\partial y^2} + \frac{\partial^2 \bar{\pi}}{\partial z^2} \right] - \frac{\partial u' u'}{\partial x} - \frac{\partial u' v'}{\partial y} - \frac{\partial u' w'}{\partial z} \\
\frac{\partial \bar{w}}{\partial t} + u \frac{\partial \bar{w}}{\partial x} + v \frac{\partial \bar{w}}{\partial y} + w \frac{\partial \bar{w}}{\partial z} &= -\frac{1}{\rho} \frac{\partial \bar{p}}{\partial z} + \nu \left[ \frac{\partial^2 \bar{\pi}}{\partial x^2} + \frac{\partial^2 \bar{\pi}}{\partial y^2} + \frac{\partial^2 \bar{\pi}}{\partial z^2} \right] - \frac{\partial u' u'}{\partial x} - \frac{\partial u' v'}{\partial y} - \frac{\partial u' w'}{\partial z}
\end{align*}
\] (4.4.1)
4.4.2 Reynolds stresses

The result is that the equations are written such that the large scale flows are separated from the small scale, fluctuating flows. From equation 4.4.1 a closure problem arises. The RANS equation can only be solved by modeling the Reynolds stresses, \( R_s = -\rho \overline{u'v'} \). The Reynolds stresses contain three normal stresses and three shear stresses [Versteeg and Malalasekera, 2007]. The resulting set of unknowns is larger than the available equation. This closure problem can be resolved by applying the Boussinesq hypothesis, where the stresses are related to the mean flow with the concept of eddy viscosity [Versteeg and Malalasekera, 2007].

\[
\frac{-w_i u_j}{u_i u_j} = \nu_t \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} k
\]  

(4.4.2)

The eddy viscosity \( \nu_t \) is represented by:

\[
\nu_t = C_\mu \frac{k^2}{\epsilon} \tag{4.4.3}
\]

4.4.3 Turbulence closures

Several turbulence closures are available to apply in a numerical model. The turbulence models are based on empirical relationships between turbulence and mean flow. The Boussinesq hypothesis allows to relate the small scale fluctuations to the mean flow. Turbulence closure models are necessary to include turbulence in a volume element much larger than the scale of turbulence itself. With CFD modeling, the Navier-Stokes equations are solved for the volume elements in the computational domain.

The most common and widely validated turbulence closure model is the standard \( k - \epsilon \) model [Launder and Spalding, 1974]. This is a two equation model, which gives a description of the turbulence by means of two transport equations. The first transported variable is the turbulent kinetic energy, \( k \). The second transport variable is the turbulent dissipation, \( \epsilon \) which determines the rate of dissipation for the turbulent kinetic energy. The constants used in the standard \( k - \epsilon \) are empirically found for a wide range of turbulent flows. Appendix B gives the equations of this turbulence closure. The standard \( k - \epsilon \) works well for free shear flow with high Reynolds numbers [Bardina et al., 1997]. Therefore, this turbulence closure is used for the CFD computations in this thesis. However, the standard \( k - \epsilon \) does not work well for low Reynolds regions and flows involving rotation, boundary layers, separation and swirling flows.

4.5 Equation of state

The density of water is calculated from measurements of temperature (T), salinity or conductivity (S) and pressure (p). This calculation is done with the equation of state which relates density to the previous quantities. For incompressible fluid the pressure effects can be neglected and the equation of state simplifies [Andersson et al., 2012]. This relation is non-linear and is given by the equation of state, where the density is depending on the salinity, temperature and pressure.

\[
\rho = \rho(S, T, p) \tag{4.5.1}
\]

The salinity of the water influences the relation of the temperature to the density. Density increases when salinity increases and the density decreases as the temperature increases. However, the amount
the density changes due to temperature depends on the salinity. These variations are small, but very important for the study of buoyant jets.

For many buoyant flows, faster convergence is obtained by applying the Boussinesq approximation of thermal expansion. Instead of treating the fluid density as a function of temperature, the Boussinesq approximation treats density as a constant value in all solved equation, except for the buoyancy term in the momentum equation [OpenFOAM-UserGuide, 2012]. By applying the Boussinesq approximation the density is assumed to vary linearly with the temperature. The density variations due to thermal expansion are then given by equation 4.5.2. Where $\beta$ is the thermal expansion coefficient. The magnitude of the buoyancy is governed by the relative density difference not the absolute value of the density. According to Sverdrup et al. [1942], the expansion coefficient for seawater with a salinity of 35 PSU at a temperature of 20 °C is $\beta = 257 \cdot 10^{-6} \, ^\circ C^{-1}$.

$$\rho = \rho_{ref} - \beta(T - T_{ref}) \quad (4.5.2)$$

4.6 Turbulent Prandtl number

The Prandtl number $Pr$ is defined as the ratio of momentum diffusivity to thermal diffusivity. For $Pr \ll 1$, thermal diffusivity dominates and for $Pr \gg 1$ momentum diffusivity dominates. For 20 °C seawater the Prandtl number is found to be 7.2.

$$Pr = \frac{\nu}{\alpha} = \frac{\text{viscous diffusion rate}}{\text{thermal diffusion rate}} = \frac{c_p \mu}{k} \quad (4.6.1)$$

where:

- $\nu$: Kinematic viscosity, $\nu = \mu/\rho, (m^2/s)$
- $\alpha$: Thermal diffusivity, $\alpha = k/(\rho c_p), (m^2/s)$
- $\mu$: Dynamic viscosity, $(Pa s = N s / m^2)$
- $k$: Thermal conductivity, $(W/(m K))$
- $c_p$: Specific heat, $(J/(kg K))$
- $\rho$: Density, $(kg/m^3)$

The turbulent Prandtl number is a non-dimensional term, which is defined as the ratio between the momentum eddy diffusivity and the heat transfer eddy diffusivity. It is used for solving heat transfer problems for turbulent flows. By setting a constant turbulent Prandtl number, the turbulent heat fluxes can be computed based on the turbulent eddy viscosity $\mu_t$, which the turbulence models predict.

The eddy diffusivity for momentum transfer $\epsilon_M$ and heat transfer $\epsilon_H$ are defined as [Hasan, 2007]:

$$-\overline{u'v'} = \epsilon_M \frac{\partial \overline{u}}{\partial y} \quad \text{and} \quad -\overline{v'T'} = \epsilon_H \frac{\partial \overline{T}}{\partial y} \quad (4.6.2)$$

Where $-\overline{u'v'}$ is the turbulent shear stress and $-\overline{v'T'}$ is the turbulent heat flux. The turbulent Prandtl number is then defined as:

$$Pr_t = \frac{\epsilon_M}{\epsilon_H} \quad (4.6.3)$$
In this thesis, the $k - \epsilon$ model is used for computations including buoyant flows induced by a temperature gradient. For these types of buoyant flows an generation term of $k$ due to buoyancy needs to be added in the turbulent kinetic energy transport equation [Versteeg and Malalasekera, 2007]. This is given by the term $P_b$, which includes the thermal expansion coefficient $\beta$, the component of gravitational vector in considered direction, the turbulent viscosity $\nu$ and the change of temperature $T$.

$$P_b = \beta g_i \mu_t \frac{\partial T}{\partial x_i} \tag{4.6.4}$$

Depending on the Prandtl number of the considered fluid, the $Pr_t$ ranges between 0.7 to 0.9 [Hasan, 2007]. From experimental data it is found that $Pr_t$ has an average value of 0.85. A sensitivity study of the turbulent Prandtl number in performed in appendix C.

### 4.7 Pressure

The total fluid pressure can be divided in to a hydrostatic part $p_h$ and a hydrodynamic (non-hydrostatic) part $p_d$. The hydrostatic pressure part is independent of the fluid acceleration, while the non-hydrostatic pressure $p_d$ is contributing to the total pressure due to the acceleration of the fluid. The hydrostatic pressures consist of several pressure components including the atmospheric pressure $p_{atm}$, the static pressure $p_0$, the barotropic pressure $p_{btr}$ and the baroclinic pressure $p_{bcl}$. However, atmospheric pressure and barotropic pressure can be assumed zero for the situation considered in this thesis.

$$p = p_h + p_d \tag{4.7.1}$$

The pressures are calculated for the fully vertical velocities and accelerations. The non-hydrostatic pressure approach fully solves the third equation in equation 4.4.1. The baroclinic pressure $p_{bcl}$ is the variation in pressure due to density differences compared to a reference density. This pressure component is the most important for considering buoyant jets. Due to this pressure component vertical acceleration take place for fluids with different densities [Zijl, 2002]. This causes the jet to deflect to the surface (positively buoyant) or to the bottom (negatively buoyant).
Chapter 5

Methods

5.1 OpenFOAM

The type of model used for this thesis is the CFD software package OpenFOAM [OpenFOAM-UserGuide, 2012]. CFD computations allow the user to model many different outfall configurations. OpenFOAM is a software package produced by OpenCFD Ltd and is maintained by The OpenFOAM Foundation, which is sponsored by the ESI Group, the owner of the trademark to the name OpenFOAM. OpenFOAM stands for Open Field Operation And Manipulation and is used in a wide range of scientific and engineering areas.

OpenFOAM is supplied with integrated pre- and post-processing environments and can also interact with other (open source) software packages. It is written in the C++ programming language and uses applications which are divided into solvers and utilities. The solver used in this thesis is the buoyantBoussinesqSimpleFoam solver, which is an advanced steady-state solver based on the SIMPLE algorithm for a single-phased model. This solver is specified for buoyant, turbulent flow of incompressible fluids [OpenFOAM-UserGuide, 2012]. Appendix A gives more information about OpenFOAM, the solver and the simple algorithm. One of the main disadvantages of OpenFOAM is that it does not have an integrated graphical user interface.

The computation of the jets in OpenFOAM are solved by computational fluid dynamics (CFD). This entails the solving of fluid flows with computer based computations. With a finite number of control volumes the Navier-Stokes equations are solved to finally obtain solutions for the total volume of the computational domain. In OpenFOAM, the finite volume method is used as a discretization scheme. At every control volume the center point is used together with the flux over the faces to compute the steady state or transient solution. The chosen numerical method in this thesis is the RANS approach. This approach needs a turbulence closure. Mortensen et al. [2013] and Aziz et al. [2008] concluded that a round turbulent buoyant jet can well be represented using the standard $k-\epsilon$ closure model. The standard $k-\epsilon$ model is a widely validated and used turbulence closure.

The domain is restricted to a square box around the FLNG cooling water outfall. This box is modeled with open boundaries for mass and heat. The open boundaries allow for a steady-state situation the numerical domain. In reality, the buoyant plume will spread further into the ambient waters outside the numerical domain. Figure 5.4 gives good illustration of the boundary of the numerical domain with respect to the scale of the buoyant plume.

Figures 5.1, 5.2 and 5.3 illustrate the physical mixing and transport processes found for the near-field jet mixing and far-field buoyant plume. The numerical domain is indicated by the dashed red line.
5.1.1 Initial and boundary conditions

To run a successful CFD simulation, the initial- and boundary conditions need to be chosen correctly. The results are obtained from a steady state solution, the final results for different initial conditions will be the same. However, a good setup of the initial conditions can reduce the computational time to reach the steady state situation. Boundary conditions affect the whole domain and simulation results. If the boundary condition of one of the quantities is chosen incorrect, the simulation will not produce incorrect results.

The numerical boundaries are given in figure 5.4. In OpenFOAM, the boundaries are separated into different types [OpenFOAM-UserGuide, 2012]. As a base type, the boundary is defined as a Wall or a Patch. The Patch-type boundary contains no geometric or topological information and is mostly used for inlets and outlets. The Wall-type is defined for wall boundaries and the application of turbulent wall-functions used by the turbulence models. For all transport quantities, pressure, turbulence properties and the turbulent kinematic viscosity the boundary conditions on all walls and patches have to be specified. The following types of boundary conditions are used in this thesis [OpenFOAM-UserGuide, 2012].

- **fixedValue** - Also known as the Dirichlet boundary condition, where the boundary defines a constant value \( \Phi \) at the specified boundary.
- **zeroGradient** - Also known as the Neumann boundary condition, where the normal gradient of \( \Phi \)
CHAPTER 5. METHODS

(a) Physical spreading of the buoyant plume  (b) Limited numerical domain of the buoyant plume

Figure 5.4: Illustration of numerical domain with respect to physical scale of the buoyant plume

at the boundary is zero. \( \frac{\partial \Phi}{\partial n} = 0 \). This boundary condition makes physical sense if the boundaries are far away from high gradients and quantities do not change through the boundary.

- **inletOutlet** - This boundary condition applies the zeroGradient condition on the boundary for outflow, when there is inflow the user should specify a fixedValue.

- **pressureInletOutletVelocity** - This is a combination of pressureInletVelocity and inletOutlet boundary condition. ZeroGradient is applied for outflow and for inflow the velocity is derived from the normal component of the internal cell. This boundary condition forms a combination with the pressure boundary condition totalPressure, they are self-stabilizing boundary conditions and needed to generate a well-posed problem.

- **totalPressure** - The total pressure \( p_0 = p + \frac{1}{2} \rho U^2 + \rho gz \) is fixed, when \( U \) changes, \( p \) is adjusted accordingly.

- **calculated** - Boundary value \( \Phi \) is derived by using other fields. Therefore, there is no need to specify a value for this boundary condition because it can calculate itself.

- **WallFunction** - An empirical function for obtaining suitable conditions near the wall, without the need of an extremely fine grid. Wall functions need to be adjusted to fit the Wall function to a certain quantity or turbulent coefficient (e.g. kqRWallFunction, epsilonWallFunction).

- Next to these boundary conditions there are two specific turbulent inlet boundary conditions for \( k \) and \( \epsilon \). Respectively turbulentIntensityKineticEnergyInlet, which needs and intensity percentage, and secondly the turbulentMixingLengthDissipationRateInlet, which needs a specified mixing length.

The boundary conditions of the input parameters are given in table 5.1. The turbulent boundary conditions are given in table 5.2. The value of \( k \), \( \epsilon \) and \( v_t \) at the inlet can be calculated with equation B.0.7, B.0.8 and B.0.2 in Appendix B. For the inlet turbulentIntensityKineticEnergyInlet this results in \( k = 0.05 \) with a turbulence intensity \( I = 0.06 \) [Versteeg and Malalasekera, 2007]. For the turbulentMixingLengthDissipationRateInlet, \( \epsilon = 0.07 \) and the eddy viscosity \( v_t = 5.5 \cdot 10^{-4} \) are used.

The turbulence parameters are chosen zeroGradient for all boundaries, expect the FLNG side. This side is defined as a wall, where the other sides are defined as patches. On these patches no turbulent wall functions can exist. Different boundary conditions are used between the validation study and the parameter study. For the validation study, all boundaries are chosen “open” and transport of heat and mass through all boundaries is possible. The parameter study includes the presence of the water surface and the bottom are incorporated. In this case, the bottom has a no-slip boundary condition and the surface boundary a slip condition. The heat flux through the surface is also chosen zero, this assumes an isolated wall at the surface, where no heat can leave or get in through the surface boundary. The sensitivity of the heat flux is discussed in appendix C.
CHAPTER 5. METHODS

<table>
<thead>
<tr>
<th>Boundary type</th>
<th>Boundary type</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet</td>
<td>FixedValue</td>
</tr>
<tr>
<td>FLNG</td>
<td>FixedValue</td>
</tr>
<tr>
<td>Bottom</td>
<td>Slit</td>
</tr>
<tr>
<td>Sides</td>
<td>PressureInletVelocity</td>
</tr>
<tr>
<td>Surface</td>
<td>Slit</td>
</tr>
<tr>
<td>Outlet</td>
<td>inletOutlet</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Boundary</th>
<th>Boundary type</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet</td>
<td>FixedValue</td>
</tr>
<tr>
<td>FLNG</td>
<td>FixedValue</td>
</tr>
<tr>
<td>Bottom</td>
<td>Slit</td>
</tr>
<tr>
<td>Sides</td>
<td>PressureInletVelocity</td>
</tr>
<tr>
<td>Surface</td>
<td>Slit</td>
</tr>
<tr>
<td>Outlet</td>
<td>inletOutlet</td>
</tr>
</tbody>
</table>

Table 5.1: Boundary condition types for variable input parameters

<table>
<thead>
<tr>
<th>Boundary</th>
<th>Boundary type</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet</td>
<td>turbulentIntensityKineticEnergyInlet</td>
</tr>
<tr>
<td>FLNG</td>
<td>kqRWallFunction</td>
</tr>
<tr>
<td>Bottom</td>
<td>zeroGradient</td>
</tr>
<tr>
<td>Sides</td>
<td>zeroGradient</td>
</tr>
<tr>
<td>Surface</td>
<td>zeroGradient</td>
</tr>
<tr>
<td>Outlet</td>
<td>zeroGradient</td>
</tr>
</tbody>
</table>

Table 5.2: Boundary condition types for turbulent input parameters

5.1.2 Discretization schemes

The discretization schemes used in the computations are given in the `fvSchemes` directory in OpenFOAM. This file is given in appendix G. The time discretization is a first order implicit method. The convective terms are calculated with a Gauss upwind scheme, this is a second order, unbounded scheme [OpenFOAM-UserGuide, 2012]. For the diffusion schemes a Gauss Linear Corrected discretization is applied. This is an unbounded, second order, conservative scheme. All the discretization schemes are default schemes for the buoyantBoussinesqSimpleFoam solver.

5.1.3 Courant number

The solver uses the SIMPLE algorithm. The SIMPLE algorithm uses implicit schemes for time discretisation. Large grid cells can be favourable regarding computational time, however a minimum grid size is also applicable for the stability related to the Courant number. For purely explicit schemes, the Courant number should not exceed the value of 1. Large Courant numbers are more favorable considering computational time. However, for numerical accuracy, it is recommended that the Courant number should not exceed 12. If very large Courant numbers are found in the simulation, unphysical numerical diffusion can occur. In equation 5.1.1 the expression for the courant number is given with $U$ the local velocity, $\Delta t$ the time step of the computation and $\Delta x$ the size of the grid cell.

$$C_0 = \frac{U \Delta t}{\Delta x}$$  \hspace{1cm} (5.1.1)
5.2 Computational mesh

For accurate numerical simulation, the quality of the grid is of great importance. Bad grid configurations can cause numerical diffusion, large truncation errors in the discretization schemes and can cause the model to become unstable. OpenFOAM offers different options and utilities to generate a computational grid (e.g. blockMesh, snappyHexMesh). For this thesis, the commercial software package ANSYS Designmodeler v.14.5 is used to generate a grid. The grid can be converted to a OpenFOAM format and the software offers a lot of freedom in the grid design process.

By using the RANS approach, the cell sizes can be larger than using LES or DNS. For the generation of grids, the user should make a consideration between computational costs and numerical accuracy. Therefore, it is wise to refine the grid in the areas of interest and areas with large gradients. Next to local refinements the user should pay attention to the orthogonality of adjacent cells and a smooth transition from very fine to coarse areas.

The cooling water outfall of the FLNG is defined as the inlet in the computational domain. Similar to the FLNG outfall the jet has a round structure. Preferably, the domain has as many squared cells as possible to increase the number of cells orthogonal to each other. However, the transition from the round jet structure to a squared grid introduces some non-orthogonal cell structures. Figure 5.5 shows the grid configuration for the jet inlet area. The inner circle is the inlet of the jet with a diameter $D_0 = 1\text{m}$.

The grid is developed for simulations with buoyancy effects. Therefore, the upper side of the jet has an extra refinement near the surface, see figure 5.6. The grid is refined in the all directions towards the center of the jet orifice.

![Figure 5.5: Grid configuration around the jet inlet](image)

![Figure 5.6: Side view of computational mesh, inlet on the left side](image)
5.2.1 Grid convergence test

A grid convergence test is performed in order to make the solution independent of the grid resolution. In general, the closer to zero the cell size becomes, the better accuracy will be achieved. However, this will result in unnecessarily long computational time and large data consumption.

The grid convergence test was performed with three different grid configurations. The total grid cells for Grid 1, Grid 2 and Grid 3 are respectively $N = 300 \cdot 10^3$, $N = 500 \cdot 10^3$ and $N = 900 \cdot 10^3$. These grids are referred to as N3, N5 and N9. To make a choice for the final grid configuration one should consider the accuracy of the solution and the computational time, which is directly related to the number of cells. Therefore, the coarsest grid that follows closely the converged solution is the most favorable one. For the three grids, the inlets are shown in figure 5.7a, 5.7b and 5.7c. The results of the grid convergence test are shown in section 6.1.

![Figure 5.7: Three different grid configurations for the grid convergence test](image)

5.3 Validation

The results from OpenFOAM need to be compared with experimental data sets. This will be done for both pure jets and buoyant jets. Several experimental datasets $^1$ are used to validate the model against physical laboratory results. The results of the validation study are presented in section 6.3.

5.3.1 Pure jet

From the theory in chapter 3 it is found that a pure jet development consists of two zones. The zone of flow establishment (ZFE) and the zone of established flow (ZEF). The ZEF starts at approximately $6 \times D_0$. From this point the jet is called fully developed. The validation study in OpenFOAM is performed with a domain of 30 meters deep, 50 meters wide and 50 meters long. The jet is located at a depth of 15 meter with $D_0 = 1$ and $U_0 = 3m/s$. Both the centerline velocity decay and the radial spreading of the jet will be examined. Besides the centerline velocity decay, also the radial spreading of the jet velocity is examined. From section 3.3.1 and figure 3.4 it became clear that in the zone of established flow, the normalized velocity profile proves to be self-similar and fits well to a Gaussian distribution. To compare the data, the velocity and spreading radius are made dimensionless.

5.3.2 Buoyant jet

In addition to the pure jet, also a validation study on the buoyant jet is performed. An initial density difference is introduced by increasing the temperature of the jet relative to the ambient water temperature.

$^1$From large database used to benchmark the jet model CORMIX, http://www.mixzon.com/benchmark
The background water temperature has a constant value of $T = 20^\circ C$. Two jets with a temperature of respectively $T = 40^\circ C$ ($\Delta T_0 = 20^\circ C$) and $T = 70^\circ C$ ($\Delta T_0 = 50^\circ C$) are studied with an initial velocity of $U_0 = 3m/s$ and diameter $D_0 = 1m$.

Both the centerline trajectory and the centerline dilution of the jet will be validated. The centerline dilution is represented by the decay of centerline temperature, which is a measure to examine the diffusion of the initial temperature. The centerline dilution is given by $S_c$, which is given by equation 5.3.1.

$$S_c = \frac{\Delta T_0}{\Delta T_c} = \frac{(T_0 - T_a)}{(T_c - T_a)}$$  \hspace{1cm} (5.3.1)

**Thermal expansion coefficient $\beta$**

The amount the density changes due to temperature differences depends on the salinity of the water. However, the salinity cannot be included in the OpenFOAM solver. By looking at the relative densities for the equation of state by Gill [1982] and the relative densities for the Boussinesq approximation the salinity can be implicitly be included in OpenFOAM. By taking the thermal expansion coefficient $\beta$ as a variable, the equation of state by Gill [1982] with $S=35$ PSU can be approximated. This can be done by varying $\beta$ as a function of temperature $\beta = \beta(T)$, or by taking a representative constant $\beta$ value. The last approach is chosen, because no additional programming in OpenFOAM is required. Figure 5.8 shows the result for different constant thermal expansion coefficients $\beta$.

Table 5.3 gives an overview of the density of seawater for different temperatures. Next to that it provides an overview of the most realistic relative density from Gill [1982] and for the Boussinesq approximation with $\beta = 257 \cdot 10^{-6} ^\circ C^{-1}$. To provide a good representation of the saline water relative densities, the $\beta$ should be corrected to follow the relative densities from [Gill, 1982] as good as possible.

![Figure 5.8: Relative density change Boussinesq approximation and [Gill, 1982], with $S = 35$ PSU and $T_{ref} = 20^\circ C$](image)

Figure 5.8 gives an indication which thermal expansion coefficient represents the correct relative density the best. This depends on the temperature range. For the jet with $T = 40^\circ C$, $\beta = 300e^{-6} ^\circ C^{-1}$ is used. For the jet with $T = 70^\circ C$, $\beta = 350e^{-6} ^\circ C^{-1}$ gives the overall best representation. For both cases, the $\beta$ underestimates the value for the highest temperatures. However, this region is mostly dominated by the jet momentum. When buoyancy effects become the dominant process, the jet temperature dropped to a value where the chosen $\beta$’s give a good approximation of the relative densities.
CHAPTER 5. METHODS

Table 5.3: Seawater (S=35 PSU) densities for varying temperatures according to [Gill, 1982] equation of state and the Boussinesq approximation with $T_{ref} = 20^\circ C$

<table>
<thead>
<tr>
<th>T ($^\circ C$)</th>
<th>Seawater density Gill (kg/L)</th>
<th>$\rho/\rho_{ref}$ Gill, S=35 PSU</th>
<th>$\beta = 257 \cdot 10^{-6} ^\circ C^{-1}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>20</td>
<td>1.0248</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>25</td>
<td>1.0233</td>
<td>0.9986</td>
<td>0.9987</td>
</tr>
<tr>
<td>30</td>
<td>1.0217</td>
<td>0.9970</td>
<td>0.9975</td>
</tr>
<tr>
<td>35</td>
<td>1.0199</td>
<td>0.9953</td>
<td>0.9962</td>
</tr>
<tr>
<td>40</td>
<td>1.0180</td>
<td>0.9934</td>
<td>0.9950</td>
</tr>
<tr>
<td>45</td>
<td>1.0159</td>
<td>0.9913</td>
<td>0.9937</td>
</tr>
<tr>
<td>50</td>
<td>1.0136</td>
<td>0.9891</td>
<td>0.9825</td>
</tr>
<tr>
<td>55</td>
<td>1.0113</td>
<td>0.9869</td>
<td>0.9812</td>
</tr>
<tr>
<td>60</td>
<td>1.0089</td>
<td>0.9846</td>
<td>0.9900</td>
</tr>
<tr>
<td>65</td>
<td>1.0066</td>
<td>0.9822</td>
<td>0.9887</td>
</tr>
<tr>
<td>70</td>
<td>1.0043</td>
<td>0.9800</td>
<td>0.9875</td>
</tr>
</tbody>
</table>

5.4 Outfall parameter sensitivity study

Floating LNG cooling water outfalls can be designed in many different configurations with varying outflow conditions. The variables are given by the outflow velocity $U_0$, the outflow temperature difference $\Delta T_0$, the outfall depth $d$, outfall diameter $D$, and the outfall angle $\alpha$. The objective is to find the most important parameters influencing the ambient temperature and to find relations between the governing parameters to determine the temperature increase.

By varying the outfall velocity, diameter and temperature, the outflow quantities change. This involves changes in discharge volumes, amount of heat and momentum of the jet. The amount of heat can be calculated using the specific enthalpy of water. The specific enthalpy of a fluid entails the amount of energy per unit mass, (kJ/kg) for a given temperature. To increase the temperature of 1 kg of seawater with 1 $^\circ C$, an energy input of 4.18 kJ is needed. Using the specific enthalpy of water makes it possible to calculate the amount of discharged heat at the outfall. The discharged heat is indicated with $kJ/s$. The amount of discharged heat is governed by the discharge rate, $Q$ and the temperature of the outfall with respect to its ambient water, $\Delta T_0$.

Table 5.4 provides an overview of the model input parameters for the different outfall scenarios. The OpenFOAM plots belonging to these scenarios are shown in figure 7.3. The plots give an indication of the temperature distributions, jet trajectories and outflow magnitude. Different color scaling is used, therefore it is not possible to visually compare the temperature magnitudes in these plots. For every scenario, the ambient water temperature has a constant value of $T_a = 20^\circ C$. All the simulations show the steady state solutions.

5.5 Dimensional analysis

In engineering and science, the approach of dimensional analysis is often used. This analysis involves finding relationships between different physical quantities by identifying their fundamental dimensions [Sonin, 2001]. The fundamental dimensions relevant for this thesis are length, mass, time and temperature.

Dimensional analysis is a powerful tool to find physical relations between different dimensionless groups of parameters. It is used to compare and relate experimental data with different initial conditions. This is used with the validation of the numerical model to the available experimental data sets. The data used for the dimensionless analysis can be obtained by measurements, physical model tests or numerical model simulations.

The focus of this section will be on the formation of various dimensionless groups of parameters. These groups of parameters can be used in chapter 7 to find relations between the different dimensionless
groups. The relations that exist can be used to form an expression of the governing parameters, which can eventually predict the temperature rise, \( T_r \), as a function of the outfall parameters. The independence of the dimensional parameters to the system of units corresponds to a scale-invariance of the model. This could possible lead to a wider implementation of the results.

\[
T_r = f(U_0, \Delta T_0, D, \alpha, d, X) \tag{5.5.1}
\]

### 5.5.1 Dimensionless parameters

In the past, an extensive amount of studies have been performed on jets. In many of these studies the densimetric Froude number \( (Fr_d) \) proved to be an important dimensionless parameter. The value of the densimetric Froude number gives an indication of the magnitude of jet inertia over jet buoyancy. In addition, the densimetric Froude number allows to scale and compare different model results. The densimetric Froude number is given by equation 3.4.1. For the FLNG cooling water outfalls, the initial density differences are introduced by a temperature difference. By applying the Boussinesq approximation (equation 4.5.2), as used in the numerical model, equation 3.4.1 can be written as equation 5.5.2. The result of applying the Boussinesq approximation to the momentum length scale \( L_M \) is similar (equation 5.5.3).

\[
Fr_d = \frac{U_0}{\sqrt{(\beta \Delta T_0)gD}} \tag{5.5.2}
\]
\[ L_M = \frac{U_0(\pi D^2_0/4)^{1/4}}{\sqrt{\beta \Delta T}} g \]  

(5.5.3)

The momentum length scale includes the initial outfall velocity \( U_0 \), the initial jet temperature difference \( \Delta T_0 \), and the outfall diameter \( D \). These parameters are expected to play a significant role in the surface temperature distribution. To develop a feeling of the magnitude of the densimetric Froude number and the momentum length scale, the values are calculated for the scenarios in table 5.4. The results are given in table 5.5.

<table>
<thead>
<tr>
<th>Run</th>
<th>( U_0 ) [m/s]</th>
<th>( \Delta T_0 ) [°C]</th>
<th>( D ) [m]</th>
<th>( Fr_d ) [-]</th>
<th>( L_M ) [m]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>10</td>
<td>2.5</td>
<td>3.7</td>
<td>8.7</td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>10</td>
<td>2.5</td>
<td>7.4</td>
<td>17.4</td>
</tr>
<tr>
<td>3</td>
<td>3</td>
<td>10</td>
<td>2.5</td>
<td>11.1</td>
<td>26.0</td>
</tr>
<tr>
<td>4</td>
<td>4</td>
<td>10</td>
<td>2.5</td>
<td>14.7</td>
<td>34.7</td>
</tr>
<tr>
<td>5</td>
<td>5</td>
<td>10</td>
<td>2.5</td>
<td>18.4</td>
<td>43.4</td>
</tr>
<tr>
<td>6</td>
<td>3</td>
<td>10</td>
<td>2.5</td>
<td>11.1</td>
<td>26.0</td>
</tr>
<tr>
<td>7</td>
<td>3</td>
<td>10</td>
<td>2.5</td>
<td>11.1</td>
<td>26.0</td>
</tr>
<tr>
<td>8</td>
<td>3</td>
<td>10</td>
<td>2.5</td>
<td>11.1</td>
<td>26.0</td>
</tr>
<tr>
<td>9</td>
<td>3</td>
<td>10</td>
<td>2.5</td>
<td>11.1</td>
<td>26.0</td>
</tr>
<tr>
<td>10</td>
<td>3</td>
<td>5</td>
<td>2.5</td>
<td>15.6</td>
<td>36.8</td>
</tr>
<tr>
<td>11</td>
<td>3</td>
<td>7</td>
<td>2.5</td>
<td>13.2</td>
<td>31.1</td>
</tr>
<tr>
<td>12</td>
<td>3</td>
<td>15</td>
<td>2.5</td>
<td>9.0</td>
<td>21.3</td>
</tr>
<tr>
<td>13</td>
<td>3</td>
<td>20</td>
<td>2.5</td>
<td>7.8</td>
<td>18.4</td>
</tr>
<tr>
<td>14</td>
<td>3</td>
<td>10</td>
<td>1</td>
<td>17.5</td>
<td>16.5</td>
</tr>
<tr>
<td>15</td>
<td>3</td>
<td>10</td>
<td>1.5</td>
<td>14.3</td>
<td>20.2</td>
</tr>
<tr>
<td>16</td>
<td>3</td>
<td>10</td>
<td>2</td>
<td>12.4</td>
<td>23.3</td>
</tr>
<tr>
<td>17</td>
<td>3</td>
<td>10</td>
<td>2.5</td>
<td>11.1</td>
<td>26.0</td>
</tr>
<tr>
<td>18</td>
<td>6</td>
<td>5</td>
<td>2.5</td>
<td>31.3</td>
<td>73.6</td>
</tr>
<tr>
<td>19</td>
<td>4.3</td>
<td>7</td>
<td>2.5</td>
<td>18.9</td>
<td>44.4</td>
</tr>
<tr>
<td>20</td>
<td>2</td>
<td>15</td>
<td>2.5</td>
<td>6.0</td>
<td>14.2</td>
</tr>
<tr>
<td>21</td>
<td>2</td>
<td>10</td>
<td>2.5</td>
<td>7.4</td>
<td>17.4</td>
</tr>
<tr>
<td>22</td>
<td>2</td>
<td>10</td>
<td>2.5</td>
<td>7.4</td>
<td>17.4</td>
</tr>
<tr>
<td>23</td>
<td>2</td>
<td>10</td>
<td>2.5</td>
<td>7.4</td>
<td>17.4</td>
</tr>
<tr>
<td>24</td>
<td>2</td>
<td>10</td>
<td>2.5</td>
<td>7.4</td>
<td>17.4</td>
</tr>
</tbody>
</table>

Table 5.5: Base scenarios with corresponding densimetric Froude number and momentum length scale

### 5.5.2 Dimensionless groups

A new method is introduced by forming dimensionless groups of the characteristic outfall parameters of the jet. All the parameters involved in the cooling water outfall need to become dimensionless to perform a dimensional analysis. This includes the OpenFOAM temperature results. The surface temperature rise \( T_r \), obtained from OpenFOAM, is divided by the initial jet temperature difference \( \Delta T_0 \). This leads to the relative dimensionless temperature rise:

\[ T_r = \frac{T_r}{\Delta T_0} \]  

(5.5.4)

The momentum length scale \( L_M \), has unity [m]. To form a dimensionless group, \( L_M \) needs to be divided with the same base dimension [m]. \( L_M \) is a length scale in the flow direction of the jet. If \( X \), in the
direction of the jet, is used to form a dimensionless group, this results in the dimensionless group 5.5.5. This dimensionless parameter gives a measure for the distance from the jet in terms of $L_M$.

$$\left( \frac{L_M}{X} \right)$$

(5.5.5)

A dimensionless reduction factor needs to be found, which results in the decrease of $T_r$ for increasing $(X)$. The new dimensionless group is chosen to be $(X/D)$, which gives an indication of the distance in terms of jet diameters.

$$\left( \frac{X}{D} \right)$$

(5.5.6)

The depth of the outfall is included in the relative submergence of the jet. This is given as the ratio of outfall depth over outfall diameter. The relative submergence is also used to examine the classification of submergence and the possible deflection of the jet because of the Coanda effect (section 3.5). The dimensionless group of the relative submergence is given by:

$$\left( \frac{d}{D} \right)$$

(5.5.7)

All the parameters from table 5.4 are included in the dimensionless groups. In chapter 7, the influence of the individual parameters are examined. Equation 5.5.8 gives a general formulation of the dimensionless temperature rise as a function of the dimensionless groups.

$$T_r = f\left( \frac{L_M}{X}, \frac{X}{D}, \frac{d}{D}, \alpha \right)$$

(5.5.8)
Chapter 6

Results - Model performance

6.1 Grid convergence test

The grid convergence test is performed in order to make the solution independent of the grid resolution. The simulated results are from a buoyant jet with $\Delta T = 50^\circ C$. The grid convergence is studied at the point where buoyancy begins to take over from the jet momentum. This location captures both the jet momentum and the buoyancy effects. The convergence is examined for the temperature, velocity and turbulent kinetic energy. The results are given in figure 6.1, 6.2 and 6.3. From the previous mentioned criteria it is concluded that the grid configuration N5, with $500 \cdot 10^3$ grid cells is the most favorable one regarding accuracy and computational costs.

![Graph showing temperature distribution](image)

*Figure 6.1: T for N3, N5 and N9*
CHAPTER 6. RESULTS - MODEL PERFORMANCE

6.2 Qualitative model results

To examine if the model correctly represents the physical jet behaviors, the three dimensional current patterns, velocities and temperature contours are plotted. In figure 6.4 and figure 6.5 a vector plot of the current direction is given. Both the direction and the magnitude of the current are presented. The entrainment behavior of the jet is in agreement with the streamlines around the jet in figure 2.3. Both results imply that the physics of the jet and the boundary conditions are well represented in the model.

Figure 6.6 shows the results of a buoyant jet with a contour line of $\Delta T = 2^\circ C$. Every point within this plume has a temperature rise larger than $2^\circ C$. The jet has an initial outflow velocity $U_0 = 2 m/s$ and initial temperature difference $\Delta T_0 = 10^\circ C$. The temperature contour profiles can provide important insight in the distribution of the temperature plume. From this results it is visible that the buoyant jet does not penetrate the surface and deflects to the sides around the point of surface impingement. This is also what is expected from the boundary conditions and reality.

Figure 6.7 shows the results of the temperature contour profiles for $\Delta T = 1^\circ C$, $\Delta T = 2^\circ C$ and $\Delta T = 3^\circ C$. These are represented by respectively the green, orange and purple colors. From the contour line of $\Delta T = 1^\circ C$ it becomes clear that, after surface impingement, the temperature spreads over the water column. This is similar to the theoretical behavior of passive ambient diffusion processes in figure 3.9. From these results it is concluded that the results show physically correct behaviors based on the qualitative model performance.
CHAPTER 6. RESULTS - MODEL PERFORMANCE

Figure 6.4: OpenFOAM vector plot of the flow directions, side view

Figure 6.5: OpenFOAM vector plot of the flow directions, back view

Figure 6.6: OpenFOAM temperature contour of $\Delta T = 2^\circ C$
6.3 Validation

In this section the results of OpenFOAM are compared with experimental data and analytical solutions. The method to perform this validation study has already been discussed in section 5.3.

6.3.1 Pure jet

Figure 6.8 shows the steady state situation for this jet configuration. Boundary conditions are chosen to allow outflow through all walls, except the wall of the inlet. Figure 6.10 shows the absolute velocity decay for different distances from the jet.

The decay of the centerline velocity for a pure jet has been studied extensively. The experimental data are all in good agreement with the theory. Figure 6.9 shows experimental results from Albertson et al. [1950], Labus et al. [1972] and Rosler and Bankoff [1963] together with the results obtained with OpenFOAM. A clear distinction between the zone of flow establishment and zone of established flow is visible. The results regarding centerline velocity decay are in good agreement with the various experimental data sets. Furthermore, OpenFOAM gives a good representation of both jet stages. This implicitly means that the growing turbulent shear layer is represented very well in OpenFOAM.
Besides the centerline velocity decay, also the radial spreading is examined. Figure 6.10 gives the absolute values of the velocity profiles for several distances from the jet. In figure 6.11 the axis values are made dimensionless. The X-axis represents the radial spreading \( r \) divided by the velocity half width \( b_y \), which is the distance where the local velocity becomes \( e^{-1} U_m \). The Y-axis represents the local velocity divided by the maximum velocity along the cross section. The spreading of the jet in OpenFOAM in figure 6.11 is in perfect agreement with the theory and the experimental results in figure 3.4.

![Figure 6.9: Centerline velocity decay](image1)

![Figure 6.10: Velocity profiles across the jet](image2)

![Figure 6.11: Dimensionless radial velocity profiles](image3)
6.3.2 Buoyant jet

In this section the results of the validation study for a buoyant jet are presented. The $\beta$ value of the thermal expansion coefficient influences the jet trajectories. Figure 6.12 and 6.13 show the centerline trajectory for respectively $\Delta T = 20^\circ C$ and $\Delta T = 50^\circ C$. The absolute difference between the corrected and original thermal expansion coefficient $\beta$ for both $\Delta T = 20^\circ C$ and $\Delta T = 50^\circ C$ are given in figure 6.14. The lines represent the centerline trajectory of the jets.

![Figure 6.12: Buoyant jet trajectory for $\Delta T = 20^\circ C$](image)

Several datasets are available for inclined horizontal buoyant jets. Figure 6.15 and 6.16 show the normalized vertical trajectory $Z/L_M$ as a function of $X/L_M$ for the OpenFOAM results and the experimental data sets. Visual observations are used to determine the trajectories, resulting in the data scatter for these types of experiments.

The normalized jet trajectories for both $\Delta T = 20^\circ C$ and $\Delta T = 50^\circ C$ are in good agreement with the experimental data. Due to the different momentum length scales $L_M$, the OpenFOAM results do not have the same length. For both cases it can be concluded that OpenFOAM produces realistic results.

Figure 6.17 gives the results from OpenFOAM for both $\Delta T = 20^\circ C$ and $\Delta T = 50^\circ C$ against experimental data. The model results are in good agreement with the experimental data up to $Z/L_M = 1.5$. 

48
Figure 6.14: Buoyant jet trajectory for $\Delta T = 20^\circ C$ and $\Delta T = 50^\circ C$ with and without corrected $\beta$

The range for this thesis is $Z/L_M < 1.5$. Within this range, the model results are in good agreement with the data.

Conclusion - Using the model

This chapter provided the results of the validation of the OpenFOAM model with the chosen setup and boundary conditions, against experimental datasets. For both the pure jet and the buoyant jet, the model proved to be in good agreement with the experimental data.
CHAPTER 6. RESULTS - MODEL PERFORMANCE

Figure 6.15: Normalized jet trajectory for $\Delta T = 20^\circ C$

Figure 6.16: Normalized jet trajectory for $\Delta T = 50^\circ C$

Figure 6.17: Normalized centerline dilution for $\Delta T = 20^\circ C$ and $\Delta T = 50^\circ C$
Chapter 7

Results - Outfall simulations

7.1 Results - Outfall scenarios

The scenarios used for this study are given in table 5.4. Every outfall configuration results in a different temperature distribution in the domain. At some point from the jet orifice, the jet will interact with the surface. This point of surface impingement is very much depending on the initial flow and design conditions. At the point of surface impingement, the surface temperature is maximum (figure 7.1). The result of the surface temperature profiles of the outfall scenarios are given in figure 7.4. The location of the maximum temperature rise restriction, governed by the World Bank, are in general located after the point of surface impingement.

![Figure 7.1: Surface temperature related to point of surface impingement](image)

It is difficult to distinguish the temperature transport processes by only considering the surface temperature results. In addition, figure 7.5 gives the result of the temperature depth profile. This gives more information regarding the distribution of temperature over the water column. These results are supported with the velocity profiles over the water column in figure 7.6. The velocity profiles can be used to give an indication of the importance of the currents induced by initial jet momentum on the temperature transport.

Both the temperature and the velocity profiles are given for a distance $X = 100m$ from the source. Appendix E gives the temperature and velocity results for $X = 50m$. For the depth profiles, only the results of the minimum and maximum value of the parameter are given. In these plots the temperatures...
are given in Kelvin, where $T = 293.15^\circ K = 20^\circ C$. Combining the results of three figures can give an indication of the governing processes and the influence of that particular outfall parameter.

**Outfall velocity - $U_0$**

Figures 7.4a, 7.5a, 7.6a show the results for varying outflow velocities. The jet with $U = 1 m/s$ has the least amount of jet momentum. The point of surface impingement is close to the jet orifice, resulting in a high peak temperature. Due to the limited amount of jet momentum, advective transport to the far-field is negligible. The spreading of the temperature is dominated by diffusion processes and buoyant spreading. This results in round spatial spreading of the plume at the surface. The jet with $U = 5 m/s$ has the largest amount of discharge volumes, momentum and heat. The flow pattern is strongly influenced by the initial jet momentum. The surface peak temperature is significantly lower and located further from the jet orifice. This is the result of the large outflow velocity, resulting in large entrainment rates. The entrainment of ambient fluid is depending on the jet centerline velocity (equation 3.6.2) and is illustrated in figure 7.2a.

For $U_0 = 5 m/s$, a surface current of $0.8 m/s$ is found at $X = 100 m$. This results in large advective transports of temperature in the upper layer. As an additional effect, the velocity profile shows a steep gradient over the water column. This introduces shear between two layers of different velocities. The mixing process between the upper part and the lower part of the water column is illustrated in figure 7.2b. As a consequence the temperature spreads over the whole water column. This is clearly visible in figure 7.5a. The heat is less concentrated at the surface, but is mixed throughout the whole water column.

**Outfall depth - $d$**

Figures 7.4b, 7.5b and 7.6b give the result for outfall depths of $d = 3 m$ to $d = 15 m$. The difference in surface temperatures are substantial for distances smaller than $X < 60 m$. For deeper outfall depths, the trajectories become longer. Therefore, the point of jet impingement is located at a further distance for deeper jet locations. For distances larger than $X > 60 m$, the temperature differences for varying outfall depths are small. Surface diffusion and turbulent jet diffusion are considered equal for $d = 3 m$ and $d = 15 m$. Varying the jet outfall depths do not result in large surface temperature differences in the far-field region. For $X = 100 m$, also the current patterns look similar. The transport of temperature

---

**Figure 7.2: Momentum dominated mixing processes**

(a) Turbulent jet entrainment, induced by turbulent shear and low pressures in the center of the jet

(b) Vertical mixing induced by turbulent shear due to vertical velocity gradients
is similar for both cases. It becomes clear that for \( d = 15 \text{m} \), the amount of heat lost due to a longer distance through the ambient water is minimal.

**Outfall diameter - \( D_0 \)**

Figures 7.4c, 7.5c and 7.6c give the results for varying outfall diameters, \( D \). The discharge is related to the diameter by \( Q \sim D^2 \). By adjusting the diameter, both the discharge and the amount of heat input changes to the power two. However, the surface temperature does not increase by a power two. The relation of surface temperature rise for increasing outfall diameters is approximately linear. This is the result of the larger mixing surface for larger outfall diameters. The entrainment of ambient fluid is increased, decreasing the rise of surface temperature. The outflow velocities are constant, however the discharged volume of water is 6.25 times higher for \( D = 2.5 \text{m} \). The same factor applies to the jet momentum. Due to the large momentum, the velocity at \( X = 100 \text{m} \) is larger for \( D = 2.5 \text{m} \). The temperature in the lower part is significantly higher for \( D = 2.5 \text{m} \), this is induced by the larger velocity gradient over depth and the larger amount of heat input, enhancing the diffusion.

**Outfall temperature - \( \Delta T_0 \)**

Figures 7.4d, 7.5d and 7.6d give the results for varying outfall temperatures, \( \Delta T_0 \). The amount of heat added to the ambient environment depends on the discharge and the outfall temperature. From the properties of the specific enthalpy of water it is found that there is a linear relation between the initial jet temperature and the amount of discharged energy (heat). This is also visible in the surface temperature results. Increasing the amount of discharged heat by a factor two, increases the surface temperature by approximately a factor two. Increasing the initial temperature has a significant influence on the surface temperature. Increasing the outfall temperature has also a large contribution to the total increase of the water column temperature. This is the result of the passive diffusion governed by the high temperature gradients. The velocity profiles over depth are similar. However, the velocity for small temperature differences is larger because of the smaller buoyancy effects. The amount of heat input is more important than a larger gradient in velocity.

**Constant energy input**

Figures 7.4e, 7.5e and 7.6e show the results for a constant energy outflow from the jet. The discharge and temperature difference are both adjusted to obtain a constant energy output for different outflow velocities. From figure 7.4e it becomes clear that a low outfall velocity, combined with a high temperature difference results in the largest increase in surface temperature. The entrainment for this low velocity jet is less, resulting in less mixing with the ambient water. Diffusive transport, which is dominated by the magnitude of the temperature gradient between two volumes of water, is the dominant transport process. The temperature in the whole water column is higher for \( U_0 = 2 \text{m/s} \) with \( \Delta T_0 = 20 \text{°C} \), while it has significantly lower velocity magnitudes and gradients.

**Outfall angle - \( \alpha \)**

Figures 7.4f, 7.5f and 7.6f show the result of the effect of different outfall angles. Positive angles indicate a downward directed jet. Increasing the outfall angle results in a decrease of surface temperature. Increasing the outfall angle, results in a decay of temperature at a distance \( X = 100 \text{m} \). The surface velocity for \( \alpha = 0 \text{°} \) is higher, which advects the temperature to \( X = 100 \text{m} \) in a shorter amount of time. For \( \alpha = 30 \text{°} \), the temperature diffuses more effectively in the surrounding ambient water. The jets with an positive angle travel a longer distance through the ambient water. This increases the surface contact with the ambient water significantly, causing the surface temperature to decrease. Discharging under an angle proves to be an effective method to decrease the surface temperature rise, without significantly increasing the temperature of the water column at \( X = 100 \text{m} \). Most of the heat in the water column is concentrated in the area \( X < 70 \text{m} \).
From the results in figure 7.7 it is concluded that the temperature distribution of a jet with angle $\alpha$, can be approximated by multiplying the surface temperature rise with a factor $\cos(\alpha)$. The resulting value of $\cos(\alpha)$ functions as a reduction factor, based on the outfall angle. The reduction factor $\cos(\alpha)$ proves to be valid for different initial jet conditions.

### 7.1.1 Conclusion

From the results of the parameter study on the ambient temperature increase it is concluded that the governing parameters consist of the outflow velocity $U_0$, outflow temperature difference $\Delta T_0$, outfall diameter $D$ and outfall angle $\alpha$. The outfall depth, $d$ has hardly any influence in the region where the environmental assessments take place. Therefore, the dimensionless group including the outfall depth is not taken into account for the equation development. This results in the new general formulation of the temperature rise as a function of the dimensionless groups given in equation 7.1.1. Advection plays an important role in the transport of temperature. High velocity gradients over the water depth induce turbulent shear mixing between layers of water with different velocities. High momentum outfalls, which introduce large surface currents, do not transport temperature most effectively. The turbulent diffusive transport, governed by the large vertical velocity gradients is the most dominant transport process for this steady state situation.

$$T_r = f\left(\frac{L_M}{X}, \frac{X}{D}, \alpha\right)$$  \hspace{1cm} (7.1.1)
CHAPTER 7. RESULTS - OUTFALL SIMULATIONS

Figure 7.3: Visualization of the jet trajectories for different outfall scenarios. The temperature plots for the minimum and maximum value of the variable are shown. Every plot has its own temperature scale. \( \Delta T_0 \) is indicated by the red color. The value of \( \Delta T_0 \) is different for Figures (e), (f), (i) and (j). All other plots have \( \Delta T_0 = 10^\circ C \).
Figure 7.4: Visualization of the OpenFOAM results for the surface temperature rise $T_r$. Each plot shows the variation of the surface temperature rise for different initial jet conditions. The magnitude and the location of surface impingement are governed by the initial jet conditions.
CHAPTER 7. RESULTS - OUTFALL SIMULATIONS

(a) $U_0 = 1 \text{ m/s}$, $U_0 = 5 \text{ m/s}$

(b) $d = 3 \text{ m}$, $d = 15 \text{ m}$

(c) $D = 1 \text{ m}$, $D = 2.5 \text{ m}$

(d) $\Delta T_0 = 5 ^\circ \text{C}$, $\Delta T_0 = 20 ^\circ \text{C}$

(e) $U_0 = 6 \text{ m/s}$ with $\Delta T_0 = 5 ^\circ \text{C}$, $U_0 = 2 \text{ m/s}$ with $\Delta T_0 = 20 ^\circ \text{C}$

(f) $\alpha = 0 ^\circ$, $\alpha = 30 ^\circ$

*Figure 7.5: Temperature profile over depth for different outfall design scenarios*
Figure 7.6: Velocity profile over depth for different outfall design scenarios.
7.2 Results - Dimensional analysis

In the previous section it is found that the outfall depth does not play a significant role in the surface temperature rise. Therefore, the dimensionless group including the outfall depth can be neglected. The following dimensionless groups of parameters need to be used in the dimensional analysis.

\[ \overline{T_r} = \frac{T_r}{\Delta T_0} \quad (7.2.1) \]

\[ \left( \frac{L_M}{X} \right) \quad (7.2.2) \]

\[ \left( \frac{X}{D} \right) \quad (7.2.3) \]

Figure 7.8a gives the relative temperature rise \( \overline{T_r} \), as a function of the momentum length scale \( L_M \). The figure shows the results for the distances \( X = 50m \), \( X = 80m \) and \( X = 100m \). Figure 7.8a includes the results for varying \( U_0 \) (runs 1-5), \( \Delta T_0 \) (runs 10-13) and constant energy input (runs 18-20). The corresponding values of the momentum length scale can be found in table 5.5. For high values of \( L_M \), the value of \( \overline{T_r} \) becomes constant. This implies that for an increasing jet momentum, while keeping \( \Delta T_0 \) constant, the temperature does not further increase. Discharging a larger volume of water does not necessarily lead to an increase of \( \overline{T_r} \).

For \( X = 50m \), the shape of the solution is different than the shape at \( X = 80m \) and \( X = 100m \). For \( X = 50m \), a decrease of \( \overline{T_r} \) is visible for values of \( L_M > 25m \). With the results in figure 7.4a, the shape of this solution can be explained. For distances \( X > 60m \), the largest momentum jet results in the highest temperature increase. For \( X < 40m \), the largest jet momentum results in the lowest relative temperature rise. For jets with a large momentum length scale, the point of surface impingement is located at a larger distance from the jet orifice (figure 7.1). The high momentum jets have not yet reached their maximum surface temperature for distances close to the jet. Therefore, a decay of \( \overline{T_r} \) is visible at \( X = 50m \) for large momentum length scales. The effects becomes stronger for decreasing distances.
Figure 7.8b shows the result for the distances $X = 50m$, $X = 80m$ and $X = 100m$. The same relations of $T_r$ as a function of $U_0$, $\Delta T_0$ and a constant heat input exist. The next step is to obtain a relation of $T_r$, independent of the distance $X$. It is found that the temperature rise $T_r$ is decreasing over the distance by a factor $(X/D)^{-0.72}$. Figure 7.8c shows the result of this approach for the scenarios (1-20).

For increasing values of $L_M/X$, the result in figure 7.8c goes to a constant value for $X > 60m$. The value of the horizontal asymptote does not have a physical meaning, but is a direct results of the amplification factor $(X/D)^{0.72}$. It is assumed that for $(L_M/X) > 0.7$, the relative temperature rise, $T_r$ stays constant. This is true for a wide range of momentum length scales and distances. However, for very high momentum length scales in combination with distances close to the jet, this will introduce some incorrect results.
Figure 7.8: Relations between dimensionless groups of parameters for the distances $X = 50\text{m}$, $X = 80\text{m}$ and $X = 100\text{m}$
7.2.1 Empirical relation

The characteristics of the relation visible in figure 7.8c are similar to other physical diffusion processes. These called 'S' curved functions are often found in diffusion processes where the growth of a concentration or temperature slowly stops for increasing time or space. In this case, the relative temperature rise is depending on the magnitude of the momentum length scale. The standard logistic function is given by equation 7.2.4 and represented by the S-curve in figure 7.9.

\[ f(x) = \frac{1}{1 + e^{-x}} \quad (7.2.4) \]

The maximum asymptotic value of the logistic function is governed by the numerator of equation 7.2.4. The steepness of the S-curve and the position on the x-axis are governed by the power function of \( e \). The standard logistic function is modified to fit the OpenFOAM results. This results in equation 7.2.5. Figure 7.10 shows both the modified standard logistic function together with the OpenFOAM results.

As expected, the logistic function and the OpenFOAM results for very high momentum length scales in combination with low values of \( X \) do not fit very well.

Equation 7.2.5 can be rewritten in terms of the surface temperature rise as a function of the outfall parameters, \( T_r = f(X, U_0, \Delta T_0, D, \alpha) \). In section 7.1 it is found that the surface temperature of a jet reduces by a factor \( \cos(\alpha) \) for discharging under an angle. Combining the results will lead to equation 7.2.6.

\[ \frac{T_r}{\Delta T_0} \left( \frac{X}{D} \right)^{0.72} = \frac{3}{1 + e^{-11(L_M/X)}} \]

\[ T_r = \cos(\alpha) \Delta T_0 \left( \frac{3}{1 + e^{-11(L_M/X)}} \right) \left( \frac{X}{D} \right)^{-0.72} \]

with \( L_M = \frac{U_0(\pi D_0^2/4)^{1/4}}{\sqrt{\beta \Delta T}} g \)

\[ (7.2.6) \]
CHAPTER 7. RESULTS - OUTFALL SIMULATIONS

Figure 7.10: Logistic distribution fit to model results

7.2.2 Performance of equation

The equation results and the model results are compared for different outfall configurations and distances. Figure 7.11 gives the performance of the equation. The input parameters from scenario (1-20) in table 5.4 are used for these data points. The largest errors occur distances closest to the outfall. This is in line with the expectations in the previous section. The dashed lines indicate a 0.3 °C lower and upper bounds, compared to the model results. Almost every data point lies within this 0.3 °C bounded area.

In section 7.1 it is found that the decrease of surface temperature for outfalls with an outflow angle can be approximated with a reduction factor \( \cos(\alpha) \). The results of different outfall angles, combined with different initial jet characteristics are presented in figure 7.12. Table 7.1 provides the initial values of the scenarios used in figure 7.12 with the corresponding discharge and length scales. It proves that the equation also gives accurate predictions of the temperature rise for jets discharged with an angle.

<table>
<thead>
<tr>
<th>Run</th>
<th>( U_0 [m/s] )</th>
<th>( d [m] )</th>
<th>( \Delta T_0 [^{\circ}C] )</th>
<th>( D_0 [m] )</th>
<th>( \alpha [^{\circ}] )</th>
<th>( Q [m^3/s] )</th>
<th>( L_M [m] )</th>
</tr>
</thead>
<tbody>
<tr>
<td>31</td>
<td>2</td>
<td>6</td>
<td>10</td>
<td>2.5</td>
<td>5</td>
<td>9.8</td>
<td>17.4</td>
</tr>
<tr>
<td>32</td>
<td>2</td>
<td>6</td>
<td>10</td>
<td>2.5</td>
<td>15</td>
<td>9.8</td>
<td>17.4</td>
</tr>
<tr>
<td>33</td>
<td>2</td>
<td>6</td>
<td>10</td>
<td>2.5</td>
<td>30</td>
<td>9.8</td>
<td>17.4</td>
</tr>
<tr>
<td>34</td>
<td>2</td>
<td>6</td>
<td>10</td>
<td>2.5</td>
<td>45</td>
<td>9.8</td>
<td>17.4</td>
</tr>
<tr>
<td>35</td>
<td>2</td>
<td>6</td>
<td>10</td>
<td>2.5</td>
<td>30</td>
<td>24.5</td>
<td>43.4</td>
</tr>
<tr>
<td>36</td>
<td>3</td>
<td>6</td>
<td>15</td>
<td>2.5</td>
<td>30</td>
<td>14.7</td>
<td>21.3</td>
</tr>
</tbody>
</table>

Table 7.1: Input values for scenarios 31-36

Figure 7.11 and 7.12 show a good correlation of the equation results to the model results. For the combined results, it is found that the Root Mean Squared Error (RMSE) = 0.233. The bias of the equation results are found to be \(-0.11\). A negative bias means that equation results give an overestimation compared to the model results.
CHAPTER 7. RESULTS - OUTFALL SIMULATIONS

Figure 7.11: Results of run 1-20 with 0.3 °C upper and lower bounds

Figure 7.12: Results of run 31-36 with 0.3 °C upper and lower bounds
CHAPTER 7. RESULTS - OUTFALL SIMULATIONS

7.2.3 Robustness of equation

The robustness of the equation it is tested for the extreme outfall scenarios. These values are hardly encountered in the practical design range of FLNG outfalls. However, these extreme values prove a valuable insight in the use of the equation and its limitations. The parameter cube in figure 7.13 gives a visual representation of the initial parameters of the model runs. Every point inside the cube resembles one scenario with the initial conditions of $U_0$, $\Delta T_0$ and $D$. The extreme values of the initial conditions exist on the corners of the parameter cube. The performance of the equation is tested for these corner values.

The results of the eight extreme values are given in figure 7.14 with an upper and lower bound of 0.3 $^\circ$C. Scenarios 38, 42 and 44 result in an over estimation of the temperature rise. These scenarios have a small outfall diameter, resulting in small discharge rates. For scenario 38, $Q = 3.9 m^3/s$ and for scenario $44$, $Q = 0.8 m^3/s$. These small flow rates in combination with high temperature differences $\Delta T_0$, result in an overestimation of the surface temperature rise. Some results have an error up to 100%. Only scenarios 44 and 38 fall outside the 0.3 $^\circ$C error bounds.

![Figure 7.13: Initial value parameter cube](image)

Table 7.2: Extreme value scenarios

<table>
<thead>
<tr>
<th>Run</th>
<th>$U_0 [m/s]$</th>
<th>$d [m]$</th>
<th>$\Delta T_0 [^\circ C]$</th>
<th>$D_0 [m]$</th>
<th>$\alpha [^\circ]$</th>
<th>$Q [m^3/s]$</th>
<th>$L_M [m]$</th>
</tr>
</thead>
<tbody>
<tr>
<td>37</td>
<td>5</td>
<td>6</td>
<td>15</td>
<td>2.5</td>
<td>0</td>
<td>24.5</td>
<td>35.4</td>
</tr>
<tr>
<td>38</td>
<td>5</td>
<td>6</td>
<td>15</td>
<td>1</td>
<td>0</td>
<td>3.9</td>
<td>22.4</td>
</tr>
<tr>
<td>39</td>
<td>1</td>
<td>6</td>
<td>5</td>
<td>2.5</td>
<td>0</td>
<td>4.9</td>
<td>12.3</td>
</tr>
<tr>
<td>40</td>
<td>1</td>
<td>6</td>
<td>5</td>
<td>1</td>
<td>0</td>
<td>0.8</td>
<td>7.8</td>
</tr>
<tr>
<td>41</td>
<td>5</td>
<td>6</td>
<td>5</td>
<td>2.5</td>
<td>0</td>
<td>24.5</td>
<td>61.4</td>
</tr>
<tr>
<td>42</td>
<td>5</td>
<td>6</td>
<td>5</td>
<td>1</td>
<td>0</td>
<td>3.9</td>
<td>38.8</td>
</tr>
<tr>
<td>43</td>
<td>1</td>
<td>6</td>
<td>15</td>
<td>2.5</td>
<td>0</td>
<td>4.9</td>
<td>7.1</td>
</tr>
<tr>
<td>44</td>
<td>1</td>
<td>6</td>
<td>15</td>
<td>1</td>
<td>0</td>
<td>0.8</td>
<td>4.5</td>
</tr>
</tbody>
</table>

7.2.4 Applicability range of equation

Distance

Equation 7.2.6 is not able to correctly predict the temperature rise for all combinations of the momentum length scales and distances. The distance for which the expression is applicable, depends on the jet initial conditions. Figure 7.15 shows the surface temperature profiles of both scenario 1 and 5 from table 5.4,
CHAPTER 7. RESULTS - OUTFALL SIMULATIONS

Figure 7.14: Performance of equation for extreme value scenarios

together with the corresponding results from equation 7.2.6. For \( U_0 = 5 \text{ m/s} \) the equation gives reliable results for \( X > 50 \text{ m} \). However, for \( U_0 = 1 \text{ m/s} \) the equation gives reliable results for \( X > 20 \text{ m} \). The minimum distance where the equation provides reliable results depends on the outflow conditions.

An important parameter to determine the applicability range proves to be the ratio between the momentum length scale \( L_M \) and the distance \( X \). From figure 7.8c it is concluded that the expression provides reliable results up to \( (L_M/X) < 0.7 \). This implies that the equation is valid for distances of approximately \( X > 1.5L_M \).

Figure 7.15: Equation results compared to model results

Angle

The magnitude of the momentum length scale in combination with the jet angle proves to be very important for shallow water FLNG. If the jet momentum length scale, \( L_M \) is larger than the path
from the jet orifice to the bottom $L_\alpha$, the jet will touch the bottom. This is visible for scenario 35 in figure 7.16. This results in higher temperatures near the bottom than at the surface. At the point where the buoyancy becomes dominant over jet momentum, the warmer water will rise. This results in a temperature increase of the whole water column. The equation results will not correspond to the temperature results of the model. The limiting condition for jet momentum length scale and jet outfall angle is given by equation 7.2.8. Where $Z$ is the total water depth, $d$ the outfall depth from the water surface and $\alpha$ the outfall angle.

\begin{equation}
L_\alpha = \frac{(Z - d)}{\sin(\alpha)} \quad (7.2.7)
\end{equation}

\begin{equation}
L_M < L_\alpha \quad (7.2.8)
\end{equation}

\textbf{7.2.5 Conclusion}

For a wide range of initial conditions, the proposed equation provides results in line with the model output. For the chosen initial conditions, which come close to flow rates and design values for FLNG, the equation provides results within a 0.3 °C error margin to the model results. In addition, a range of applicability is proposed. This range gives a minimum distance $X$ for where the equation provides realistic results, based on the jet initial conditions. The proposed equation can also predict the surface temperature for jets with a downward directed outfall angle. Because of the angle, the jet travels a longer distance through the ambient water. The increased entrainment decreases the temperature significantly. For shallow water FLNG applications, a restriction to the angle and the flow parameters is presented. If the momentum length scale is longer or equal than the centerline path to the bottom, the jet will attach to the bottom. For very low flow rates and low momentum length scales, the proposed equation overestimates the surface temperature rise.
Chapter 8

Discussion

FLNG cooling water outfalls can be classified as high momentum buoyant jets with a relatively low submergence. Insight into the mixing and distribution of temperature in these buoyant jets is of fundamental importance in reducing the environmental impact. With the increasing demand for FLNG facilities worldwide it is important to gain insight into the fundamental physical processes governing the evolution of these jets and to improve our ability to predict their evolution and downstream spreading.

Moreover it is equally important to accurately simulate the physical processes in these turbulent jets using fully non-hydrostatic numerical models while at the same time finding a balance with the computational time. Traditionally CORMIX has been widely used in jet studies, as it does not solve the RANS equations but instead relies on empiricism it is computationally very efficient, but can only be applied for simple domains. Consequently it does not resolve the physics processes, or allow for complex geometries.

Here OpenFOAM, a state of the art CFD code has been used. The model validation showed that the software is capable of modeling round turbulent buoyant jets. OpenFOAM simulations are time consuming, therefore a quicker method is required to examine various outfall designs and the corresponding temperature distributions. The approach adopted in this thesis is that an advanced CFD code has been applied to perform a dimensional analysis on the most important parameters influencing the high momentum buoyant jet. A new method has been found to relate different groups of dimensionless parameters to predict the surface temperature rise.

While, highly idealized conditions are used for the outfall simulations, one can obtain a clear perspective on the dominant mixing processes and the surface temperature rise. The implication of the simplifications are discussed further in this section.

Dimensional analysis

The purpose of a dimensional analysis is to find useful dimensionless groups of variables to provide a basis for similarity between physical models, experimental datasets and numerical models [Sonin, 2001]. Fischer [1979] used the method of dimensional analysis to form the jet length scales (section 3.2).

From the dimensional analysis it is found that the relative temperature rise follows a logistic distribution to the momentum length scale, after the point of surface impingement. The logistic distribution reaches a constant relative surface temperature for increasing values of the momentum length scale. Reaching a constant relative surface temperature rise for increasing momentum length scales (increasing discharge) implies that the heat is transported to other parts in the domain. The increased momentum results in additional mixing processes. This can be explained with two momentum induced diffusion processes. An increase in outflow velocity, which will result in an increase in turbulent jet entrainment. And increased mixing due to large currents near the surface, induced by the jet momentum. This results in a steeper
vertical velocity gradient. The steeper vertical velocity gradient will increase the vertical mixing of the temperature. Both mixing processes have already been illustrated in figure 7.2a and figure 7.2b.

According to Sobey et al. [1988], the submergence of the jet is an important parameter for the jet trajectory up to the point of surface impingement. In this thesis it is found that, after the point of surface impingement, the surface temperature rise becomes independent of the outfall depth. Without ambient currents, the shape of the buoyant surface plume is governed by the outfall depth, the jet momentum and jet temperature. It is found in section 7.1 that the relative submergence \((d/D)\) has no influence on the temperature profile for the range considered in this thesis. However, it is found to be an important parameter for the applicability range of the equation. According to the classification system of Lee and Jirka [1981], the outfall of the FLNG is considered very shallow. The maximum relative submergence \((d/D)\) used in this thesis is \((15/2) = 7.5\). For jet submergence smaller than 7.5, the surface temperature rise proved to be independent to the outfall depth. Buoyant jets with a larger relative submergence \((d/D) > 7.5\) result in a different shape of the buoyant surface plume, this is illustrated in figure 8.1. If the jet becomes plume-like before surface interaction, the spreading of the buoyant surface plume will be radial. This is a different shape than used to determine the empirical relation. For larger jet submergence, the equation will probably produce incorrect results.

Figure 8.1: Different submergence resulting in different buoyant surface plumes

The empirical relation, resulting from the dimensional analysis, accurately predicts the surface temperature rise within the applicability ranges; \(X > 1.5L_M, L_M < \frac{(Z-d)}{\sin(\alpha)}\) and \((d/D) < 7.5\). Equation 8.0.1 directly provides a measure of the influence of a parameter on the surface temperature rise. Besides the applicability ranges, one must be aware that the equation is found under highly idealized conditions. The equation can be used to examine outfalls for still water conditions, without waves and currents. Equation 8.0.1 is a prediction tool for the surface temperature rise at a location along the X-axis in figure 5.1 and 5.2.

\[
T_r = \cos(\alpha) \Delta T_0 \left( \frac{3}{1 + e^{-11(L_M/X)}} \right) \left( \frac{X}{D} \right)^{-0.72} \text{ with } L_M = \frac{U_0(\pi D_0^2/4)^{1/4}}{\sqrt{(3\Delta T)g}} \tag{8.0.1}
\]

Current patterns around a FLNG facility are complicated. Large turbulent eddies and wakes occur around the stern of the FLNG. The effects of large scale fluid motions on the buoyant jet are difficult to predict. The jet-wake interactions can affect the entrainment rates and advect the thermal plume to further distances [Cederwall, 1971]. Furthermore, waves affect the diffusion of the buoyant plume. The orbital current movements under the waves influence the mixing of the buoyant plume with the ambient water [Yue and Wang, 2009]. Without considering the effects of waves and currents, the equation will lead to a conservative approach regarding the environmental regulations. The temperature increase at the
surface will most probably be overestimated. Vertical mixing or entrainment can also be induced by wind acting on the surface. This introduces a shear stress, which develops turbulence in the upper layer. This upper layer turbulence can result in entrainment from the layer beneath. Wind shear stresses are not taken into account in OpenFOAM.

Beyond the numerical boundaries the buoyant surface spreading and mixing continues if the source keeps discharging heat. Mixing processes continue to take place and for still water conditions, the total water column will eventually increase in temperature for large time scales. A realistic maximum physical time scale, would apply to approximately 1/4 of a tidal cycle. This is related to the change from ebb to flood and the other way around. This tidal phase results in constant currents, which can be modeled in steady state computations. Computations longer than this physical time scale do not represent realistic physical results for environments influence by tides.

Wider application

In this thesis, the buoyant jet has outflow quantities corresponding to FLNG outfalls. The accuracy of the equation is guaranteed for the proposed applicability ranges and under the given assumptions. In addition to FLNG outfalls, the equation can be used for other buoyant jet applications, e.g. outfalls for onshore LNG facilities, onshore power plants and buoyant sewage outfalls.

In the classification system of Shimada et al. [2004] for jet submergence, \((d/D)\) it is found that the outfall is influenced by presence of the water surface. Shimada et al. [2004] proposed conditions of the relative submergence, \((d/D)\) and the relative water depth, \((d/H)\) of the jet. Three different regions exist, the region where the jet is only influence by the surface, the region where both surface and bottom influence the jet and the region where the jet is only influenced by the bottom. For FLNG cooling water outfalls, it is found that the presence of the bottom does not influence the jet trajectory and diffusion. From the vertical temperature profiles it is found that for some outflow conditions, the temperature increases throughout the whole water column. This results in a re-entrainment of the warmer water into the jet and it is expected that this does not result in significant different flow patterns and temperature profiles. This implies that the empirical equation can also be used for outfalls in deep water conditions.

Scaling

Scaling the empirical equation could result in a wider application in the field of buoyant jets. The empirical relation is found from different groups of dimensionless parameters following a logistic distribution. The logistic distribution, given by equation 8.0.3, is fitted to the relations with the constants \(c_1\), \(c_2\) and \(c_3\). The value of these constants for the scale considered in this thesis are given by:

\[
c_1 = -0.72, \quad c_2 = 3, \quad c_3 = -11.
\]  

\[
T_r = \cos(\alpha) \Delta T_0 \left( \frac{c_2}{1 + e^{c_3(L_M/X)}} \right) \left( \frac{X}{D} \right)^{c_1} \quad \text{with} \quad L_M = \frac{U_0(\pi D_0^2/4)^{1/4}}{\sqrt{\beta \Delta T}} \frac{g}{g}
\]  

The constants describe respectively the reduction factor for increasing distances, the maximum dimensionless temperature rise and the steepness of the logistic distribution. If all the dimensionless groups are scaled in the same order, then the same logistic distribution is expected for the dimensionless temperature profile. However, the constants of the equation can change for different outflow or geometry scales. The constants found in thesis are only applicable within the given applicability ranges and for initial outflow values within the parameter cube of figure 7.13.

The constants in the equation include unknown processes which could not be included in the equation. If these processes are changed due to scaling, the constants will change. For other geometric scales
CHAPTER 8. DISCUSSION

and turbulence intensities, the thermal dissipation of the buoyant jet to the surrounding fluid could be different. From other scaling studies [Sobey et al., 1988] it is found that the jet Reynolds number, densimetric Froude number and the Prandtl number are important in the scaling process. The constants themselves can be a function of these parameters.

\[ c_1, c_2, c_3 = f(Re, Fr_d, Pr, \ldots) \]  
\[ (8.0.4) \]

For different geometrical and outflow scales, the equation can be scaled, but will probably lead to other values of the constants. An additional study on the dependence of the constants on the dimensionless parameters is required to obtain an equation, which is applicable for the whole range of buoyant jet scales. The dimensionless limits \( X > 1.5L_M, L_M < \frac{(Z-d)}{sin(\alpha)} \) and \((d/D) < 7.5\) still need to be fulfilled.

Turbulence

The results of this thesis are obtained with RANS computations, using the standard \( k-\epsilon \) turbulence closure. This turbulence closure is widely validated and also successfully used by Aziz et al. [2008] in studying round turbulent jets. The standard \( k-\epsilon \) [Lauder and Spalding, 1974] works well for free shear flow with high Reynolds numbers [Bardina et al., 1997]. For simple shear flows, like round- and axisymmetric jets, a simple constant eddy viscosity (turbulent viscosity) can be used [Bird et al., 2002]. If the turbulent viscosity is constant, the equations of motions for laminar flow can be used and the viscosity in the RANS equations can be replaced by the eddy viscosity [Bird et al., 2002]. The realizible \( k-\epsilon \) model is developed for more complicated flows involving rotation, boundary layers, separation and swirling flows. Aziz et al. [2008] evaluated the accuracy of different \( k-\epsilon \) closure models to the decay of centerline velocities, jet growth, velocity profiles and kinetic energy profiles. Aziz et al. [2008] found that the standard \( k-\epsilon \) scheme performed equally well and in some cases better than other turbulence schemes for round turbulent jets.

The results in this thesis obtained with the standard \( k-\epsilon \) are in good agreement with the experimental data. For these type of round turbulent jets, the standard \( k-\epsilon \) can give realistic representations of jet trajectory, spreading rates and dilution rates. The found jet trajectories, starting point of the zone of established flow (ZFE) and Gaussian shaped spreading rates are in line with the theoretical values of Fischer [1979] and experimental values of Albertson et al. [1950]. In addition, the far-field shapes of the buoyant plume are in line with the description of buoyant plume development by Jirka et al. [1992]. Next to the RANS approach, also LES can be used in OpenFOAM. Using LES will result in a time dependent surface temperature profile. The temperature rise for a fixed location would result in a temperature probability distribution. This results in higher maximum temperatures than obtained by means of RANS computations. The use of LES is far more expensive in computational time and data production than RANS. Therefore LES was not used for the outfall simulations.

The standard \( k-\epsilon \) and the realizible \( k-\epsilon \) model need additional wall-functions to realistically represent the turbulent shear in the boundary layer near a wall. Besides the standard \( k-\epsilon \) and the realizible \( k-\omega \) turbulence closure can be used [Wilcox, 2008]. The \( k-\omega \) turbulence model is also a two-equation model, but more applicable for low Reynolds regions. These low Reynolds number are often obtained near the wall. The \( k-\omega \) model does not require wall functions and represents flows near the wall better than the \( k-\epsilon \) models. To include the best of both turbulence closures, the SST (Menter’s Shear Stress Transport) turbulence model is developed [Menter, 1994]. The SST-model combines the high Reynolds flow prediction of the \( k-\epsilon \) models and the wall turbulence modeling of the \( k-\omega \) model. The SST makes use of the \( k-\omega \) model from the boundary layer to the wall. It switches to the \( k-\epsilon \) model for the free-stream turbulence. Basically, the SST model is a four-equation turbulence model. Because of the included \( k-\omega \) model, a fine mesh close to the wall is required. The jets in this thesis are well represented by the \( k-\epsilon \) model. The surface has a slip boundary condition, which indicated that no boundary layer exists at the surface and no wall function or special wall turbulence models is required. Using the realizible \( k-\epsilon \), \( k-\omega \) or SST turbulence closure most probably do not lead to different results.
CHAPTER 8. DISCUSSION

Type of model

OpenFOAM is a very powerful tool, able to correctly simulate the jet trajectories and diffusion for many different outfall configurations. Various solver packages are available, which include the equations for very specific problems. The results in this thesis offer the possibilities to extend the application of OpenFOAM in jet and outfall studies and form a basis for future research. Although, the configuration of the model in this thesis is relatively simple, the model setup takes a lot of time and clear documentation of the setup is not available. Next to OpenFOAM, many other commercial CFD packages are available. Commercial software often include a graphical user interface, this makes the use of commercial software in general more convenient.

CORMIX uses a sequence of relatively simple simulations, which predict the trajectories and dilution of the jet in the given flow conditions. CORMIX can only perform calculations in steady-state situations together with relatively easy bathymetry [CORMIX-UserGuide, 1996]. The model is built from a flow classification system, consisting of a database that can distinguish many flow patterns of jets and outfalls. Jet characteristic length scales play an important role in the classification system. Jet physics are not calculated in the model, but implicitly included in the jet shapes. This limits the possibilities to study jets and the corresponding physical behaviour. The main advantage is that CORMIX computes the jet trajectories in a short amount of time, with good accuracy for the validated range of input parameters.

For engineering project purposes, one should consider if a CFD model, together with the obtained accuracy, is necessary for the study objective. Setting up a CFD run is time consuming and some computational background and basic knowledge of the numerical processes is required. A CFD model is recommended if the user is interested in the turbulence and mixing processes of turbulent jets and studies including complicated bathymetries and flow patterns. If only a temperature or concentration profile is required for steady state situation and simple geometries, simpler and faster models, such as CORMIX can be more valuable with respect to time and costs if the parameters fall within the validated range. A simple geometry is used in this thesis, therefore it could be valuable to compare the equation results with CORMIX. This could improve the reliability of the equation for future applications in buoyant jet studies.

Validation

The salinity of the water influences the magnitude of relative density difference for fixed temperature differences, which in turn affect the jet trajectories and diffusion processes. In this thesis, the influence of the salinity is taken into account by modifying the thermal expansion coefficient \( \beta \). The linear thermal expansion coefficient \( \beta \) is used to calibrate the model towards the most realistic values of the equation of state, which is non-linear. This proved to be successful. Using \( \beta \) to add salinity effects can be a useful to apply in studies using the Boussinesq assumption for incompressible fluids. The jet trajectories are sensitive towards the value of \( \beta \), therefore adjusting \( \beta \) needs to be done with caution.

Jets have been studied extensively in the past and many data sets are available to validate the centerline velocity decay, centerline trajectories, spreading rates and centerline dilution rates of jets. These datasets have been used for the validation of the jet in OpenFOAM. The results proved to be in good agreement with the experimental data. However, no data sets were available to validate the surface spreading of the jet. It is assumed that the model represents the buoyant surface spreading realistically. This assumption is based on the fact that the dilution of temperature, before the point of surface impingement, is found to be in good agreement with the data. In addition, the shape of the buoyant surface plume is in line with the experimental and theoretical findings.

Implications of model assumptions

Single-phase computations have been used in this thesis. This means that there is no interface between water surface and air, only a box completely filled with water is considered. In reality, the water surface acts as a barrier to the upward motion of the buoyant jet. Depending on the kinetic energy of the
jet a small surface rise will occur around the point of surface impingement. For a single-phase model, this surface rise is not visible and cannot be studied. In addition, due to low pressure effects, created by the jet entrainment demand, one side of the jet will attach to the water surface (Coanda effect) [Shimada et al., 2004]. These low pressure effects can create a water level decrease just before the point of surface impingement. Shimada et al. [2004] also found that under certain conditions, the point of surface impingement could oscillate as a result of the surface rise due to the buoyant jet upward kinetic energy and the surface level decay due to pressure effects. The effects of the jet on the water surface elevations cannot be taken into account for single-phase models. To take into account surface level effects, an air to water surface interface needs to be used. Multi-phase computations are more sensitive to instabilities and the computational time increases significantly. In addition, a multi-phase computation does not contribute to the objective of this thesis.

In this thesis one outfall pipe is used with an equivalent diameter of multiple outfall pipes. Jirka [2006] and Papps [1995] have studied the mechanism of merging jets and jet interaction. The contact surface with the ambient water is larger for three outfall pipes than it is for one equivalent pipe diameter. The result is that the entrainment, and therefore temperature dilution, is more effective for multiple outfall jets. Using only one equivalent jet diameter results in an overestimation of the temperature rise. The effect of multiple jet mixing is expected to have a significant effect on the surface temperature rise. The magnitude of the influence for multiple outfall pipes in the temperature rise study needs to be examined in future studies.
Chapter 9

Conclusions and recommendations

In this thesis a three-dimensional CFD model is used to evaluate the cooling water outfall of a Floating Liquefied Natural Gas (FLNG) facility. An empirical relationship is found between the outfall design parameters and the temperature rise of the surrounding waters. Some recommendations are proposed for additional and further research on cooling water outfall systems.

9.1 Conclusions

The outfall discharge of a FLNG can be characterized as a large momentum buoyant jet relatively close to the surface. High jet momentum, together with large outfall volumes and low submergence results in significant surface currents which transport the buoyant plume in the ambient water. The surface currents result in a large vertical velocity gradient. The high velocity gradients introduce shear stresses between fluid layers of different velocity. This process enhances the turbulent mixing and temperature diffusion in the vertical direction.

From a parameter sensitivity study it is concluded that the main input parameters influencing the surface temperature rise are the outflow velocity $U_0$, initial outflow temperature difference $\Delta T_0$, the outflow diameter $D$ and the outfall angle $\alpha$. It is found that increasing the outfall depth, $d$ does not lead to a significant reduction in surface temperature rise. This is true for distances behind the point of surface impingement. Increasing the outfall angle, results in a temperature decay in the far-field region of the jet, it is found that the temperature decreases by a factor $\cos(\alpha)$.

From the dimensional analysis it is concluded that the relative surface temperature rise follows a logistic distribution to the momentum length scale, after the point of surface impingement. For distances larger than the surface impingement, the relative temperature rise ($T_r/\Delta T_0$) does not increase for increasing momentum length scales ($L_M$). Increasing the outfall velocity, while keeping the temperature difference constant, does not necessarily lead to an increase in surface temperature. This is the result of increased mixing processes induced by the jet turbulent entrainment and the increased vertical velocity gradient.

For increasing distances from the jet orifice, the surface temperature rise decreases. By multiplying the relative temperature rise ($T_r/\Delta T_0$) with a dimensionless factor $(X/D)^{0.72}$, the relations of ($L_M/X$) to the relative temperature rise ($T_r/\Delta T_0$) becomes independent of the distance $X$. This factor includes the decay of temperature for increasing distances. The dimensional analysis results in an empirical relationship, which proves to accurately predict the surface temperature rise for the considered outfall scenarios after the point of surface impingement. Three conditions need to be fulfilled for the equation to be applicable; $X > 1.5L_M$, $L_M < \frac{(Z-d)}{\sin(\alpha)}$ and $(d/D) < 7.5$. First, the distance from the jet orifice for which the equation is applicable is given by $X > 1.5L_M$. Secondly, for shallow water FLNG
facilities a restriction is found to avoid the risk of bottom attachment, \( L_M < \frac{(Z-d)}{\sin(\alpha)} \). Finally, the relative submergence of the outfall should not exceed \((d/D) < 7.5\).

The study on the robustness of the equations showed that for small outflow diameters in combination with large outfall temperatures, the equation overestimates the surface temperature rise \( T_r \). Furthermore it is found that the jet trajectories and diffusion processes are not influenced by the presence of the bottom for the proposed submergence ranges. Therefore, the empirical equation can also be used for deep water buoyant jets, as long as the buoyant jet configuration meets the applicability ranges.

FLNG outfalls need to be designed with known outfall discharges and temperature. It is found that for equal outflow volumes and temperatures, a large initial velocity in combination with a small port diameter results in the most effective mixing of temperature. The large jet entrainment in combination with large vertical velocity gradient mix the heat through the water column most effectively.

The OpenFOAM results for a pure jet show a very good agreement to experimental data sets. The standard \( k-\epsilon \) turbulence closure very well models the jet trajectories, spreading rates and temperature diffusion. The results of OpenFOAM are not sensitive to the turbulent Prandtl number, \( Pr_t \). The heat flux through the water surface is allowed to be zero. Adding a heat flux, resulting from free or forced convection does not lead to significant temperature decay in the top layer of the water column. The solver in OpenFOAM used for this thesis is not able to include salinity. The salinity of the water affects the change of density as a result of temperature differences. The thermal expansion factor \( \beta \) can be used to represent the non-linear equation of state. The thermal expansion coefficient can be used as a variable to correct the change of density as a result of temperature change for saline waters.

9.2 Recommendations

The following recommendations are given for future research on FLNG cooling water outfalls. Future research can go into more detail on the physical processes and accuracy of the jet simulation, or can perhaps extend the research presented in this thesis to include LES studies.

The domain of the study needs to be extended to give an indication of the recirculation risks. A part of the FLNG intake needs to be incorporated in the model to examine the recirculation risks. Currents around a FLNG facility are complicated and include many turbulent eddies. Including the hull geometry can also results in the modeling of the jet-wake interactions. Furthermore, the model results can be improved by taking into account multiple outfall jet. The additional mixing and diffusion of the jet for multiple jet interaction needs to be studied thoroughly. Adding the number of pipes and corresponding additional dilution to the equation would make the prediction more realistic.

More research needs to be done on the validation of the buoyant surface spreading. No data was available to validate the model results. Furthermore, it is recommended that the results of the equation are compared to CORMIX results. This could provide extra insight in the application range of the equation and the predictive capabilities of the empirical equation. It is also recommended to extend the study on the scaling possibilities of the equation. This could potentially lead to an equation only depending on the outfall characteristics, without fixed constants.

For future research it is recommended to use a commercial software package. This will result in a more convenient setup of the model. Many combinations of boundary conditions exist in OpenFOAM and many different solvers, without the support of good documentation. In addition, OpenFOAM does not have a graphical user interface. This makes the software less user friendly than commercial packages, which often do have an integrated user interface.
Bibliography


## Nomenclature

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \alpha )</td>
<td>Jet outflow angle</td>
</tr>
<tr>
<td>( \alpha_G )</td>
<td>Entrainment coefficient</td>
</tr>
<tr>
<td>( \alpha_t )</td>
<td>Thermal diffusivity</td>
</tr>
<tr>
<td>( \beta )</td>
<td>Specific buoyancy flux</td>
</tr>
<tr>
<td>( \epsilon )</td>
<td>Jet spreading factor</td>
</tr>
<tr>
<td>( \Delta \rho )</td>
<td>Thermal expansion coefficient</td>
</tr>
<tr>
<td>( \Delta T_0 )</td>
<td>Difference in density of ambient fluid and the fluid in a jet</td>
</tr>
<tr>
<td>( \epsilon_H )</td>
<td>Temperature difference between jet fluid and ambient fluid</td>
</tr>
<tr>
<td>( \epsilon_H )</td>
<td>Heat transfer eddy diffusivity</td>
</tr>
<tr>
<td>( \epsilon_M )</td>
<td>Momentum transfer eddy diffusivity</td>
</tr>
<tr>
<td>( \kappa_t )</td>
<td>Kinematic turbulent thermal conductivity</td>
</tr>
<tr>
<td>( \lambda )</td>
<td>Ratio of velocity half-width over concentration half-width</td>
</tr>
<tr>
<td>( \mu )</td>
<td>Dynamic viscosity</td>
</tr>
<tr>
<td>( \mu )</td>
<td>Specific mass flux</td>
</tr>
<tr>
<td>( \nu )</td>
<td>Kinematic viscosity</td>
</tr>
<tr>
<td>( \nu_k )</td>
<td>Turbulent kinematic viscosity</td>
</tr>
<tr>
<td>( \Omega )</td>
<td>Angular velocity vector</td>
</tr>
<tr>
<td>( \rho )</td>
<td>Density</td>
</tr>
<tr>
<td>( \rho_a )</td>
<td>Density of ambient fluid</td>
</tr>
<tr>
<td>( \rho_{ref} )</td>
<td>Reference density</td>
</tr>
<tr>
<td>( A )</td>
<td>Cross-sectional area</td>
</tr>
<tr>
<td>( B )</td>
<td>Buoyancy flux of a jet</td>
</tr>
<tr>
<td>( b )</td>
<td>Measure for the width of the jet</td>
</tr>
<tr>
<td>( B_0 )</td>
<td>Initial buoyancy flux of a jet</td>
</tr>
<tr>
<td>( b_g )</td>
<td>Velocity half-width</td>
</tr>
<tr>
<td>( b_{gc} )</td>
<td>Concentration half-width</td>
</tr>
<tr>
<td>( c )</td>
<td>Concentration</td>
</tr>
<tr>
<td>( c_0 )</td>
<td>Initial jet concentration</td>
</tr>
<tr>
<td>( c_{cm} )</td>
<td>Centerline concentration</td>
</tr>
<tr>
<td>( C_p )</td>
<td>Specific heat capacity</td>
</tr>
<tr>
<td>( D )</td>
<td>Jet outfall diameter</td>
</tr>
<tr>
<td>( d )</td>
<td>Jet distance below water surface</td>
</tr>
<tr>
<td>( f )</td>
<td>Coriolis frequency</td>
</tr>
<tr>
<td>( F_a )</td>
<td>Buoyancy force</td>
</tr>
<tr>
<td>( Fr_d )</td>
<td>Densimetric Froude number</td>
</tr>
<tr>
<td>( g )</td>
<td>Acceleration of gravity</td>
</tr>
<tr>
<td>( g' )</td>
<td>Effective acceleration of gravity</td>
</tr>
<tr>
<td>( H )</td>
<td>Depth water column</td>
</tr>
<tr>
<td>( h_c )</td>
<td>Convective heat transfer coefficient</td>
</tr>
<tr>
<td>( k )</td>
<td>Thermal conductivity</td>
</tr>
<tr>
<td>Symbol</td>
<td>Definition</td>
</tr>
<tr>
<td>--------</td>
<td>------------</td>
</tr>
<tr>
<td>$l$</td>
<td>Turbulent kinetic energy</td>
</tr>
<tr>
<td>$L_b$</td>
<td>Spacing between jets</td>
</tr>
<tr>
<td>$L_M$</td>
<td>Characteristic length scale for buoyancy and ambient current</td>
</tr>
<tr>
<td>$L_q$</td>
<td>Momentum length scale for a buoyant jet</td>
</tr>
<tr>
<td>$M$</td>
<td>Characteristic length scale for a pure jet</td>
</tr>
<tr>
<td>$M_0$</td>
<td>Momentum flux of a jet</td>
</tr>
<tr>
<td>$m$</td>
<td>Specific momentum flux</td>
</tr>
<tr>
<td>$n$</td>
<td>Number of jets per unit length</td>
</tr>
<tr>
<td>$p$</td>
<td>Initial momentum flux of a jet</td>
</tr>
<tr>
<td>$Pr$</td>
<td>Pressure</td>
</tr>
<tr>
<td>$Pr_t$</td>
<td>Prandtl number</td>
</tr>
<tr>
<td>$Q$</td>
<td>Number of jets per unit length</td>
</tr>
<tr>
<td>$q$</td>
<td>Turbulent Prandtl number</td>
</tr>
<tr>
<td>$Q_0$</td>
<td>Heat flux</td>
</tr>
<tr>
<td>$r$</td>
<td>Initial volume flux of a jet</td>
</tr>
<tr>
<td>$R_0$</td>
<td>Radial distance from jet center</td>
</tr>
<tr>
<td>$Re$</td>
<td>Jet Richardson number</td>
</tr>
<tr>
<td>$Re$</td>
<td>Reynolds number</td>
</tr>
<tr>
<td>$S$</td>
<td>Centerline dilution coefficient</td>
</tr>
<tr>
<td>$S_c$</td>
<td>Salinity</td>
</tr>
<tr>
<td>$T$</td>
<td>Temperature</td>
</tr>
<tr>
<td>$T_a$</td>
<td>Ambient fluid temperature</td>
</tr>
<tr>
<td>$T_r$</td>
<td>Temperature rise of the surface relative to ambient temperature</td>
</tr>
<tr>
<td>$T_{ref}$</td>
<td>Reference temperature, often ambient fluid temperature</td>
</tr>
<tr>
<td>$U_0$</td>
<td>Initial outflow velocity</td>
</tr>
<tr>
<td>$U_c$</td>
<td>Centerline velocity for buoyant jet</td>
</tr>
<tr>
<td>$U_m$</td>
<td>Centerline velocity for pure jet</td>
</tr>
<tr>
<td>$v_e$</td>
<td>Entrainment velocity</td>
</tr>
<tr>
<td>$x$</td>
<td>Cartesian coordinate in direction of jet outflow</td>
</tr>
<tr>
<td>$y$</td>
<td>Cartesian coordinate horizontally perpendicular to $x$</td>
</tr>
<tr>
<td>$z$</td>
<td>Cartesian coordinate vertically upward</td>
</tr>
</tbody>
</table>
# List of Figures

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Source/Details</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.1</td>
<td>Artist impression of Shell Prelude Floating Liquefied Natural Gas (FLNG) facility, source: <a href="http://www.shell.com">www.shell.com</a></td>
<td>1</td>
</tr>
<tr>
<td>1.2</td>
<td>Illustration of the buoyant jet surface spreading, side view</td>
<td>2</td>
</tr>
<tr>
<td>1.3</td>
<td>Illustration of the buoyant jet surface spreading, looking at the stern</td>
<td>2</td>
</tr>
<tr>
<td>2.1</td>
<td>Sewage outfall, Broward County, Florida, source: [<a href="http://www.marinephotobank.org">http://www.marinephotobank.org</a>]</td>
<td>5</td>
</tr>
<tr>
<td>2.2</td>
<td>Inclined buoyant jet in stagnant ambient environment, [Jirka and Harleman, 1979]</td>
<td>6</td>
</tr>
<tr>
<td>2.3</td>
<td>Turbulent entrainment of ambient fluid</td>
<td>7</td>
</tr>
<tr>
<td>2.4</td>
<td>Jets with different momentum length scales, $L_M$</td>
<td>7</td>
</tr>
<tr>
<td>2.5</td>
<td>Buoyant jet surface interaction</td>
<td>8</td>
</tr>
<tr>
<td>2.6</td>
<td>Turbulent jet computation with DNS (a), LES (b) and RANS (c), source: [<a href="http://www.psc.edu">http://www.psc.edu</a>]</td>
<td>9</td>
</tr>
<tr>
<td>2.7</td>
<td>Illustration of the domain considered in this thesis</td>
<td>11</td>
</tr>
<tr>
<td>3.1</td>
<td>Round turbulent jet with two jet stages</td>
<td>15</td>
</tr>
<tr>
<td>3.2</td>
<td>Profile of time averaged velocity in a round jet, [Lee and Chu, 2003]</td>
<td>15</td>
</tr>
<tr>
<td>3.3</td>
<td>Gaussian distribution of velocity and concentration, [Chu, 1996]</td>
<td>16</td>
</tr>
<tr>
<td>3.4</td>
<td>Profile of time averaged velocity in a round jet, [Chu, 1996]</td>
<td>17</td>
</tr>
<tr>
<td>3.5</td>
<td>Profile of time averaged concentration in a round jet, [Chu, 1996]</td>
<td>17</td>
</tr>
<tr>
<td>4.1</td>
<td>Turbulent velocity fluctuation as a function of time</td>
<td>20</td>
</tr>
<tr>
<td>6.1</td>
<td>OpenFOAM vector plot of the flow directions, side view</td>
<td>45</td>
</tr>
<tr>
<td>6.2</td>
<td>OpenFOAM vector plot of the flow directions, back view</td>
<td>46</td>
</tr>
<tr>
<td>6.3</td>
<td>OpenFOAM temperature contour of $\Delta T = 2^\circ C$</td>
<td>46</td>
</tr>
<tr>
<td>6.4</td>
<td>OpenFOAM temperature contours of $\Delta T = 1^\circ C$, $\Delta T = 2^\circ C$, $\Delta T = 3^\circ C$</td>
<td>46</td>
</tr>
<tr>
<td>6.5</td>
<td>Jet cross section, indicted by lines, with distances from jet orifice: $X/D_0 = 0$, $X/D_0 = 3$, $X/D_0 = 6$, $X/D_0 = 10$, $X/D_0 = 15$, $X/D_0 = 20$</td>
<td>46</td>
</tr>
<tr>
<td>6.6</td>
<td>Centerline velocity decay</td>
<td>47</td>
</tr>
<tr>
<td>6.7</td>
<td>Velocity profiles across the jet</td>
<td>47</td>
</tr>
<tr>
<td>6.8</td>
<td>Dimensionless radial velocity profiles</td>
<td>47</td>
</tr>
<tr>
<td>6.9</td>
<td>Buoyant jet trajectory for $\Delta T = 20^\circ C$</td>
<td>48</td>
</tr>
<tr>
<td>Figure</td>
<td>Description</td>
<td>Page</td>
</tr>
<tr>
<td>--------</td>
<td>-----------------------------------------------------------------------------------------------</td>
<td>------</td>
</tr>
<tr>
<td>6.13</td>
<td>Buoyant jet trajectory for $\Delta T = 50^\circ C$</td>
<td>48</td>
</tr>
<tr>
<td>6.14</td>
<td>Buoyant jet trajectory for $\Delta T = 20^\circ C$ and $\Delta T = 50^\circ C$ with and without corrected $\beta$</td>
<td>49</td>
</tr>
<tr>
<td>6.15</td>
<td>Normalized jet trajectory for $\Delta T = 20^\circ C$</td>
<td>50</td>
</tr>
<tr>
<td>6.16</td>
<td>Normalized jet trajectory for $\Delta T = 50^\circ C$</td>
<td>50</td>
</tr>
<tr>
<td>6.17</td>
<td>Normalized centerline dilution for $\Delta T = 20^\circ C$ and $\Delta T = 50^\circ C$</td>
<td>50</td>
</tr>
<tr>
<td>7.1</td>
<td>Surface temperature related to point of surface impingement</td>
<td>51</td>
</tr>
<tr>
<td>7.2</td>
<td>Momentum dominated mixing processes</td>
<td>52</td>
</tr>
<tr>
<td>7.3</td>
<td>Visualization of the jet trajectories for different outfall scenarios. The temperature plots for the minimum and maximum value of the variable are shown. Every plot has its own temperature scale. $\Delta T_0$ is indicated by the red color. The value of $\Delta T_0$ is different for Figures (e),(f),(i) and (j). All other plots have $\Delta T_0 = 10^\circ C$.</td>
<td>55</td>
</tr>
<tr>
<td>7.4</td>
<td>Visualization of the OpenFOAM results for the surface temperature rise $T_r$. Each plot shows the variation of the surface temperature rise for different initial jet conditions. The magnitude and the location of surface impingement are governed by the initial jet conditions.</td>
<td>56</td>
</tr>
<tr>
<td>7.5</td>
<td>Temperature profile over depth for different outfall design scenarios</td>
<td>57</td>
</tr>
<tr>
<td>7.6</td>
<td>Velocity profile over depth for different outfall design scenarios</td>
<td>58</td>
</tr>
<tr>
<td>7.7</td>
<td>The results of $\alpha = 0^\circ$ with the reduction factor $\cos(\alpha)$, compared to the model results of $\alpha = 30^\circ - 45^\circ$</td>
<td>59</td>
</tr>
<tr>
<td>7.8</td>
<td>Relations between dimensionless groups of parameters for the distances $X = 50 m$, $X = 80 m$ and $X = 100 m$</td>
<td>61</td>
</tr>
<tr>
<td>7.9</td>
<td>Standard logistic distribution</td>
<td>62</td>
</tr>
<tr>
<td>7.10</td>
<td>Logistic distribution fit to model results</td>
<td>63</td>
</tr>
<tr>
<td>7.11</td>
<td>Results of run 1-20 with 0.3 $^\circ C$ upper and lower bounds</td>
<td>64</td>
</tr>
<tr>
<td>7.12</td>
<td>Results of run 31-36 with 0.3 $^\circ C$ upper and lower bounds</td>
<td>64</td>
</tr>
<tr>
<td>7.13</td>
<td>Initial value parameter cube</td>
<td>65</td>
</tr>
<tr>
<td>7.14</td>
<td>Performance of equation for extreme value scenarios</td>
<td>66</td>
</tr>
<tr>
<td>7.15</td>
<td>Equation results compared to model results</td>
<td>66</td>
</tr>
<tr>
<td>7.16</td>
<td>Outfall scenario 35</td>
<td>67</td>
</tr>
<tr>
<td>8.1</td>
<td>Different submergence resulting in different buoyant surface plumes</td>
<td>70</td>
</tr>
<tr>
<td>C.1</td>
<td>Sensitivity study for the Turbulent Prandtl number, $Pr_t$</td>
<td>93</td>
</tr>
<tr>
<td>D.1</td>
<td>Development of velocity near the jet orifice</td>
<td>95</td>
</tr>
<tr>
<td>D.2</td>
<td>Development of temperature near the jet orifice</td>
<td>95</td>
</tr>
<tr>
<td>D.3</td>
<td>Development of pressure near the jet orifice</td>
<td>96</td>
</tr>
<tr>
<td>D.4</td>
<td>Development of turbulent kinetic energy near the jet orifice</td>
<td>96</td>
</tr>
<tr>
<td>D.5</td>
<td>Development of turbulent dissipation near the jet orifice</td>
<td>96</td>
</tr>
<tr>
<td>E.1</td>
<td>Temperature profile over depth for different outfall design scenarios</td>
<td>98</td>
</tr>
<tr>
<td>E.2</td>
<td>Velocity profile over depth for different outfall design scenarios</td>
<td>99</td>
</tr>
<tr>
<td>F.1</td>
<td>OpenFOAM result of surface temperature rise for variable $U_0$ and $D$, with constant $Q$ and $\Delta T_0$</td>
<td>102</td>
</tr>
<tr>
<td>F.2</td>
<td>Same outflow discharge and temperature, different diameter and outflow velocities</td>
<td>102</td>
</tr>
</tbody>
</table>
List of Tables

5.1 Boundary condition types for variable input parameters ................................................. 34
5.2 Boundary condition types for turbulent input parameters ................................................. 34
5.3 Seawater (S=35PSU) densities for varying temperatures according to [Gill, 1982] equation of state and the Boussinesq approximation with $T_{ref} = 20^\circ C$ ................................................. 38
5.4 All base scenarios ............................................................................................................ 39
5.5 Base scenarios with corresponding densimetric Froude number and momentum length scale ................................................................................................................................. 40
7.1 Input values for scenarios 31-36 ....................................................................................... 63
7.2 Extreme value scenarios ................................................................................................. 65
A.1 SIMPLE algorithm [OpenFOAMWiki, September 2013] .................................................. 89
F.1 Model runs with constant $\Delta T_0$ and $Q$ ....................................................................... 102
F.2 Quick Assessment Tool .................................................................................................... 103
Appendix
Appendix A

OpenFOAM numerical process

SIMPLE algorithm

SIMPLE algorithm stands for Semi-Implicit Method for Pressure-Linked Equation algorithm. This is a steady state algorithm, which follows an iterative procedure for solving the pressure and velocity coupling. Essentially the SIMPLE algorithm is a guess-and-correct procedure for calculating the pressure. The pressure field is initiated by the boundary conditions to a guessed pressure field, \( p^* \). Following, the discretized momentum equations are solved so that the intermediate velocity field is computed. After that, the pressure corrector, \( p' \) is defined to calculated the correct pressure. Under-relaxation is used to reduce the changes per iteration and prevent the pressure correction equation from diverging [OpenFOAM-UserGuide, 2012].

\[
p = p^* + p'
\]  

(A.0.1)

This procedure is similar for the velocity fields.

<table>
<thead>
<tr>
<th>SIMPLE algorithm</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Set the boundary conditions</td>
</tr>
<tr>
<td>2. Solve for the discretized momentum equation to compute the intermediate velocity field</td>
</tr>
<tr>
<td>3. Compute the mass fluxes at the cells faces</td>
</tr>
<tr>
<td>4. Solve the pressure equation and apply under-relaxation</td>
</tr>
<tr>
<td>5. Correct the mass fluxes at the cell faces</td>
</tr>
<tr>
<td>6. Correct the velocities on the basis of the new pressure field</td>
</tr>
<tr>
<td>7. Update the boundary conditions</td>
</tr>
<tr>
<td>8. Repeat till convergence</td>
</tr>
</tbody>
</table>

Table A.1: SIMPLE algorithm [OpenFOAMWiki, September 2013]
OpenFOAM Solver

For this thesis the OpenFOAM solver buoyantBoussinesqSimpleFoam is used. OpenFOAM has almost 80 pre-programmed solvers. Next to that the user is able to create or modify the solvers to its specific needs. The buoyantBoussinesqSimpleFoam solver is a advanced steady-state solver based on the SIMPLE algorithm for a single-phased model. This solver is specified for buoyant, turbulent flow of incompressible fluids. While the SIMPLE algorithm loop is running, the solver code includes the velocity equation, temperature equation and pressure equation. Thereafter, the turbulence variables are corrected and the solver is executed over the given execution time [OpenFOAM-UserGuide, 2012]. Applying a single-phase model means that the total domain consists of one specific medium of fluid. An open-water to air surface needs to be computed by a multi-phase solver. However, water level changes are expected to be very small and expected to have no significant influence on the model results. The single-phase model is computational much less expensive than the multi-phase models.
Appendix B

$k$ — $\epsilon$ turbulence closure

For the RANS equation given in equation 4.4.1, there are more unknowns than equations. This closure problems arises in the RANS equations. A turbulence closure model is necessary to model and compute the Reynolds stresses. Launder and Spalding [1974] developed the standard $k$ — $\epsilon$ model. This is a two-equation turbulence model to represent the turbulent properties of the flow. Boussinesq proposed the concept of eddy viscosity, where turbulence stresses are related to the mean flow [Versteeg and Malalasekera, 2007].

\[ -u_i u_j = \nu_t \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} k \]  \hspace{1cm} (B.0.1)

The eddy viscosity $\nu_t$ is represented by:

\[ \nu_t = C_{\mu} \frac{k^2}{\epsilon} \]  \hspace{1cm} (B.0.2)

Where the model constant $C_{\mu} = 0.09$. For the $k$ — $\epsilon$ model the transport equations are given for the kinetic turbulent energy $k$ and for the dissipation of turbulent kinetic energy, $\epsilon$. Both are defined as [Versteeg and Malalasekera, 2007]:

\[ k = \frac{1}{2} \left( \overline{u'v'} \right) \]  \hspace{1cm} (B.0.3)

\[ \epsilon = \nu \frac{\partial u'}{\partial y} \frac{u'}{\partial y} \]  \hspace{1cm} (B.0.4)

The final equations for the two properties $k$ and $\epsilon$ take the final form:

\[ \frac{\partial k}{\partial t} + m \frac{\partial k}{\partial x_j} = \nu_t \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial u_j} + \frac{\partial}{\partial x_j} \left[ \nu + \frac{\nu_t}{\sigma_k} \frac{\partial k}{\partial x_j} \right] - \epsilon \]  \hspace{1cm} (B.0.5)
\[
\frac{D\epsilon}{Dt} = C_{\epsilon 1} \frac{\epsilon}{k} \left( - \frac{u_i'u_j'}{\epsilon} \right) \frac{\partial u_i}{\partial x_j} + \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{\nu}{\sigma_{\epsilon}} \right) \frac{\partial \epsilon}{\partial x_j} \right] - C_{\epsilon 2} \frac{\epsilon^2}{k}
\] (B.0.6)

Where the eddy viscosity \( \nu_t \) is expressed in equation B.0.4 and the empirically based model constants for a wide range of turbulent flows for the standard \( k - \epsilon \) are given by:

\[ C_{\epsilon 1} = 1.44, \quad C_{\epsilon 2} = 1.92 \]

\[ \sigma_k = 1.0, \quad \sigma_{\epsilon} = 1.3 \]

As all other transport properties, boundary conditions have to be specified for both \( k \) and \( \epsilon \). For inlets, like jets, the inlet distribution for \( k \) and \( \epsilon \) can be obtained from [Versteeg and Malalasekera, 2007]

\[ k = \frac{3}{2} (U_0 I)^2 \] (B.0.7)

With the initial turbulence intensity \( I \) [%] and initial velocity \( U_0 \). And for \( \epsilon \):

\[ \epsilon = C^2_\mu \left( \frac{k^2}{I} \right) \] (B.0.8)

With the initial mixing length \( l = 0.07D_0 \) [Versteeg and Malalasekera, 2007].
Appendix C

Sensitivity of OpenFOAM input parameters

C.0.1 Turbulent Prandtl number

In section 4.6 an introduction to the turbulent Prandtl number is given. For the buoyantBoussinesqSimpleFoam solver, the turbulent Prandtl number needs to be given as a constant value. From experimental data it is found that the turbulent Prandtl number, $Pr_t$, varies between 0.7 and 0.9. In this section the sensitivity of the solution to the turbulent Prandtl number is tested for the range 0.7 - 0.9. The result on the surface temperature profile is given in figure C.1. It is found that the maximum surface temperature difference is $0.25^\circ C$. Therefore, it can be concluded that the surface temperature rise is not very sensitive to the turbulent Prandtl number. For the whole range of experimentally found values, the maximum temperature difference is considered small.

![Figure C.1: Sensitivity study for the Turbulent Prandtl number, $Pr_t$.](image)

C.0.2 Surface heat flux

The surface heat flux of the model is chosen zero. This assumes an isolated boundary at the surface where no heat can leave the domain. To test if this is a valid assumption, the convective heat transport is analytically calculated and compared to the amount of heat present in a specific volume of water. The equation for convection is given by equation C.0.1. The magnitude of the heat transfer coefficient
APPENDIX C. SENSITIVITY OF OPENFOAM INPUT PARAMETERS

depends on the relative velocity between the water and the air. If the magnitude of the current is considered small, the velocity difference is governed by the local wind velocity. This will result in a forced convective transport. If no velocity differences are considered, the transport is called a free convective transport. A relationship exists for the heat transfer coefficient as a function of wind velocity [Randall and Osczevski, 1995]. This relation is given by equation C.0.2. Where $v$ is the wind speed applicable for $v = 2 - 20 m/s$.

$$q = h_c \ A \ \Delta T \quad \text{(C.0.1)}$$

Where:

- $q =$ heat transferred per unit time ($W = J/s$)
- $A =$ heat transfer area ($m^2$)
- $h_c =$ convective heat transfer coefficient ($W/(m^2 K)$)
- $\Delta T =$ difference between water surface temperature and air temperature

$$h_c = 10.45 - v + 10v^{1/2} \quad \text{(C.0.2)}$$

To test if the assumption of zero heat flux is valid, the magnitude of heat transfer is calculated and compared to the total heat available in the surface. Two cases are considered. A free and a forced convective transport with wind conditions respectively taken as $v = 0 m/s$ and $v = 10 m/s$. The water surface temperature is chosen $T_s = 22^\circ C$ and the air temperature $T_a = 15^\circ C$. The surface considered is $A = 1m^2$. The calculated heat transport for free convection is:

$$q = 15 \cdot 1 \cdot 7 = 120 J/s \quad \text{(C.0.3)}$$

For forced convection:

$$q = 40 \cdot 1 \cdot 7 = 320 J/s \quad \text{(C.0.4)}$$

To lower the temperature of $1m^3$ of water with $1^\circ C$, the amount of transferred heat needs to be $4180kJ$. If only the top $10cm$ is considered, $418kJ$ of heat needs to be transferred through the surface. For the forced convection this would lead to $t = 1306s$ to cool down a $1m^2 \cdot 0.1m$ volume of water by $1^\circ C$. However, the jet causes a surface current varying from $U_s = 0.2 - 1m/s$. If $U_s = 0.2m/s$ is used, the contact time of the surface water within 100 meter of water would be $t = 500s$, causing the surface temperature to decrease by $T = 0.5^\circ C$. Because, this value is obtained by means of a conservative approach, the general conclusion is that the zero flux assumption is valid.
Appendix D

Jet outflow near orifice

At the point where the jet enters the ambient fluid, large gradients in velocity, temperature and pressure take place. The plots of the OpenFOAM simulation near the jet orifice are given below. The initial outflow velocity $U_0 = 3\text{ m/s}$, the outflow temperature difference $\Delta T_0 = 10^\circ\text{C}$ with an outfall diameter of $D = 2\text{ m}$. The velocity and temperature profile look similar, which is also expected.

Figure D.1: Development of velocity near the jet orifice

Figure D.2: Development of temperature near the jet orifice

Figure D.3 clearly shows the pressure gradient over the jet cross sectional plane. The pressure in the center of jet in the zone of flow establishment is equal to the dynamic pressure term $q = \frac{1}{2}\rho u^2$. This
pressure gradient results in a flow directed towards the jet. This results in the entrainment of the ambient fluid, which causes the jet to grow.

Figure D.4 and D.5 give the turbulent kinetic energy and the dissipation of the turbulent kinetic energy. The turbulent shear layer is clearly visible in figure D.4. The core of the jet in the zone of flow establishment does not include any turbulence and reaches up to approximately six times the jet diameter.

Figure D.3: Development of pressure near the jet orifice

Figure D.4: Development of turbulent kinetic energy near the jet orifice

Figure D.5: Development of turbulent dissipation near the jet orifice
Appendix E

Vertical temperature and velocity profiles for $X=50m$

The vertical temperature and velocity profiles for $X=100m$ have been used in the main report. In this appendix the results are presented for $X=50m$. These results are used to support the description of the temperature surface results in section 7.1.
APPENDIX E. VERTICAL TEMPERATURE AND VELOCITY PROFILES FOR X=50M

\( (a) \ U_0 = 1\ m/s, \ U_0 = 5\ m/s \)

\( (b) \ d = 3\ m, \ d = 15\ m \)

\( (c) \ D = 1\ m, \ D = 2.5\ m \)

\( (d) \ \Delta T_0 = 5^\circ C, \ \Delta T_0 = 20^\circ C \)

\( (e) \ U_0 = 6\ m/s \ with \ \Delta T_0 = 5^\circ C, \ U_0 = 2\ m/s \ with \ \Delta T_0 = 20^\circ C \)

\( (f) \ a = 0^\circ, a = 30^\circ \)

Figure E.1: Temperature profile over depth for different outfall design scenarios
APPENDIX E. VERTICAL TEMPERATURE AND VELOCITY PROFILES FOR X=50M

(a) $U_0 = 1\text{ m/s}$, $U_0 = 5\text{ m/s}$

(b) $d = 3\text{ m}$, $d = 15\text{ m}$

(c) $D = 1\text{ m}$, $D = 2.5\text{ m}$

(d) $\Delta T_0 = 5\text{ C}$, $\Delta T_0 = 20\text{ C}$

(e) $U_0 = 6\text{ m/s} \text{ with } \Delta T_0 = 5\text{ C}$, $U_0 = 2\text{ m/s} \text{ with } \Delta T_0 = 20\text{ C}$

(f) $a = 0^\circ$, $a = 30^\circ$

Figure E.2: Velocity profile over depth for different outfall design scenarios
Appendix F

Quick Assessment Tool

In the thesis report, the model results are validated, a parameter study is performed, subsequently the most important parameters are determined and a relation between the most important parameters and the backwater temperature rise is found. The equation proved to give very accurate results. In addition an applicability range for the equation is established. The equation is useful to quickly examine different outfall configurations without the need of complicated model studies. This equation needs to be part of an tool which can tell if outfall designs comply to the environmental restrictions. Many different options can be assessed within a short amount of time. The following equation, restriction and applicability range should be part of the Quick Assessment Tool:

\[ T_r = \cos(\alpha) \Delta T_0 \left( \frac{3}{1 + e^{-10(L_M/X)}} \right) \left( \frac{X}{D} \right)^{-0.72} \]

with \( L_M = \frac{U_0(\pi D_0^2/4)^{1/4}}{\sqrt{(\beta \Delta T) g}} \)

Applicable range and restriction:

\[ X > 1.4L_M \quad \text{and} \quad L_M < \frac{(Z - d)}{\sin(\alpha)} \quad \text{and} \quad \left( \frac{d}{D} \right) < 7.5 \]

Outfall design

Designing an cooling water outfall is an multidisciplinary procedure. The outfall temperature is depending on the amount of heat that needs to be lost in the LNG cooling process. The effectivity of this cooling process is also depending on the flow rate of the cooling water. The flow rate is governed by the amount of pumps and the pipe diameter. From an outfall designing perspective, the amount of heat and the flow rate are inputs which are given by the FLNG Process Engineers. This results in a constant flow rate \( Q \) and added temperature \( \Delta T_0 \). The variable design parameters at the outfall are the outflow velocity \( U_0 \), the outfall pipe diameter \( D \) and the outfall angle \( \alpha \). To obtain a constant discharge rate, many combinations of \( U_0 \) and \( D \) exist. Figure F.2 illustrates the difference in surface contact and entrainment rates for constant outflow discharges. Table F.1 gives the input values of the scenarios.

Figure F.1 shows the surface temperature rise for the different scenarios in Table F.1. The different combinations of outflow velocity and pipe diameter give significantly different results. With low outflow velocities, the turbulence intensity is low, resulting in less turbulent entrainment of the ambient fluid.
APPENDIX F. QUICK ASSESSMENT TOOL

This result is visible for scenario 45. The lowest outflow velocity results in the highest temperature rise over the whole surface. The largest outflow velocity results in the largest decrease of surface temperature. The large velocity gradients result in a larger turbulence intensities. This causes the jet to entrain more of the ambient fluid. Besides the increased turbulence, another effect plays an important role. The large outflow velocities result in a large momentum length scales. Therefore, the jet deflection towards the surface is less strong. This results in a longer jet trajectory through the ambient water. Therefore, increasing the jet velocity results in an increase of both turbulent entrainment and of the jet trajectory through the ambient fluid. The outfall temperature diffuses more efficiently with large outflow velocities.

Figure F.1: OpenFOAM result of surface temperature rise for variable $U_0$ and $D$, with constant $Q$ and $\Delta T_0$

Figure F.2: Same outflow discharge and temperature, different diameter and outflow velocities

<table>
<thead>
<tr>
<th>Run</th>
<th>$U_0$ [m/s]</th>
<th>$d$ [m]</th>
<th>$\Delta T_0$ [°C]</th>
<th>$D_0$ [m]</th>
<th>$\alpha$ [°]</th>
<th>$Q$ [m$^3$/s]</th>
<th>$L_M$ [m]</th>
</tr>
</thead>
<tbody>
<tr>
<td>45</td>
<td>2.25</td>
<td>6</td>
<td>10</td>
<td>2.5</td>
<td>0</td>
<td>11</td>
<td>19.5</td>
</tr>
<tr>
<td>46</td>
<td>3.5</td>
<td>6</td>
<td>10</td>
<td>2</td>
<td>0</td>
<td>11</td>
<td>27.2</td>
</tr>
<tr>
<td>47</td>
<td>6.2</td>
<td>6</td>
<td>10</td>
<td>1.5</td>
<td>0</td>
<td>11</td>
<td>41.7</td>
</tr>
</tbody>
</table>

Table F.1: Model runs with constant $\Delta T_0$ and $Q$

The Quick Assessment Tool consists of all the gathered insights, information and relations found in this thesis. Table F.2 gives an indication of the Quick Assessment Tool. The input for the tool consists of the flow rate $Q$, initial temperature difference $\Delta T_0$, the location $X$ where user wants to know the surface temperature rise and the outfall angle $\alpha$. These parameters need to be chosen by the designer of the specific outfall. The fixed input parameters are used to determine the restriction for bottom attachment. The design options provide several outfall diameters in combination with outflow velocities. This design choice influences the temperature rise $T_r$ at the given location $X$. In addition the range of applicability
and outfall restrictions are calculated. The Quick Assessment Tool is build in Microsoft Excel and color scaling are used to indicate if the results are within all ranges (green), come close to the given ranges (orange) or exceed the ranges (red).

<table>
<thead>
<tr>
<th>Variable input</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Q</td>
<td>15 m³/s</td>
</tr>
<tr>
<td>ΔT₀</td>
<td>10 °C</td>
</tr>
<tr>
<td>X</td>
<td>100 m</td>
</tr>
<tr>
<td>α</td>
<td>10 °</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Fixed input</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Z</td>
<td>30 m</td>
</tr>
<tr>
<td>d</td>
<td>6 m</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Design options</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>U</td>
<td>1 2 3 4 5 6 m/s</td>
</tr>
<tr>
<td>D</td>
<td>4.4 3.1 2.5 2.2 2.0 1.8 m</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Temperature rise</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Tr</td>
<td>1.68 1.78 1.79 1.74 1.66 1.59 °C</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Range of applicability</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Lₘ/X</td>
<td>0.11 0.19 0.26 0.32 0.38 0.44 &lt; 0.7</td>
</tr>
<tr>
<td>Lₘ/L₀</td>
<td>0.08 0.14 0.19 0.23 0.28 0.32 &lt; 1</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Pipe diameter</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td># pipes</td>
<td>5</td>
</tr>
<tr>
<td>D per pipe</td>
<td>2.21 1.56 1.27 1.10 0.99 0.90</td>
</tr>
<tr>
<td>Lₘ</td>
<td>11.47 19.29 26.15 32.45 38.36 43.98</td>
</tr>
<tr>
<td>L₀</td>
<td>138.21</td>
</tr>
</tbody>
</table>

*Table F.2: Quick Assessment Tool*
Appendix G

OpenFOAM input files

The most important files from the OpenFOAM setup are given below. This improves the possibilities to reproduce and extend the thesis results. In addition, this can be valuable files for future studies.

Boundary- and initial conditions file, velocity U

```
FoamFile
{
    version 2.0;
    format ascii;
    class volVectorField;
    object U;
}

// * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * * *
dimensions [0 1 -1 0 0 0 0];
internalField uniform (0 0 0);

boundaryField
{
    bottom_sym
    {
        type slip;
    }
    flng_sym
    {
        type fixedValue;
        value uniform (0 0 0);
    }
    inlet_1m
    {
        type fixedValue;
        value uniform (3 0 0);
    }
    inlet_2_5m
    {
        type fixedValue;
        value uniform (3 0 0);
    }
    side_sym
    {
        type pressureInletOutletVelocity;
        value uniform (0 0 0);
    }
    surface_sym
    {
        type slip;
    }
    sym_plan
    {
        type symmetryPlane;
    }
    wall_outlet_sym
```
APPENDIX G. OPENFOAM INPUT FILES

{  
  type inletOutlet;
  inletValue uniform (0 0 0);
  value uniform (0 0 0);
}

// ************************************************************************* //
fvSchemes

// ************************************************************************* //

transportProperties
APPENDIX G. OPENFOAM INPUT FILES

class dictionary;
object transportProperties;
}

/* *************************************************************************/
transportModel Newtonian;

// Laminar viscosity
nu
[0 2 -1 0 0 0] 1.05e-06; // kinematic visc

// Thermal expansion coefficient
beta
[0 0 0 1 0 0 0] 300e-06;

// Reference temperature
TRef
TRef [0 0 1 0 0 0] 293.15;

// Laminar Prandtl number
Pr
Pr [0 0 0 0 0 0] 7.2;

// Turbulent Prandtl number
Prt
Prt [0 0 0 0 0 0] 0.85;

Cp0 600;

/* *************************************************************************/

FoamFile
{
version 2.0;
format ascii;
class dictionary;
location "system";
object controlDict;
}

application buoyantBoussinesqSimpleFoam;
startFrom latestTime;
startTime 0;
stopAt endTime;
endTime 7200;
deltaT 0.2;
writeControl timeStep;
writeInterval 2500;
purgeWrite 0;
writeFormat ascii;
writePrecision 6;
writeCompression off;
timeFormat general;
timePrecision 6;
runTimeModifiable true;

functions
{
probes

type probes;
functionObjectLibs ("libsampling.so");
}

/* *************************************************************************/
APPENDIX G. OPENFOAM INPUT FILES

(0.5 5 30)
(100 5 30)
(150 5 30)
); fields
{
    T
    U
    }
};

// ********************************************************************************

// ******************************************************************************