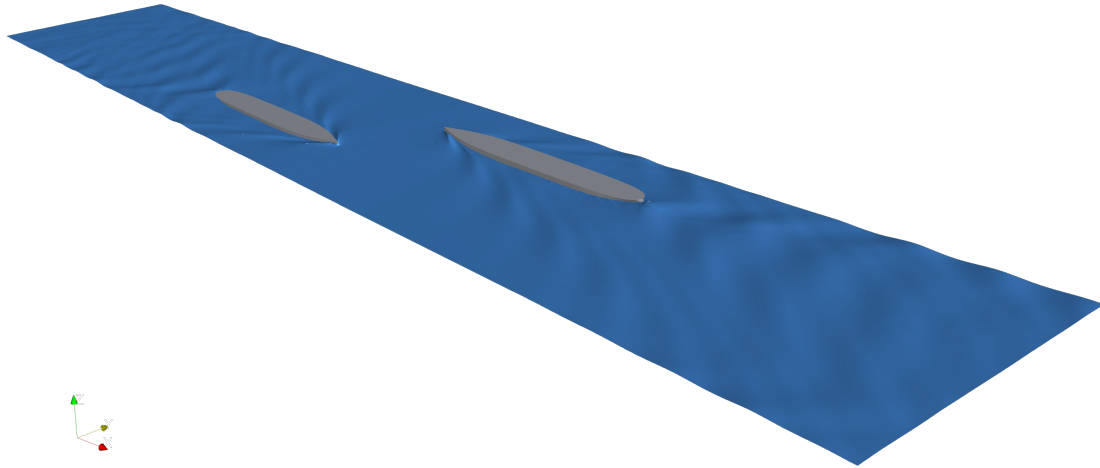




CHALMERS
UNIVERSITY OF TECHNOLOGY



Hydrodynamic Interaction and Steering of Ships in Confined Waterways using Overset Mesh

Master's thesis in Applied Mechanics

Zahran Saeed Ghazy

DEPARTMENT OF MECHANICS AND MARITIME SCIENCES

CHALMERS UNIVERSITY OF TECHNOLOGY
Gothenburg, Sweden 2026
www.chalmers.se

MASTER'S THESIS IN APPLIED MECHANICS

Hydrodynamic Interaction and Steering of Ships in Confined Waterways using Overset Mesh

Zahran Saeed Ghazy



CHALMERS
UNIVERSITY OF TECHNOLOGY

Department of Mechanics and Maritime Sciences
Division of Fluid Dynamics
CHALMERS UNIVERSITY OF TECHNOLOGY
Gothenburg, Sweden 2026

Hydrodynamic Interaction and Steering of Ships in Confined Waterways using
Overset Mesh
Zahran Saeed Ghazy

© Zahran Saeed Ghazy 2026.

Supervisor: Håkan Nilsson, Department of Mechanics and Maritime Sciences
Examiner: Håkan Nilsson, Department of Mechanics and Maritime Sciences

Master's Thesis 2026
Department of Mechanics and Maritime Sciences
Chalmers University of Technology
SE-412 96 Gothenburg
Sweden
Telephone +46 31 772 1000

Cover: ParaView visualization showing the free surface wave elevations as two ship hulls approach each other in a confined channel.

Typeset in L^AT_EX
Gothenburg, Sweden 2026

Hydrodynamic Interaction and Steering of Ships in Confined Waterways using Overset Mesh

Zahran Saeed Ghazy

Department of Mechanics and Maritime Sciences

Division of Fluid Dynamics

Chalmers University of Technology

Abstract

Safe navigation in confined waterways, such as narrow channels, is a challenging aspect of maritime operations, particularly when two vessels meet or pass each other at close distance. In these situations, hydrodynamic interaction effects between the ships and the surrounding channel boundaries can significantly influence the vessels' motion, maneuverability, and steering behavior. If these effects are not properly anticipated, they may compromise the safety of such encounters.

This thesis investigates the applicability of Computational Fluid Dynamics for simulating ship encounters in confined waterways. The work focuses on the use of OpenFOAM together with overset mesh and the Volume of Fluid method to model free-surface flow and moving vessels. A numerical simulation strategy is developed in several stages, starting from a single-hull case in a channel and progressing toward a two-hull meeting scenario. The aim is to assess whether the selected numerical methods can capture relevant hydrodynamic interaction effects and support the development of steering strategies for safe passage, using only accessible local data such as the ship's yaw angle and distance to obstacles. The purpose is to contribute to the development of steering methods that reduce reliance on external satellite-based navigation systems such as GPS.

Keywords: Computational Fluid Dynamics, Confined Waterways, Hydrodynamic Interaction, OpenFOAM, Overset Mesh, Ship Steering, Volume of Fluid

Preface

This report presents the outcome of my master’s thesis project carried out at the Department of Mechanics and Maritime Sciences at Chalmers University of Technology during the spring of 2026.

Acknowledgements

I would like to express my gratitude to everyone who has supported me throughout my education at Chalmers. I am especially grateful to my family, friends, teachers, and colleagues who have contributed to making these years both educational and memorable.

I would like to sincerely thank Håkan Nilsson, for giving me the opportunity to undertake this thesis. His courses provided an important foundation for this work and gave me the opportunity to become familiar with computational fluid dynamics and OpenFOAM. I am very grateful for his guidance, insightful feedback, and support throughout the project, especially during the more challenging parts of the work. I have learned a great deal during this thesis, and this project would not have been possible without his help.

I would also like to thank Rickard Bensow for providing the Lucy Ashton hull geometry and for his help with the meshing process and general technical guidance during the early stages of the project. I am also grateful to Ola Benderius for his support and valuable input during the development of the steering strategies.

A special thanks goes to Eeshan Ganla, Nils Strömö, Najmeh Asadimoghaddam, Liang Wang, Bala Kumares, and the rest of my colleagues and friends at the department, for making the office a welcoming and enjoyable place to work. The daily discussions, support, and good company has enriched this experience.

Finally, I want to thank my family. Their unwavering belief in me and their constant support throughout my education have been my anchor. None of this would have been possible without them.

Zahran Saeed Ghazy, Gothenburg, June 2026

The computations were enabled by resources provided by the National Academic Infrastructure for Supercomputing in Sweden NAISS, partially funded by the Swedish Research Council through grant agreement no. 2022-06725.

List of Acronyms

Below is the list of acronyms that have been used throughout this thesis listed in alphabetical order:

AIM	Aim-Point Steering Case
CAD	Computer Aided Design
CFD	Computational Fluid Dynamics
CoR	Center of Rotation
CPU	Central Processing Unit
DILU	Diagonal Incomplete LU preconditioner
DOF	Degree of Freedom
GPS	Global Positioning System
ITTC	International Towing Tank Conference
MULES	Multidimensional Universal Limiter for Explicit Solution
NAISS	National Academic Infrastructure for Supercomputing in Sweden
NSC	National Supercomputer Center
PBiCGStab	Preconditioned Bi-Conjugate Gradient Stabilized solver
PID	Proportional-Integral-Derivative controller
PIMPLE	Combined PISO-SIMPLE pressure-velocity coupling algorithm
PISO	Pressure-Implicit with Splitting of Operators
RANS	Reynolds-Averaged Navier-Stokes
SIMPLE	Semi-Implicit Method for Pressure Linked Equations
UKC	Under Keel Clearance
VOF	Volume of Fluid

Nomenclature

Below is the nomenclature of indices, symbols, and subscripts that have been used throughout this thesis.

Indices

i, j	Spatial coordinate indices (index notation); i also used as donor cell counter in overset interpolation
--------	---

Symbols

B	Ship beam
B_{ch}	Channel width
c_α	Interface compression coefficient (VOF)
C_μ	Turbulence model coefficient
d_i	Distance between cell centers (overset interpolation)
$\mathbf{e}_x$	Streamwise unit vector
$\mathbf{f}_i$	Body force vector
F_x	Streamwise force
F_y	Lateral force
Fr	Froude number
Fr_h	Depth-dependent Froude number
g	Gravitational acceleration
h	Water depth
h_{ref}	Reference height used in the pseudo-hydrostatic pressure definition
H	Hull height
I	Turbulence intensity
k	Dimensionless aim-point steering gain; also used for turbulent kinetic energy depending on context
$K_{D,l}$	Derivative gain on sway (PID)

$K_{D,\psi}$	Derivative gain on yaw (PID)
K_I	Integral gain (PID)
$K_{P,l}$	Proportional gain on sway (PID)
$K_{P,\psi}$	Proportional gain on yaw (PID)
l	Lateral displacement (sway)
$\dot{l}$	Sway velocity (time derivative of l)
L	Look-ahead distance (aim-point steering)
L_{ch}	Channel length
L_f	Full scale ship length
L_m	Model scale ship length
L_{pp}	Length between perpendiculars
M_x	Roll moment
M_z	Yaw moment
n	Number of donor cells (overset stencil)
N_x	Number of cells in the x direction
N_y	Number of cells in the y direction
N_z	Number of cells in the z direction
p	Static pressure
p_{rgh}	Pseudo-hydrostatic pressure
q	Generalized coordinate (rigid body motion)
S	Wetted surface area
T	Ship draught
u_i	Velocity component in x_i -direction
U	Model scale ship speed
U_f	Full scale ship speed
U_s	Ship speed (general)
$\mathbf{U}$	Velocity vector
$\mathbf{U}_c$	Artificial compression velocity (VOF)
w_i	Interpolation weight (overset inverse distance method)
x_i	Cartesian coordinate direction
y	Lateral coordinate
y_{inf}	Bank influence width
z	Vertical coordinate
α	Phase fraction (volume fraction of water)
δ	Steering (rudder) angle
Δx	Streamwise cell size
Δy	Lateral cell size

Δz	Vertical cell size
ϵ	Turbulent dissipation rate
λ	Geometric scale factor
λ_k	Kelvin wake wavelength
λ_w	Free surface wavelength
μ	Dynamic viscosity
μ_a	Dynamic viscosity of air
μ_w	Dynamic viscosity of water
ν	Kinematic viscosity
ν_t	Turbulent viscosity
ϕ	Steering error angle (aim-point model)
Φ_k	Kelvin wake angle
ψ	Yaw angle
$\dot{\psi}$	Yaw rate (time derivative of ψ)
ρ	Density
ρ_a	Density of air
ρ_w	Density of water
σ	Surface tension coefficient
τ	Integration variable (time)

Subscripts

a	Air phase
cf	Channel-fixed reference frame
f	Full scale
hf	Hull-fixed reference frame
m	Model scale
w	Water phase

Contents

List of Acronyms	ix
Nomenclature	xi
1 Introduction	1
1.1 Background	1
1.2 Purpose	1
1.3 Goals	1
1.4 Limitations	2
2 Theory	3
2.1 Governing Equations	3
2.1.1 Continuity Equation	3
2.1.2 Momentum Equation	3
2.2 Turbulence Modeling	4
2.3 Pressure-Velocity Coupling	5
2.4 Volume of Fluid Method	6
2.5 Overset Mesh	7
2.6 The overInterDyMFoam Solver	9
2.7 Froude Similarity	10
2.8 Hydrodynamic Interaction Effects	10
3 Methods	13
3.1 Workflow Overview	13
3.2 Case Description	15
3.2.1 Hull Geometry and Model Scale	15
3.2.2 Channel Geometry	17
3.3 Computational Mesh	17
3.3.1 Background Mesh	17
3.3.1.1 Inlet Extension	19
3.3.2 Overset Mesh	21
3.4 Numerical methods	25
3.4.1 Turbulence Modeling and Fluid Properties	25
3.4.2 Pseudo-Hydrostatic Pressure	26
3.4.3 Numerical Schemes	26
3.4.4 Solver Settings	27
3.4.5 Under-Relaxation	28
3.4.6 Free Surface Treatment	28
3.5 Hull-Fixed Simulation	28

3.5.1	Boundary Conditions	29
3.5.2	Rigid-Body Motion Setup	30
3.6	Channel-Fixed Reference Frame Shift	30
3.6.1	Selection of Precursor Solution	31
3.6.2	Reference-Frame Transformation	31
3.6.3	Boundary Condition Modification	31
3.6.4	Rigid-Body Motion Setup	32
3.7	Two-hull encounter	33
3.7.1	Assembly of the Two-Hull Domain	33
3.7.2	Mapping of the Flow Field	33
3.7.3	Rigid Body Motion Setup	34
3.8	The Steering	34
3.8.1	Aim-Point Steering Model	35
3.8.2	PID Controller	36
3.9	Computational Resources	37
4	Results and Discussion	39
4.1	Hull-fixed Simulations	39
4.1.1	Inlet extension and wave reflections	39
4.1.2	Offset vs. Centered Hull	41
4.1.2.1	Pressure distribution	41
4.1.2.2	Hydrodynamic Forces and Moments	41
4.2	Channel-Fixed Simulations (Single Hull)	45
4.2.1	No Active Steering	45
4.2.2	Aim-Point Steering Model	47
4.2.3	PID	49
4.3	Channel-Fixed Simulations (Full Encounter)	51
4.3.1	Mapping	51
4.3.2	No Active Steering	53
4.3.3	Aim-point Steering Model	53
4.3.4	PID	56
5	Conclusion	59
	Bibliography	61

1

Introduction

1.1 Background

Safe navigation in narrow and confined waterways has long been a central challenge in maritime operations, particularly during encounters where vessels meet or pass each other at close distances. In such situations, ships experience hydrodynamic interaction, both with each other and with nearby banks, which can significantly influence maneuverability and steering behavior and may lead to unsafe situations if not properly anticipated.

Within an ongoing research project, there is an interest in developing steering strategies that support ship captains during encounters in confined waterways without relying on satellite-based navigation systems such as GPS. This interest is motivated by the vulnerability of GPS to external interference and by its dependence on infrastructure controlled by other countries. To support the development of such strategies, there is a need to investigate the applicability of numerical methods, both for capturing these hydrodynamic interactions and for developing steering methods.

Computational Fluid Dynamics (CFD) provides a suitable framework for studying these interaction effects in detail. Furthermore, the overset mesh method, also referred to as the Chimera mesh method, enables simulations involving multiple moving bodies and large relative motion, making it a strong candidate for modeling ship encounters in confined waterways.

From a societal and ethical perspective, the work contributes to maritime safety and efficiency by supporting the development of navigation assistance methods that reduce reliance on vulnerable positioning systems. Environmental aspects are indirectly addressed through the potential for safer and more efficient ship operations.

1.2 Purpose

The purpose of this project is to assess the applicability of the overset mesh method in OpenFOAM for developing steering strategies in confined waterways, using only local information such as the yaw angle and the distance to obstacles.

1.3 Goals

In order to fulfil the purpose of the project, the developed a framework needs to achieve the following goals:

1. Capture hydrodynamic interaction effects from banks.
2. Capture hydrodynamic interaction effects between ships.
3. Implement and evaluate steering strategies.

1.4 Limitations

Due to the limited available time for the project, 20 weeks, the following limitations and simplifications are made:

- The focus is on developing the method and workflow rather than on the quantitative accuracy of the results.
- The hull's rigid-body motion is restricted to five degrees of freedom, with roll held fixed to reduce complexity.
- The propeller and rudder are not included in the geometry. The propeller is replaced by an applied thrust force, and the rudder is represented by the angle at which this force is applied.
- An idealized channel is used, with no bathymetric variation.

2

Theory

This chapter presents the theoretical framework underlying the numerical simulations performed in this project. It introduces the prerequisites required to understand the OpenFOAM solver used in this project, which is explained in further detail at the end.

2.1 Governing Equations

In computational fluid dynamics, the fluid is treated as a continuous medium rather than a collection of discrete particles. This continuum assumption is valid for virtually all macroscopic fluids. Under this assumption the motion of a fluid is governed by coupled partial differential equations: the continuity equation and the momentum equations [1].

2.1.1 Continuity Equation

The continuity equation enforces conservation of mass at every point in space and is given as

$$\frac{d\rho}{dt} + \rho \frac{\partial u_i}{\partial x_i} = 0, \quad (2.1)$$

where u_i is the velocity component in the x_i -direction, and ρ is the fluid density. It states that the rate of change of mass in a fixed volume must be balanced by the net flow into the volume.

For incompressible fluids, the first term in Equation 2.1 disappears and the equation simplifies to

$$\frac{\partial u_i}{\partial x_i} = 0. \quad (2.2)$$

Since compressible effects are negligible for water and for air with Mach number below 0.3, this is the form of the continuity equation that will be used in this project.

2.1.2 Momentum Equation

The momentum equations describe the conservation of momentum and are often referred to as the *Navier-Stokes* equations. They represent Newton's second law of motion for a fluid element, and are given in index notation for incompressible fluids with constant viscosity as

$$\rho \left(\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} \right) = -\frac{\partial p}{\partial x_i} + \mu \frac{\partial^2 u_i}{\partial x_j \partial x_j} + f_i, \quad (2.3)$$

where p is the fluid pressure, ρ is the fluid density, μ is the fluid dynamic viscosity, and f_i represents body forces.

The first and second terms in the equations make up the total inertial forces and are balanced by the driving pressure forces, viscous forces, and external body forces. The terms in the equation are also referred to as the unsteady term, the convective term, the pressure gradient term, the viscous diffusion term and the external source term, respectively.

These equations, together with the continuity equation, appropriate boundary conditions, and initial conditions, fully describe the flow. However, for turbulent flows at high Reynolds numbers, resolving all scales of motion directly is computationally expensive and statistical treatment of turbulence is required, which is described in the next section.

2.2 Turbulence Modeling

In the Reynolds-Averaged Navier-Stokes (RANS) approach, the flow variables from the governing equations are decomposed into a mean component and a fluctuating component [2]

$$u_i = \bar{u}_i + u'_i, \quad (2.4)$$

where $\bar{u}_i$ denotes the time-averaged velocity and u'_i is the turbulent fluctuation. Substituting this decomposition into the Navier-Stokes equations and time-averaging yields the RANS equations

$$\rho \left(\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} \right) = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j} \left(\mu \frac{\partial \bar{u}_i}{\partial x_j} - \rho \overline{u'_i u'_j} \right) + f_i. \quad (2.5)$$

The additional term $\rho \overline{u'_i u'_j}$ is the Reynolds stress tensor, which arises from the non-linear convective term and represents the effect of turbulent fluctuations on the mean flow. Since this tensor introduces unknown terms, the system is no longer closed and a turbulence model is needed to relate the Reynolds stresses to the mean flow quantities.

The Boussinesq hypothesis assumes that the Reynolds stresses can be approximated from mean flow properties,

$$\overline{u'_i u'_j} = -\nu_t \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) + \frac{2}{3} k \delta_{ij}, \quad (2.6)$$

where ν_t is the eddy viscosity, and $k = \frac{1}{2} \overline{u'_i u'_i}$ is the turbulent kinetic energy.

This reduces the closure problem to determining ν_t , which is the role of eddy-viscosity turbulence models.

One such model is the k - ε model. It gets its name from the quantities it uses to determine ν_t : the turbulent kinetic energy k and its dissipation rate ε . The turbulent viscosity can then be modeled from these quantities as

$$\nu_t = C_\mu \frac{k^2}{\varepsilon}, \quad (2.7)$$

where C_μ is a model coefficient.

The k - ε model is well suited for turbulent flows away from solid boundaries, but performs poorly in near-wall regions where the turbulent Reynolds number is low and viscous effects dominate. To handle this limitation, wall functions are typically used.

2.3 Pressure-Velocity Coupling

The algorithm used in this project to handle the pressure-velocity coupling in the Navier-Stokes equations is the PIMPLE algorithm. PIMPLE is a combination of the PISO and SIMPLE algorithms that has an inner loop and an outer loop. The inner pressure correction loop corresponds to the PISO part of the algorithm, while the outer corrector loop corresponds to the SIMPLE part. The algorithm is widely used in OpenFOAM and forms the basis of several transient solvers [3, 1].

In a general PIMPLE iteration, the velocity is first calculated by solving the momentum equation using the available pressure field. Then, an intermediate velocity without the pressure gradient contribution is constructed together with its face flux, and a pressure equation, derived from the continuity equation, is solved. The resulting pressure is then used to correct the face fluxes and the velocity field. This sequence may be repeated through one or more inner correctors, similarly to the PISO algorithm [3].

After the inner pressure corrections have been completed, the algorithm may either proceed to the next time step or return to the beginning of the momentum-pressure sequence. This repetition is controlled by the outer corrector loop and is the SIMPLE-like part of the algorithm. The outer loop allows the coupled equations to be iterated several times within the same time step. In OpenFOAM, the number of outer correctors is controlled by `nOuterCorrectors`, while the number of inner pressure correctors is controlled by `nCorrectors`.

Figure 2.1 shows the simplified structure of the PIMPLE algorithm. Details such as mesh motion updates, Courant number evaluation and additional solver-specific operations are omitted from the figure.

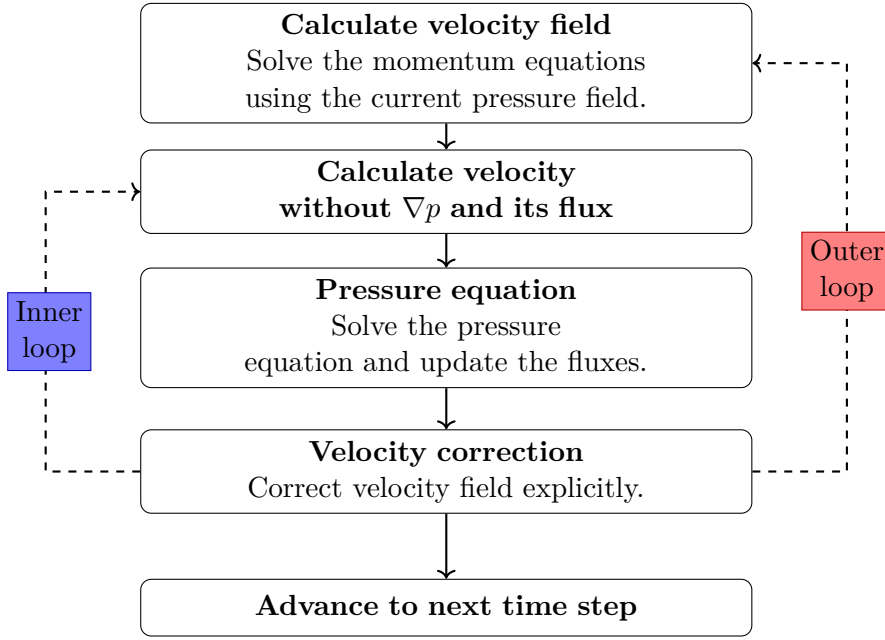


Figure 2.1: Simplified structure of the PIMPLE algorithm.

2.4 Volume of Fluid Method

Free surface flows involving two immiscible fluids, such as water and air, require a numerical method for representing the position and motion of the interface. In interface-capturing methods, the computational grid covers both fluids and the free surface is obtained from an additional transported scalar field. One such method is the Volume of Fluid (VOF) method, in which a phase fraction field, here denoted by α , is used to describe the fraction of each cell occupied by one of the phases [4].

In the present work, α denotes the water phase fraction. A cell fully occupied by water has $\alpha = 1$, a cell fully occupied by air has $\alpha = 0$, and the interface is located in cells where $0 < \alpha < 1$. The transport of the phase fraction is governed by an advection equation [1],

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot (\mathbf{U}\alpha) = 0. \quad (2.8)$$

The two fluids are treated as a single effective fluid whose properties vary according to the local value of α . The density and dynamic viscosity are therefore evaluated as

$$\rho = \alpha\rho_w + (1 - \alpha)\rho_a, \quad \mu = \alpha\mu_w + (1 - \alpha)\mu_a, \quad (2.9)$$

where the subscripts w and a denote water and air, respectively. This allows one set of momentum equations to be solved in the full domain, while the material properties change across the interface.

Although Equation (2.8) contains no physical diffusion term, numerical discretization of the phase fraction equation may introduce artificial smearing that degrades the interface over time. In OpenFOAM, an additional interface-compression contribution is therefore included in the phase fraction equation to counteract numerical smearing and maintain a

sharper interface [5]. The corresponding equation can be written in the form

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot (\mathbf{U}\alpha) + \underbrace{\nabla \cdot (\mathbf{U}_c \alpha(1 - \alpha))}_{\text{compression term}} = 0, \quad (2.10)$$

where $\mathbf{U}_c$ is an artificial compression velocity. The factor $\alpha(1 - \alpha)$ confines this contribution to the interface region, since it vanishes in cells containing only one of the phases. In OpenFOAM, the strength of the interface compression is controlled by the coefficient `cAlpha`; `cAlpha = 0` corresponds to no compression, `cAlpha = 1` to conservative compression, and values larger than one to enhanced compression [6]. Since α is a phase fraction, it must remain bounded between zero and one. OpenFOAM enforces this boundedness using the MULES algorithm, the Multidimensional Universal Limiter for Explicit Solution. MULES limits the phase fraction fluxes so that the transported scalar satisfies $0 \leq \alpha \leq 1$ [7, 6, 8, 9].

2.5 Overset Mesh

The overset mesh method, also referred to as the Chimera mesh method, allows the computational domain to be built from two or more separate meshes that are not connected by any matching faces. Instead, they are allowed to overlap, and the flow solution is transferred between them by interpolation. This makes the method useful for cases with large rigid body motion, where other dynamic mesh methods, such as mesh deformation, could lead to poor cell quality or require remeshing [10, 11, 12].

Conceptually, an overset mesh setup consists of one mesh for the main computational domain, called the background mesh, and one or more separate meshes for the objects, called overset meshes. This is illustrated in Figure 2.2. The part of the domain where the background mesh and the overset mesh occupy the same physical space is referred to as the overlap region.

After the meshes have been assembled, the cells are classified into three different cell types: calculated cells, interpolated cells, and hole cells. Calculated cells are active cells where the flow equations are solved directly. Interpolated cells, also called acceptor cells, do not obtain their values directly from the discretized equations. Instead, their values are interpolated from donor cells in the other mesh region. Blanked cells, also referred to as hole cells, are inactive cells and are excluded from the solution. Background cells that are covered by the overset body region become blanked cells, although the exact extent of this region depends on the hole cutting procedure and the overset settings used. The process of identifying the inactive cells is commonly referred to as hole cutting.

Interpolated cells exist on both the background mesh and the overset mesh. On the background mesh, the interpolation fringe is formed around the blanked region, and on the overset mesh, the interpolation fringe is located along the outer boundary of the overset mesh. These fringe cells provide the connection between the two mesh regions. The remaining cells are calculated cells.

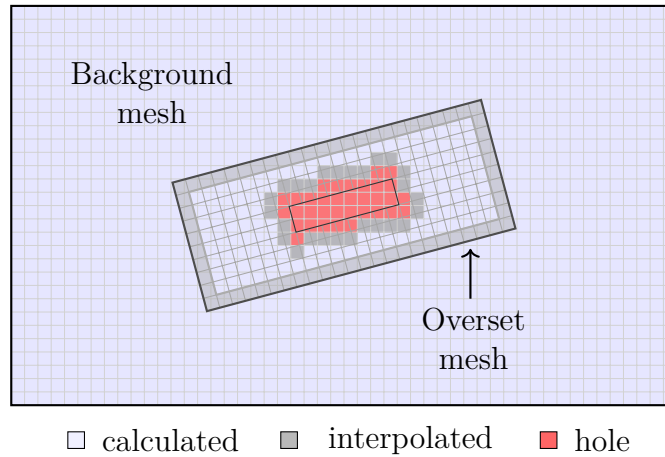


Figure 2.2: Schematic illustration of an overset mesh placed inside a background mesh. The red cells are blanked background cells that overlap with the overset mesh body. A background interpolation fringe surrounds the blanked region, while a second interpolation fringe is located along the outer edge of the overset mesh. The remaining cells are calculated cells.

The placement of the interpolation fringes is an important part of the overset method. Since the values in the fringe cells are obtained by interpolation, the interpolation should preferably take place in regions with small gradients. The overset mesh should therefore be designed so that the interpolation fringes are kept away from the most sensitive parts of the flow when possible.

The quality of the overlap region is also important. The overlap must be wide enough for each interpolated cell to find suitable donor cells in the neighboring mesh region. A common practical recommendation is that the overlap region should contain at least five cells from each mesh in the overlap direction [13, 14]. In addition, the cell sizes in the donor and acceptor regions should be comparable. Large differences in cell size between the background and overset meshes can reduce the accuracy of the interpolation, since information is then transferred between regions with different spatial resolution. For this reason, the outer part of the overset mesh and the corresponding refined region in the background mesh are usually designed to have similar cell sizes [15, 14].

In OpenFOAM, the position of the background interpolation fringe can be influenced using the `holeLayers` and `useLayer` entries for the `inverseDistance` overset interpolation method. The parameter `holeLayers` defines how many cell layers are considered away from the hole region, while `useLayer` selects which layer is used for interpolation. These settings can therefore be used to move the interpolation fringe away from the immediate vicinity of the blanked region [16].

For this project, OpenFOAM's `inverseDistance` method is used for the overset interpolation. The method constructs an interpolation stencil for each interpolated cell and computes the cell value from the corresponding donor cell values. For a transported quantity ϕ , this can be written as

$$\phi_A = \sum_{i=1}^n w_i \phi_i, \quad (2.11)$$

where ϕ_A is the interpolated value, ϕ_i is the value in donor cell i , w_i is the interpolation weight, and n is the number of donor cells in the stencil [11, 17]. The weights are determined from the distance between the interpolated cell center and the donor cell centers, see Figure 2.3. Let $\mathbf{x}_A$ be the center of the interpolated cell and $\mathbf{x}_i$ the center of donor cell i . The distance is then

$$d_i = |\mathbf{x}_A - \mathbf{x}_i|. \quad (2.12)$$

The contribution from each donor cell is then weighted by the inverse of this distance, so that closer donor cells have a larger influence on the interpolated value. The normalized weight is given by

$$w_i = \frac{1/d_i}{\sum_{j=1}^n 1/d_j}. \quad (2.13)$$

Substitution into Equation (2.11) gives

$$\phi_A = \sum_{i=1}^n \frac{1/d_i}{\sum_{j=1}^n 1/d_j} \phi_i. \quad (2.14)$$

If a valid donor stencil cannot be found, the corresponding cell is classified as an orphan cell. Orphan cells cannot receive data from a donor mesh and are commonly associated with insufficient overlap between overset grids [17]. Their occurrence indicates a local failure of the overset connectivity and should be avoided.

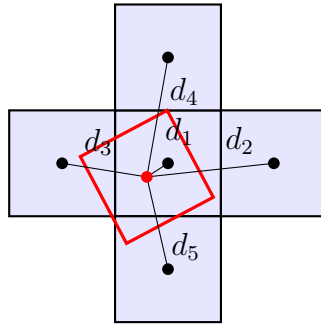


Figure 2.3: Distance between acceptor cell center and donor cell centers. Black: donor cell centers. Red: acceptor cell center.

A special case occurs when part of an overset mesh moves outside the background mesh. In OpenFOAM, overset patches are allowed to extend outside the background domain. Cells located outside all other meshes cannot be interpolated because no donor candidates are available. These cells can instead be treated as *porous* cells, which are solved locally rather than by interpolation [16].

2.6 The overInterDyMFoam Solver

The simulations in this work are performed using OpenFOAM's `overInterDyMFoam` solver. The solver is based on the `interFoam` family of solvers, which are transient solvers for two incompressible, isothermal and immiscible fluids using the Volume of Fluid method [18]. In addition, `overInterDyMFoam` includes support for dynamic mesh motion with overset meshes [19].

Within each time step, the solver combines the numerical methods described in Sections 2.1 - 2.5. The VOF formulation is used to advect the phase fraction and update the fluid properties, and the pressure and velocity fields are advanced using the PIMPLE algorithm. The overset mesh framework is used to exchange information between overlapping mesh regions as the overset mesh moves through the domain.

2.7 Froude Similarity

Free surface ship flows are commonly characterized by the Froude number,

$$Fr = \frac{U_s}{\sqrt{gL}}, \quad (2.15)$$

where U_s is the ship speed, g the gravitational acceleration and L the length of the ship [20]. The Froude number characterizes the wave making behavior and free surface deformation of a hull: geometrically similar hulls with the same Froude number generate geometrically similar waves [21]. This principle is the standard basis for scaling free surface flows in ship hydrodynamics [22]. Matching the Froude number between a full scale ship and a scaled model requires the model speed U to satisfy

$$U = \frac{U_f}{\sqrt{\lambda}}, \quad (2.16)$$

where $\lambda = L_f/L_m$ is the geometric scale factor and subscripts m and f denote model and full scale, respectively.

2.8 Hydrodynamic Interaction Effects

The basic mechanism behind bank effects is that a vessel forces water to move around and underneath the hull. In open water, this return flow can spread relatively freely. In restricted water, however, the flow is squeezed between the hull, the bottom, and the lateral boundaries. This increases the local return flow velocity and modifies the pressure distribution on the hull [23]. If the restriction is asymmetric, for example when the ship is closer to one bank than the other, the pressure on the port and starboard sides becomes unbalanced. This pressure imbalance gives rise to additional hydrodynamic forces and moments, such as a sway force and yaw moment [24]. A simplified interpretation of this mechanism can be made using Bernoulli's equation [23],

$$p + \frac{1}{2}\rho u^2 + \rho gz = \text{constant}, \quad (2.17)$$

where u is the local flow velocity and z the vertical position. The higher velocity in the restricted gap between the hull and the bank can therefore contribute to decreased pressure on that side of the hull. For a ship sailing close to a bank, the resulting lateral force is often directed towards the closest bank, while the yaw moment tends to turn the bow away from it. This phenomenon is often referred to as *bank suction* [25, 23]. Bank-induced effects may also influence resistance, sinkage, trim and heel, although the present work mainly focuses on sway motion, yaw motion and steering response [23, 24]. A bank's *influence width*, y_{infl} , represents the approximate distance from the ship within which significant bank effects may occur. This influence width is expressed as

$$y_{\text{infl}} = B(5Fr_h + 5), \quad (2.18)$$

where B is the ship beam and $Fr_h = U_s/\sqrt{gh}$ is the depth dependent Froude number, with h the water depth. Analogous to the influence width, a corresponding limit exists for the water depth, beyond which the vertical restriction no longer affects the ship. A water depth of $h = 3T$, where T is the ship draught, equivalent to an under keel clearance of 200 %, is taken as the separation between deep and restricted water [25].

The same redistribution of velocity and pressure can occur between two ships. During a head-on meeting, the interaction loads change as the relative longitudinal position of the ships changes. The positive pressure regions near the bows initially produce a repulsive force that pushes the bows apart. As the ships continue to pass, attractive forces occur when the bow or stern of one vessel comes close to the low pressure region along the side of the other vessel, causing the induced yaw moment to change direction during the encounter [26, 27].

The strength of the hydrodynamic interaction effects depends on the parameters that control the velocity in the restricted flow region. Important parameters are the ship speed, under keel clearance, distance to the bank, bank geometry, water depth, blockage and ship geometry [25, 24].

3

Methods

This chapter describes the numerical methodology used to simulate the two-hull encounter. The chapter first introduces the staged workflow and its motivation, before presenting the case geometry, computational mesh, numerical methods, stage-specific setup, and steering strategies used in the simulations.

3.1 Workflow Overview

To construct the complete two-hull encounter simulation, the numerical workflow was divided into three main stages. First, an implementation of a single hull in a hull-fixed reference frame; second, an implementation of a single hull in a channel-fixed reference frame; and third, the construction of the final two-hull encounter case. Each stage served a specific purpose in the development of the final simulation.

The first stage consisted of simulating a single hull in a numerical towing tank configuration, see Figure 3.1. In this stage, the hull position was fixed and the relative motion between the hull and the water was generated by prescribing channel inflow at the inlet rather than by moving the hull through the domain. The purpose of this stage was to develop the flow field around the hull before introducing hull motion, which was done in the next stage.

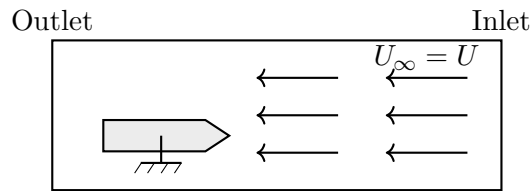


Figure 3.1: Top view illustration of the hull-fixed reference frame setup for developing the flow field (Stage 1). The inflow velocity U_∞ is equal to the ship speed U .

This avoided numerical issues that would arise from initializing the full encounter case directly from two hulls moving at full speed in initially undisturbed water. Such an initialization could generate strong non-physical transients and pressure spikes, since the fluid around the hulls would be forced to adjust rapidly from an initially quiescent state while the boundary layer, wake, and free surface disturbance developed. The alternative approach was to initialize the hull from rest and gradually accelerate to the target speed. However, this would have required a substantially longer simulation time and computational domain, since each vessel would need sufficient distance to accelerate and to fully

develop local flow fields before the encounter.

Developing the flow field around the hull in a towing tank configuration was therefore deemed the most practical and computationally efficient approach. An additional benefit of this approach was that disturbances arising from the unphysical initialization could be convected out of the domain and replaced by the prescribed inflow, rather than remaining in the domain and influencing the results. The main limitation of this approach was that it could only be applied to a single hull, since developing the flow for two hulls with opposite headings was not possible in a towing tank configuration. This stage therefore provided a precursor solution, which formed the basis for the subsequent moving-hull simulations.

In the second stage, the developed hull-fixed solution was shifted into a channel-fixed reference frame by giving the hull additional degrees of freedom and uniformly shifting the velocity field in the domain such that the free stream velocity was brought to rest, see Figure 3.2. The hull could then move through the domain at the target speed, while retaining the developed local flow field from the previous stage, which provided a more suitable initial condition than an undisturbed domain. The shifted case formed the basis for the third and final stage. This stage also served as an intermediate step, used to verify the rigid body motion setup after the reference frame transformation, and to evaluate and calibrate the steering strategies before they were applied to the final encounter simulation.

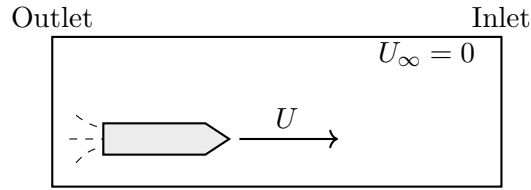


Figure 3.2: Top view illustration of the channel-fixed reference frame (Stage 2).

In the third stage, the complete two-hull encounter case was assembled from two versions of the Stage 2 case. Two copies of the case were generated, where one copy was rotated and translated such that the two domains were aligned at their respective inlets. The corresponding fields were then combined, as illustrated in Figure 3.3. In this way, the final encounter case could be initialized with developed flow around both vessels, rather than starting from two hulls in initially still water.

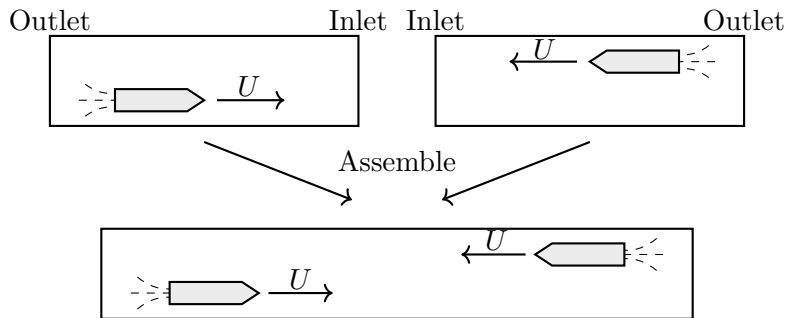


Figure 3.3: Illustration of assembling the final two-hull encounter case from Stage 2 cases.

A summary of the overall workflow is shown in Figure 3.4. The individual stages are described in further detail in their respective sections below.

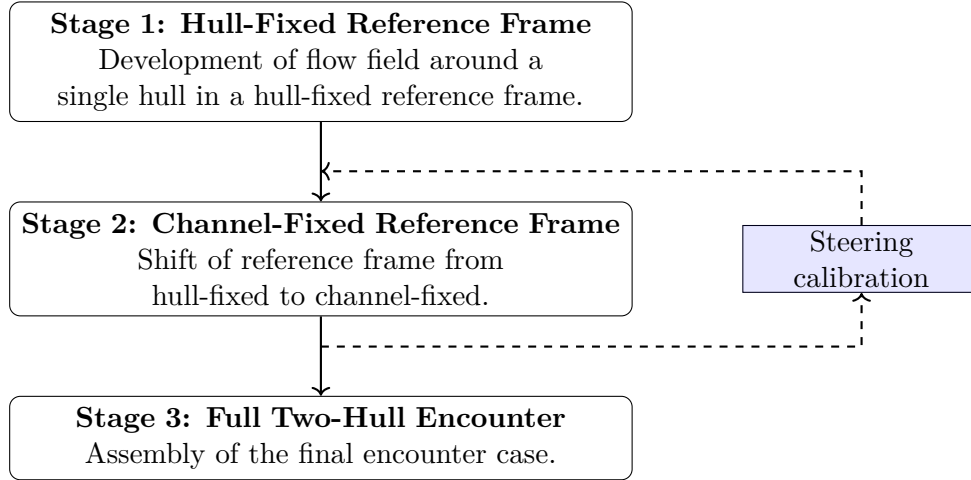


Figure 3.4: Illustration of the workflow.

3.2 Case Description

3.2.1 Hull Geometry and Model Scale

The hull geometry used in this study was based on a modified model of the Lucy Ashton, a paddle steamer that operated on the River Clyde in Scotland from 1888 to 1949. The vessel was later acquired by the British Shipbuilding Research Association and modified for full scale resistance measurements using jet propulsion, thereby avoiding propeller-hull interaction effects [28, 29]. The resulting measurements represent one of the few available datasets for full scale hull resistance. The Lucy Ashton was also used as the test case in a CFD resistance prediction workshop held at Chalmers University of Technology in 2024, where results from 40 participants were collected [30].

The Lucy Ashton hull was chosen for this project due to its relatively simple and well documented geometry, which was advantageous for the mesh generation and numerical stability of the simulations. This allowed the focus of this project to remain on establishing a numerical procedure for simulating hull interaction and steering in confined waterways rather than on extensive hull geometry preparation.

Although the data from the Chalmers workshop provide reference values, accurate resistance prediction was not the objective of the present project. Consequently, modeling choices such as near-wall resolution, turbulence models, and mesh refinement strategy were not optimized for resistance prediction.

The full scale main particulars of the Lucy Ashton are given in Table 3.1.

Designation	Symbol	Value
Length between perpendiculars	L_{pp}	58.1 m
Breadth, moulded	B	6.4 m
Draught	T	1.584 m
Wetted surface area	S	416.9 m ²

Table 3.1: Full scale main particulars of the Lucy Ashton [30].

The modified model of the Lucy Ashton used in this project excludes the rudder, and is also cropped in the vertical direction where the top of the rudder ended, modifying its depth and draught. This was done to simplify the meshing process, as the original deck and rudder increased the complexity of the geometry. The length between perpendiculars, L_{pp} , remained approximately the same, however. The simulations were carried out at model scale using a hull length of $L = 7.274$ m, corresponding to a geometric scale factor of

$$\lambda = \frac{58.1}{7.274} \approx 7.99. \quad (3.1)$$

The model speed was selected according to Froude similarity, as described in Section 2.7. A Froude number of $Fr = 0.2192$ was used, corresponding to one of the conditions considered in the CFD workshop. The resulting model scale speed was thus

$$U = Fr\sqrt{gL} = 0.2192\sqrt{9.81 \cdot 7.274} \approx 1.852 \text{ m/s}. \quad (3.2)$$

This corresponds to a full scale speed of approximately

$$U_f = U\sqrt{\lambda} \approx 5.23 \text{ m/s}, \quad (3.3)$$

or about 10.2 knots.

The selected speed is relatively high for confined waterway encounters such as the one studied in this project. However, it was chosen to remain consistent with the workshop Froude number and to increase the probability of obtaining measurable hydrodynamic interaction effects. The resulting forces and moments were therefore expected to be more clearly distinguishable from numerical noise than they would be at lower speeds [27, 23, 31].

The resulting model scale hull dimensions used in the simulations are summarized in Table 3.2.

Quantity	Symbol	Value
Length between perpendiculars	L_{pp}	7.274 m
Breadth	B	0.805 m
Draught	T	0.225 m
Height	H	0.29 m
Wetted surface area	S	6.63 m ²
Froude number	Fr	0.2192
Geometric scale factor	λ	7.99
Speed	U	1.852 m/s

Table 3.2: Model scale hull and operating parameters used in the simulations.

3.2.2 Channel Geometry

The computational domain was modeled as an idealized rectangular channel. This simplified geometry was chosen in order to reduce the number of geometric parameters in the numerical setup and to keep the focus on the staged simulation procedure, overset mesh formulation, hull-to-hull interaction, and steering response.

The channel dimensions were selected to provide a practical confined waterway configuration for the model scale hull. The domain had to be sufficiently long to allow the hull wake to develop, and to provide enough space for the two hulls to pass each other with sufficient distance to assess the steering during the full encounter. At the same time, the channel width had to allow the two hulls to pass each other without geometric overlap of their respective overset meshes, while still keeping the vessels sufficiently close to each other and the channel boundaries for lateral confinement to influence the flow.

The water depth was set to 1.87 m, giving an under keel clearance (UKC) of 1.64 m, and the channel width was set to 6 m. This provided the ships with a margin to sway and yaw, while still providing a narrow enough waterway for interaction effects to be significant. As stated in Section 2.8, for hydrodynamic bank interaction effects to be noticeable, the distance between a ship and the channel bank needs to be within the influence width of the bank effect, y_{infl} , where

$$y_{infl} = B(5Fr_h + 5) \approx 5.8 \text{ m.} \quad (3.4)$$

In a narrow channel, the ship would therefore always feel noticeable effects from the banks, unless the trajectory was centered in the channel. Any deviation from the channel centerline would thus create an asymmetry in the lateral hydrodynamic forces on the hull. The resulting depth corresponds to deep water conditions ($h \approx 8.3T$, exceeding the deep water criterion introduced in Section 2.8), and the confinement in this project is therefore purely lateral. Lateral distances between the hulls and the banks are further explained in Section 3.3.2.

Channel dimensions are summarized in Table 3.3.

Quantity	Symbol	Value
Channel length	L_{ch}	67.56 m
Channel width	B_{ch}	6.00 m
Water depth	h	1.87 m

Table 3.3: Dimensions for the full channel.

3.3 Computational Mesh

3.3.1 Background Mesh

As described in Section 3.1, the final two-hull domain was not generated directly as one complete initial case. Instead, the final two-hull encounter case was assembled from two transformed single-hull cases. The background mesh described in this section therefore corresponds to the channel mesh used for the Stage 1 and Stage 2 simulations, which was

half of the length of the final full channel.

The background mesh was generated using the OpenFOAM utility `blockMesh`, resulting in a fully structured hexahedral mesh. The mesh resolution was selected through a combination of preliminary estimates and iterative refinement. The initial streamwise cell size was chosen with the free surface wavelength in mind, which was initially estimated as [20]

$$\lambda_w = \frac{2\pi U^2}{g} \approx 2.2 \text{ m.} \quad (3.5)$$

The International Towing Tank Conference (ITTC) recommends no less than 40 cells across the shortest wavelength [32]. However, given the required computational cost for that level of refinement, and that accuracy was not the focus of the project, a compromise was made. After iterative Stage 1 simulations and mesh refinements, a final streamwise cell size of $\Delta x = 60 \text{ mm}$ was chosen, corresponding to between 30 and 35 cells per wavelength. The lateral cell size was chosen to be the same, keeping an aspect ratio of 1 in the x - y plane.

The vertical cell size in the free surface region was chosen in the same iterative fashion. An initial refinement was used to estimate the approximate range of free surface elevation around the hull. The refinement was then adjusted to cover the main part of this extent, resulting in approximately 17 cells across the wave height. The ITTC recommends a minimum of 20 cells, though the selected refinement was retained as a practical compromise between free surface resolution and total cell count [33].

In order to achieve this level of refinement over the free surface, while keeping the computational cost down, the mesh was divided into five vertically stacked blocks. The purpose of this arrangement was to obtain the desired resolution around the free surface while allowing the mesh to become progressively coarser farther away. The layered structure is illustrated schematically in Figure 3.5. Multiple blocks also gave greater control over the vertical cell size in the region which would be overlapped by the overset mesh (blocks 2, 3, and 4), which was important for the interpolation between the background and hull meshes described in Section 2.5. The central block spanned most of the free surface region and was assigned a uniform vertical cell size. The blocks directly above and below this region were vertically graded, providing a controlled transition to the coarser upper air and lower water regions.

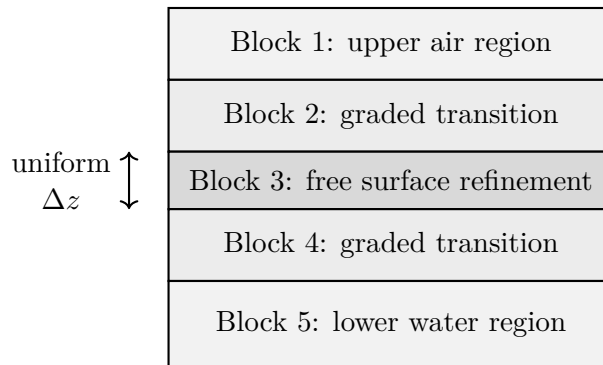


Figure 3.5: Illustration of the vertical mesh structure. The central block provides uniform refinement of the free surface, while the adjacent blocks provide graded transitions to coarser outer regions.

A coarser mesh was generated in the upper air region, since the air flow field was not expected to have significant influence on the hydrodynamic forces on the hull. Similarly, the lower part of the water column was coarsened because it was located far below the hull and the strongest velocity gradients were expected to occur closer to the hull, the free surface, and channel boundaries close to the free surface.

No dedicated boundary layer refinement was applied to the channel walls or bottom, since the local mesh refinement was deemed sufficient for the application of wall functions. This was considered acceptable because the channel wall and bottom shear stresses were not quantities of interest.

The resulting background mesh is illustrated in Figure 3.6, along with summarized mesh specifications in Table 3.4.

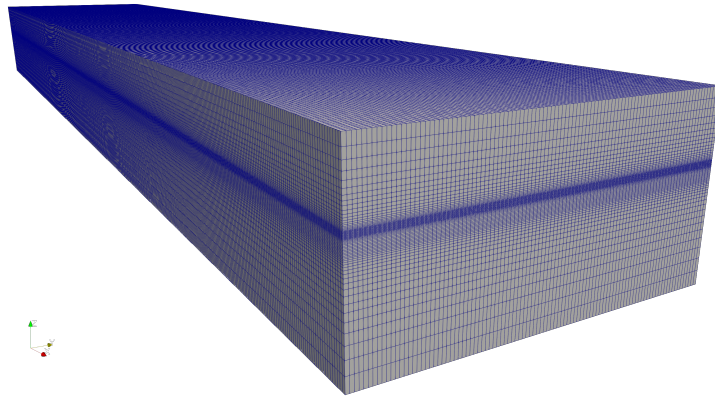


Figure 3.6: The structured background mesh.

Block	z -span [m]	$\Delta z_{\max}$ [mm]	N_z
1	0.564, 1.345	160.0	8
2	0.280, 0.564	52.2	13
3	0.200, 0.280	5.3	15
4	-0.007, 0.200	16.0	16
5	-1.643, -0.007	228.9	19
Total number of cells			3 997 300

Table 3.4: Background mesh specifications. The vertical cell expansion ratio was kept at or below 1.2 throughout.

3.3.1.1 Inlet Extension

During the hull-fixed flow development stage, long wavelength free surface disturbances were observed to form in the outlet region and propagate upstream through the channel. When these disturbances reached the inlet, where a prescribed inflow condition and fixed phase fraction distribution were imposed, they were reflected back into the domain. The reflected waves had shorter wavelengths and larger amplitudes than the original outlet disturbances, and would eventually contaminate the results and free surface throughout

the channel. This was particularly problematic for the later assembly of the full encounter channel, since a clean assembly required the inlet region to remain undisturbed to ensure that the two halves would match consistently at the fused interface.

The source of the disturbances was investigated through a series of diagnostic simulations. These included variations of the inlet and outlet boundary conditions, modifications of the channel dimensions, changes in mesh refinement, and a simulation of only the channel solved with `interFoam`, excluding the hull completely and thus also the overset interpolation. The disturbances were still observed in the hull-free case, although they appeared later and with smaller amplitude. This indicated that the issue was not caused primarily by the hull geometry or by the overset interpolation. The origin of the disturbances could not be uniquely isolated, but the tests suggested that they were associated with the channel boundary treatment.

The issue was also found to be mesh dependent. It was only observed when the background mesh was sufficiently refined, while the initial coarser meshes did not have these wave reflections. This suggested that the coarser mesh introduced sufficient numerical diffusion to attenuate the disturbances before they could propagate throughout the domain. Based on this observation, a streamwise coarsened inlet extension was added upstream of the original inlet plane. The extension is illustrated schematically in Figure 3.7.

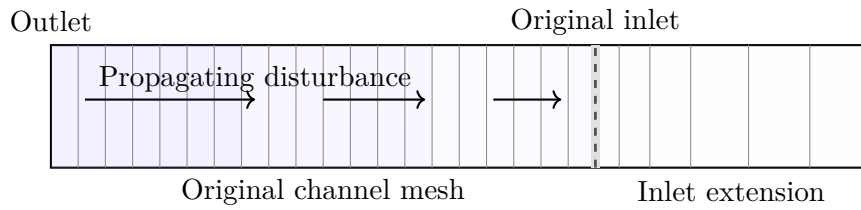


Figure 3.7: Schematic illustration of the inlet extension. Propagating disturbances weaken in the coarse extension before reaching the extended inlet.

The extension was appended upstream of the original inlet plane of the background mesh described in Section 3.3.1. Within the extension, the streamwise cell size was gradually increased away from the original channel, from regular channel resolution to approximately 3.3 m at the upstream end. The progressively coarsening cells numerically diffused the disturbances as they propagated upstream, attenuating them before they reached the inlet. This did not eliminate all disturbances from the channel, but prevented them from reaching the inlet and being reflected, leaving the original inlet region relatively undisturbed.

The extension was approximately 16.9 m long and contained 23 cells in the streamwise direction, with an overall first to last ratio of approximately 55. This added roughly 1.6×10^5 cells to the background mesh. The extension was used only during the hull-fixed stage and was removed by trimming the domain at the original inlet plane before the channel-fixed reference frame transformation described in Section 3.6. It is therefore not included in the background mesh cell count reported in Table 3.4.

The severity of the wave reflection issue was also found to be sensitive to the reference height h_{ref} used in the definition of the pseudo-hydrostatic pressure p_{rgh} . Placing h_{ref} in

the air region resulted in a less uniform p_{rgh} distribution and stronger free surface disturbances, while placing it at the initialized water level reduced the severity of the reflections significantly. The definition and role of h_{ref} are discussed further in Section 3.4.2.

3.3.2 Overset Mesh

The overset mesh was created using the OpenFOAM utility `blockMesh` to generate a hexahedral fully structured rectangular base mesh, followed by `snappyHexMesh` to snap the base cells to the CAD surface of the Lucy Ashton. The base mesh was positioned to overlap Blocks 2-5 of the background mesh and was therefore divided into four correspondingly refined vertical blocks, resulting in a reasonable match between the overset cell sizes and the background cell sizes in the overlap region.

The OpenFOAM utility `refineMesh` was then used to generate two nested internal refinement zones inside the base mesh, this is illustrated in Figure 3.8. These zones enclosed the region occupied by the hull geometry and provided a controlled refinement zone around the hull before applying `snappyHexMesh`. In each refinement zone, the local streamwise and lateral cell sizes were halved, resulting in a finest local resolution of

$$\Delta x = \Delta y = 15 \text{ mm.}$$

The purpose of these refinement zones was twofold. First and foremost, they reduced the local cell aspect ratio. Large aspect ratios in the cells surrounding the CAD surface were found to increase the risk of highly skewed and non-orthogonal cells when using `snappyHexMesh`. Second, they provided a finer mesh close to the hull surface, improving resolution and snapping of local cells to the CAD geometry. This removed the need for `snappyHexMesh` to create its own refinement levels. This was desirable because `snappyHexMesh` refinement levels split cells uniformly in all coordinate directions. Applying such refinement directly around the hull would therefore have refined not only the streamwise and lateral directions, but also the vertical direction, leading to a significantly larger number of cells. It would also have preserved the local cell aspect ratio, and introduced harder-to-control polyhedral transition cells around the refined surface. For this reason, the local cell sizes were primarily controlled before snapping.

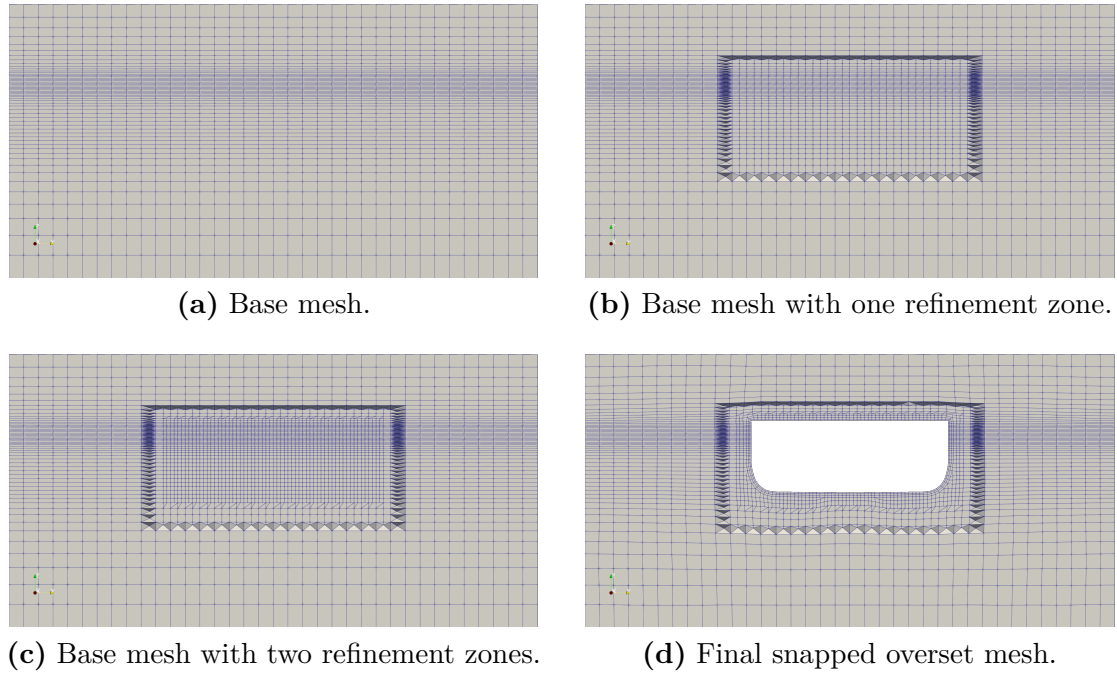


Figure 3.8: Streamwise cross sections showing the main stages of the overset mesh generation procedure.

A total of three prism layers were added on the hull surface. This was selected through iterative meshing tests, where the layer coverage, transition from prism layer cells to the surrounding mesh, and general mesh quality measures were assessed. Additional layers tended to distort the cells in the local refinement zones, especially below the keel. This can be observed in Figure 3.8(d), where some warping remains in the refinement zone below the hull. The warping became more pronounced when the refinement zone below the keel was made smaller or when additional prism layers were added.

The primary challenge with additional prism layers was that the layer thickness in the **snappyHexMesh** setup was controlled by the parameters **finalLayerThickness** and **expansionRatio**, which prescribe a single layer thickness for the entire hull patch [34]. The cells around the hull, however, did not have the same vertical refinement. The cells near the deck had the fine free surface refinement, while the cells near the keel were coarser. A single layer thickness could therefore not match the surrounding cells everywhere, and the transition between the prism layer cells and the surrounding cells became difficult to control over the entire hull surface.

Two possible approaches were considered to mitigate this issue. The first was to modify the CAD geometry cropping so that the deck region ended in a part of the overset base mesh with a vertical cell size more similar to that near the keel. This was not pursued because the hull geometry would become more complex and would have introduced additional snapping difficulties. The second option was to lower the free surface region relative to the hull, so that the keel and deck were located in more similarly refined vertical regions. This option was also abandoned because it would have reduced the effective draught of the hull and decreased hydrodynamic interaction effects.

The final prism layer configuration, three layers with an expansion ratio of 1.2, and final layer thickness of 1.5 times the minimum mesh edge length was therefore chosen as a compromise. This combination was found to produce an acceptable balance between the transition from prism layer cells to surrounding base cells, layer coverage over the hull surface, and mesh quality measures such as skewness and non-orthogonality. The resulting y^+ values lay within the intended range for the wall function treatment and were therefore considered acceptable for the purposes of the project.

The complete overset mesh overlaid on the background mesh is shown in Figure 3.9.

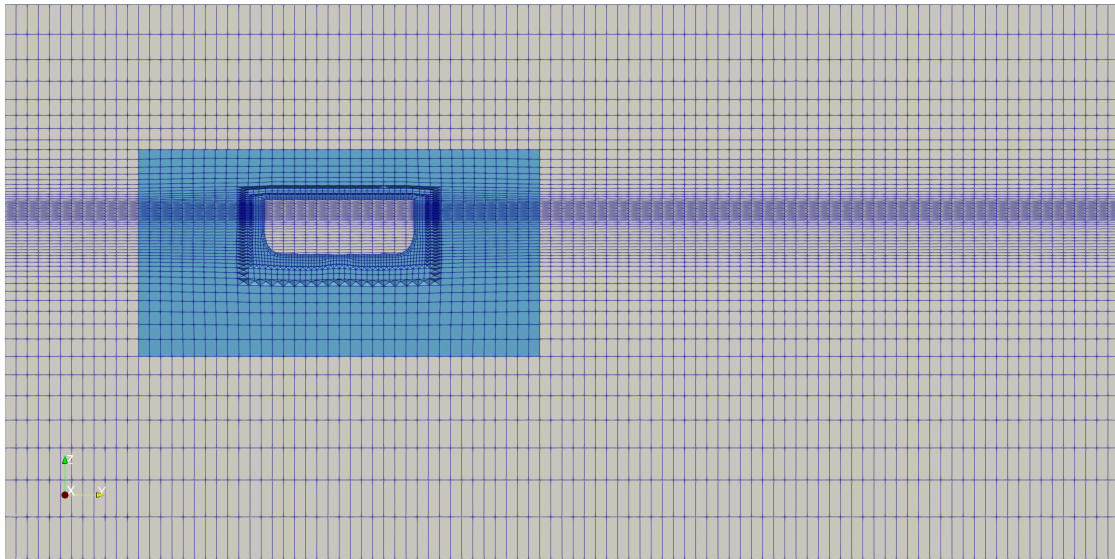


Figure 3.9: Streamwise cross section of the overset mesh (blue) overlaid on the background mesh.

The hull was positioned with a lateral offset from the channel centerline. This offset was required by the staged workflow explained in Section 3.1, creating the lateral clearance needed for the two hulls to pass each other. The distance between the hull starboard side and the closest channel wall was initialized as 1.40 m. The offset was also expected to provide enough clearance for hull motion without causing the overset mesh or the hull patch to intersect either the channel walls or the mirrored overset mesh in the final encounter case.

The number of cells in each coordinate direction from the hull patch was chosen to provide a sufficient overlap region for the interpolation between the background and the overset mesh. Since overset interpolation requires donor and acceptor cells of comparable size, the refinement zones were kept near the hull in order to reduce the number of potential donor/acceptor size mismatches. To further control the placement of the background interpolation fringe, the OpenFOAM parameters `holeLayers` and `useLayer` were set to 6 and 3, respectively.

The resulting overlap between the interpolation fringes contained approximately the fol-

lowing number of cells in each coordinate direction:

$$\begin{aligned} N_{x+} &= 8, & N_{x-} &= 8, \\ N_{y+} &= 7, & N_{y-} &= 7, \\ N_{z+} &= 6, & N_{z-} &= 8. \end{aligned}$$

Different combinations of `holeLayers` and `useLayer` alter the position of acceptor cells relative to the blanked region, which in turn affects the quality and robustness of the interpolation. The effect of the fringe-layer settings is illustrated in Figure 3.10. In order to ensure good interpolation between the background mesh and the overset mesh, it is important that interpolation fringes are placed in regions where acceptor and donor cells are of comparable sizes. The hole region was therefore extended to include most of the cells in the internal refinement zones generated with `refineMesh`, excluding the smallest cells from being donors.

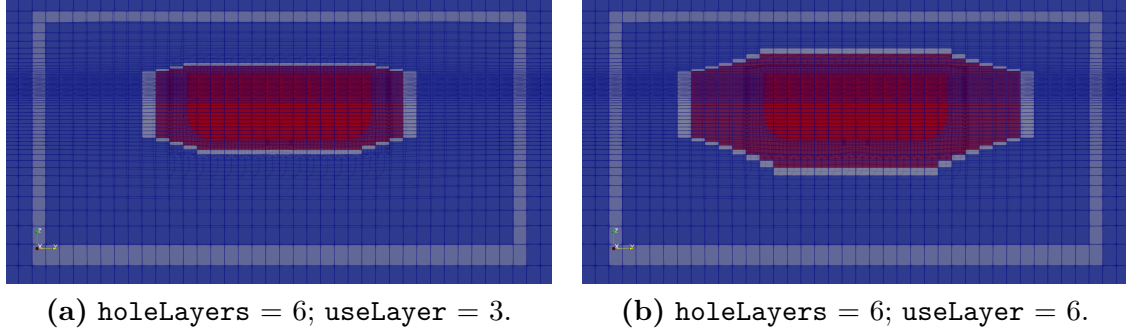


Figure 3.10: Effect of overset fringe settings on the cell type classification. Calculated cells are blue, interpolated cells are gray, hole cells are red.

The final surface mesh of the hull is shown in Figure 3.11, and final overset mesh details are summarized in Table 3.5.

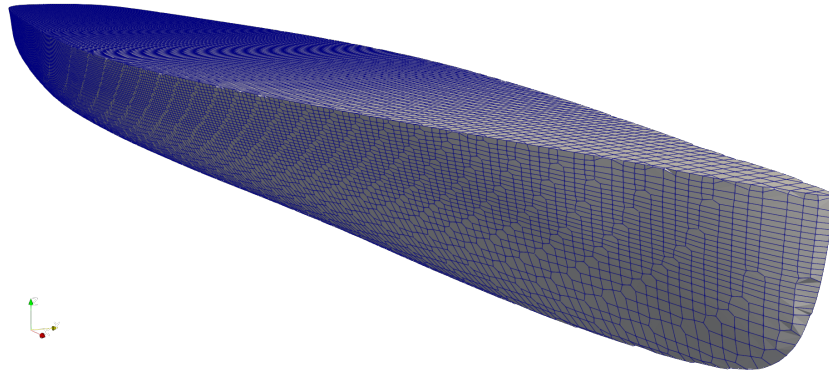


Figure 3.11: Isometric view of the meshed hull.

Quantity	Value
Overset mesh length	8.82 m
Overset mesh width	2.16 m
Overset mesh height	1.114 m
Hull–wall clearance	1.40 m
Hull–hull clearance	1.60 m
Under keel clearance	1.64 m
Overset–wall clearance	0.72 m
Overset–overset clearance	0.24 m
Overset–bottom clearance	1.09 m
Number of prism layers	3
Prism layer expansion ratio	1.2
First prism layer cell height	5.52 mm
Total number of cells	906 561
Hexahedral cells	832 776
Prism cells	3666
Polyhedral cells	70119
Maximum skewness	2.698
Maximum non-orthogonality	69.95°
Average non-orthogonality	9.82°

Table 3.5: Summary of the final overset mesh dimensions, refinement settings, clearances, cell composition, and quality metrics.

Once generated, the overset hull mesh and the background mesh were combined into a single computational mesh using the `mergeMeshes` utility, so that the moving hull mesh and the stationary channel mesh could be solved together. The overset method requires every cell to be identified as belonging to either the background mesh or the overset mesh, so that the solver can distinguish between the meshes for the cell type categorization. The cells of each mesh were grouped into separate cell sets, and each set was assigned a distinct `zoneID` value using the `setFields` utility.

3.4 Numerical methods

All simulations were carried out using the `overInterDyMFoam` solver in OpenFOAM v2306. The numerical settings described in this section were used for all stages of the simulation workflow.

3.4.1 Turbulence Modeling and Fluid Properties

Turbulence was modelled in a RANS framework using the realizable k – ε model, with wall functions providing the near-wall treatment on the hull and, in the channel-fixed stages, on the channel boundaries.

Both phases were treated as incompressible, with

$$\begin{aligned}\rho_w &= 998.8 \text{ kg/m}^3 & \nu_w &= 1.09e-06 \text{ m}^2/\text{s} \\ \rho_a &= 1 \text{ kg/m}^3 & \nu_a &= 1.48e-05 \text{ m}^2/\text{s},\end{aligned}$$

and a surface tension coefficient $\sigma = 0 \text{ N/m}$. Subscripts w and a denote water and air, respectively.

3.4.2 Pseudo-Hydrostatic Pressure

The pressure variable solved for in the simulations was the pseudo-hydrostatic pressure p_{rgh} , defined as the static pressure with the hydrostatic contribution removed,

$$p_{rgh} = p - \rho g(h - h_{\text{ref}}),$$

where h is the height in the direction opposite to gravity and h_{ref} is a reference height. This formulation is commonly used in OpenFOAM for buoyant and multiphase cases where hydrostatic pressure effects are important. By solving for p_{rgh} rather than the full static pressure p , the dominant hydrostatic contribution is separated from the pressure equation, which is numerically convenient for free surface simulations with gravity.

3.4.3 Numerical Schemes

The discretization schemes used are summarized in Table 3.6. The **Gauss** prefix in most of the schemes refers to the standard finite volume discretization using the Gauss divergence theorem. The time discretization used the first order **Euler** scheme and the spatial schemes are second order accurate. The default **Gauss linear** scheme was used for most gradients, while the cell limited variant **cellLimited Gauss linear 1** (the trailing 1 denoting full limiting) was applied only to the gradients in the turbulence convection terms in order to keep them bounded. The divergence terms of the velocity and turbulence quantities used the **linearUpwind** scheme.

In the volume fraction equation (Equation (2.8)), the convection term was discretized using the strictly bounded **vanLeer01** scheme, and the compression term was discretized using **interfaceCompression**.

The diffusion terms were discretized using **Gauss linear corrected**, where the **corrected** part applies a correction for mesh non-orthogonality. The surface normal gradients used the same **corrected** scheme. The explicit viscous stress term was discretized using **Gauss linear**. The overset interpolation between the moving hull mesh and the stationary background mesh used inverse distance (Section 2.5).

Entry	Scheme
Time derivative schemes	
default	Euler
Gradient schemes	
default	Gauss linear
limitedGrad	cellLimited Gauss linear 1
Divergence schemes	
div(rhoPhi,U)	Gauss linearUpwind grad(U)
div(phi,alpha)	Gauss vanLeer01
div(phirb,alpha)	Gauss interfaceCompression
div(phi,k)	Gauss linearUpwind limitedGrad
div(phi,epsilon)	Gauss linearUpwind limitedGrad
div(((rho*nuEff)*dev2(T(grad(U)))))	Gauss linear
Laplacian schemes	
default	Gauss linear corrected
Interpolation schemes	
default	linear
Surface-normal gradient schemes	
default	corrected

Table 3.6: Numerical schemes used in the simulations.

3.4.4 Solver Settings

The pressure-velocity coupling was handled using the PIMPLE algorithm. The values of `nOuterCorrectors` and `nCorrectors` specify the number of outer and inner correction loops described in Section 2.3. In this work, four outer correctors and four pressure correctors were used. One non-orthogonal corrector was included to account for mesh non-orthogonality. The main PIMPLE settings are summarized in Table 3.7.

Parameter	Value
<code>nOuterCorrectors</code>	4
<code>nCorrectors</code>	4
<code>nNonOrthogonalCorrectors</code>	1

Table 3.7: PIMPLE settings used for pressure-velocity coupling.

The pressure equation (`p_rgh`) was solved using the PBiCGStab solver with a DILU preconditioner. On the intermediate correctors a relative tolerance (`relTol`) of 0.05 was used, and on the final corrector `relTol` was set to zero so that the residual was driven down below the absolute tolerance. The turbulence quantities k and ε were solved to an absolute tolerance of 1×10^{-6} or a relative tolerance of 0.01. Solver tolerances are summarized in Table 3.8.

Field	Linear solver	tolerance	relTol
p_rgh	PBiCGStab (DILU)	1×10^{-9}	0.05
p_rghFinal	PBiCGStab (DILU)	1×10^{-9}	0
k, epsilon	smoothSolver (symGaussSeidel)	1×10^{-6}	0.01

Table 3.8: Linear solver tolerances.

The time step was adjusted dynamically to satisfy maximum Courant numbers of 3 for the flow and 2 for the interface region.

3.4.5 Under-Relaxation

Under-relaxation was applied to improve the numerical stability of the simulation and is presented in Table 3.9.

Field	Relaxation factor
U	0.5
k	0.5
epsilon	0.5

Table 3.9: Under-relaxation factors used.

3.4.6 Free Surface Treatment

The free surface was treated using the Volume of Fluid method described in Section 2.4. The transported phase fraction field was `alpha.water`, where $\alpha = 1$ represents water and $\alpha = 0$ represents air.

The phase fraction was limited using MULES. For the phase fraction transport, three subcycles were used together with three correction steps. The interface compression coefficient was set to $c_\alpha = 1$, which activated the artificial compression term used to reduce numerical smearing of the interface. The main free surface settings are summarized in Table 3.10.

OpenFOAM entry	Value
MULESCorr	yes
nAlphaCorr	3
nAlphaSubCycles	3
cAlpha	1

Table 3.10: Settings used for the phase fraction solution.

3.5 Hull-Fixed Simulation

The hull-fixed simulation corresponds to Stage 1 of the workflow described in Section 3.1. Two versions of this stage were prepared: one with the lateral offset of the hull explained in Section 3.3.2, and one with the hull positioned on the channel centerline for comparison.

3.5.1 Boundary Conditions

In the hull-fixed single-hull case, the hull was fixed in the streamwise direction, and the flow velocity was prescribed. The inlet represented the undisturbed upstream channel, the outlet allowed the flow to leave the domain, the atmosphere boundary represented the open air region, and the hull boundary represented the hull surface. The channel bottom and side boundaries were treated as slip walls in this stage. This prevented boundary layers from developing along the channel boundaries, in line with walls translating at the inflow speed in the hull-fixed reference frame.

The turbulence quantities for the internal field were specified using a turbulence intensity of $I = 1\%$ and an eddy-viscosity ratio of $\nu_t/\nu = 10$, giving

$$k = 5.15 \times 10^{-4} \text{ m}^2/\text{s}^2, \quad \varepsilon = 2.39 \times 10^{-3} \text{ m}^2/\text{s}^3, \quad \nu_t = 1 \times 10^{-5} \text{ m}^2/\text{s}.$$

At the inlet, `fixedValue` was used for the velocity field. This corresponds to a Dirichlet condition with the free-stream value

$$\mathbf{U} = (-1.852, 0, 0) \text{ m/s},$$

corresponding to the hull speed in the channel-fixed reference frame. The volume fraction also used `fixedValue` to maintain the initialized water level in the incoming flow. For pressure, `fixedFluxPressure` was used corresponding to a Neumann-type pressure condition, where the pressure gradient is adjusted to match the velocity flux.

At the outlet, `outletPhaseMeanVelocity` was applied for the velocity field. This condition adjusts the outlet velocity of the selected phase, here water, in order to obtain a prescribed mean phase velocity, with the mean phase speed set equal to the magnitude of the inlet velocity, preserving the mean flow rate through the domain. The pressure field used `zeroGradient`, corresponding to a homogeneous Neumann condition. The volume fraction was treated using `variableHeightFlowRate`, which constrains the phase fraction at the outlet and helps maintain bounded values of α . The turbulence quantities k and ε used `inletOutlet`, which applies a zero-gradient condition during outflow and prescribes inlet values if local backflow occurs. The turbulent viscosity ν_t was treated as a calculated field.

At the atmosphere boundary, `pressureInletOutletVelocity` was used for the velocity field. This acts as an open air boundary condition, applying a zero-gradient condition during outflow and allowing inflow based on the local flux, which is driven by the pressure gradient. The pressure was specified using `prghTotalPressure`, which imposes an atmospheric pressure condition for p_{rgh} . The volume fraction used `inletOutlet`.

On the hull, `movingWallVelocity` was used for the velocity field, which enforces the no-slip condition while accounting for patch motion. The pressure used `fixedFluxPressure` and the volume fraction used `zeroGradient`. Wall functions were applied for the turbulence variables on the hull surface.

The overset boundaries were assigned the `overset` type and were treated as interpolation interfaces rather than physical boundaries.

The boundary conditions applied to the flow variables are summarized in Table 3.11, while the corresponding turbulence boundary conditions are summarized in Table 3.12.

Boundary	U	p_{rgh}	α
Inlet	fixedValue	fixedFluxPressure	fixedValue
Outlet	outletPhaseMeanVelocity	zeroGradient	variableHeightFlowRate
Atmosphere	pressureInletOutletVelocity	prghTotalPressure	inletOutlet
Hull	movingWallVelocity	fixedFluxPressure	zeroGradient
Walls	slip	slip	slip
Overset	overset	overset	overset

Table 3.11: Boundary conditions for the flow fields in the hull-fixed case.

Boundary	k	ε	ν_t
Inlet	fixedValue	fixedValue	calculated
Outlet	inletOutlet	inletOutlet	calculated
Atmosphere	inletOutlet	inletOutlet	calculated
Hull	kqRWallFunction	epsilonWallFunction	nutkWallFunction
Walls	slip	slip	slip
Overset	overset	overset	overset

Table 3.12: Boundary conditions for the turbulence fields in the hull-fixed case.

3.5.2 Rigid-Body Motion Setup

The hull motion was computed using OpenFOAM’s `rigidBodyMotion` solver and only heave and pitch were included as degrees of freedom. The hull therefore remained fixed in its streamwise and lateral position, corresponding to an ideal straight trajectory, while still being allowed to respond dynamically to vertical forces and pitching moments generated by the flow.

The excluded degrees of freedom therefore represent an idealized perfectly steered condition. The forces and moments acting on the hull in this ideal trajectory serve as a baseline for the applied steering methods and thrust force introduced in the later channel-fixed stages.

3.6 Channel-Fixed Reference Frame Shift

The reference frame shift corresponds to Stage 2 of the simulation workflow which transforms the developed hull-fixed reference frame solution into an equivalent channel-fixed reference frame case. This allowed the hull to move through the computational domain while retaining the developed flow field obtained from the preceding hull-fixed simulation. The resulting case was used both as an intermediate verification case for the dynamic mesh setup and steering, and as the basis for the later full encounter case assembly.

3.6.1 Selection of Precursor Solution

The transformation was applied to a developed solution from the hull-fixed simulation. The selected solution was chosen after the initial transient associated with the start up of the flow had sufficiently diminished and the flow field had reached a statistically stationary state. This assessment was based on the temporal behavior of the hydrodynamic forces and moments acting on the hull, together with inspection of the wave structure around the hull. Since the hull motion exhibited periodic heave and pitch oscillations, the solution was selected at a time instant where the heave displacement and pitch angle were close to their respective mean values.

3.6.2 Reference-Frame Transformation

The reference frame transformation was performed as a Galilean shift of velocity achieved by adding a uniform velocity in all internal cells and boundary cells,

$$\mathbf{U}_{cf} = \mathbf{U}_{hf} + U\mathbf{e}_x, \quad (3.6)$$

where $\mathbf{U}_{cf}$ is the new velocity vector in the channel-fixed reference frame, $\mathbf{U}_{hf}$ is the old velocity vector from the hull-fixed reference frame, $U = 1.852$ m/s is the relative speed difference, and $\mathbf{e}_x$ is the positive streamwise unit vector.

The pressure field, phase fraction field, turbulence quantities, and mesh displacement were retained from the developed hull-fixed solution since they were frame invariant. This preserved the local wave pattern, wake, pressure distribution, and rigid body motion state of the hull. The transformed solution was then used as the starting condition for the channel-fixed single-hull simulation.

3.6.3 Boundary Condition Modification

The boundary conditions were modified to be consistent with a moving hull in a calm channel. The main difference from the hull-fixed setup was that the inlet and outlet were no longer used to drive a flow through the channel. The inlet and outlet were instead changed to open type conditions.

For the velocity field, the inlet and outlet conditions were changed to `inletOutlet`, allowing outflow while prescribing zero velocity in the case of local backflow. The pressure condition at the outlet was changed from `zeroGradient` to `fixedFluxPressure`, adjusting the gradient consistently with the local velocity flux. The phase fraction condition at the inlet was changed from a prescribed fixed value to `variableHeightFlowRate`, allowing waves to pass through as the hull approached.

The physical channel boundaries were also changed from slip boundaries to no-slip wall boundaries with wall functions for the turbulence quantities. The boundary-condition modifications are summarized in Table 3.13.

Boundary	Field	Hull-fixed case	Channel-fixed case
Inlet	$\mathbf{U}$	fixedValue	inletOutlet
	α	fixedValue	variableHeightFlowRate
	k	fixedValue	inletOutlet
	ε	fixedValue	inletOutlet
Outlet	$\mathbf{U}$	outletPhaseMeanVelocity	inletOutlet
	p_{rgh}	zeroGradient	fixedFluxPressure
Walls	$\mathbf{U}$	slip	noSlip
	p_{rgh}	slip	fixedFluxPressure
	α	slip	zeroGradient
	k	slip	kqRWallFunction
	ε	slip	epsilonWallFunction
	ν_t	slip	nutkWallFunction

Table 3.13: Boundary condition modifications introduced during reference frame shift.

3.6.4 Rigid-Body Motion Setup

The only modification of the rigid body motion setup in this stage was the number of degrees of freedom. To allow the hull to move through the channel and to respond to the hydrodynamic interaction effects, surge, sway, and yaw were added. The roll degree of freedom was not included as stated in the delimitations.

The heave and pitch state was retained from the selected hull-fixed precursor solution, including the corresponding displacement, velocity, and acceleration. The newly added degrees of freedom were initialized from rest, except for the surge velocity, which was set equal to the model speed to complete the reference frame shift,

$$\dot{q}_{\text{surge}}(0) = U = 1.852 \text{ m/s.} \quad (3.7)$$

The added surge displacement and acceleration were initialized to zero along with the added sway and yaw displacement, velocity, and acceleration. The degrees of freedom used in the two stages are summarized in Table 3.14.

Degree of freedom	Hull-Fixed	Channel-Fixed
Pitch	Active	Active
Heave	Active	Active
Surge	constrained	Active
Sway	constrained	Active
Yaw	constrained	Active
Roll	constrained	constrained

Table 3.14: Rigid body degrees of freedom during reference frame shift.

This initialization corresponds to releasing the hull from an ideal steering condition imposed by the previously restrained degrees of freedom, making this stage the first point in the workflow where the steering methods described in Section 3.8 could be applied and evaluated. This step thus served as an intermediate step in which the moving overset mesh, rigid body motion setup, and steering implementation could be tested before constructing the computationally more expensive two-hull simulation.

3.7 Two-hull encounter

This section corresponds to Stage 3 of the workflow. The purpose of this stage was to assemble the final two-hull simulation.

3.7.1 Assembly of the Two-Hull Domain

The final encounter case was assembled from two copies of the channel-fixed single-hull case, Copy A and B. Copy A was left unmodified, while Copy B was rotated by 180° about the vertical axis and translated so that the two copies would meet at their inlet planes. The rotation gave the two hulls opposite headings and placed them on opposite sides of the channel centerline, producing the initial configuration required for the meeting encounter.

The transformation of Copy B was applied both to the mesh and to the vector fields, so that the developed velocity field remained consistent with the reversed heading. Scalar fields, such as pressure, phase fraction, and turbulence quantities, are rotation-invariant and did not need separate treatment.

The two meshes were then combined using `mergeMeshes`. This created a single mesh containing both background domains coinciding at their inlets, and their respective overset hull meshes, but it did not connect the two domains. The coincident inlet planes were therefore joined using `stitchMesh`, which converted the matching inlet patches into internal faces. After this operation, the assembled mesh represented one continuous domain.

This assembly was only a geometric operation and the solution fields were not combined or retained after the assembly. The field files no longer matched the new mesh structure, and empty field files were therefore created to be populated by mapping the solutions from Copies A and B onto the assembled domain.

New cell sets were defined after the mapping for the background mesh and for the two overset regions.

3.7.2 Mapping of the Flow Field

The flow field in the assembled mesh was mapped using the `mapFields` utility. The solutions from Copy A and B were mapped onto their corresponding halves of the assembled domain, as illustrated in Figure 3.12.

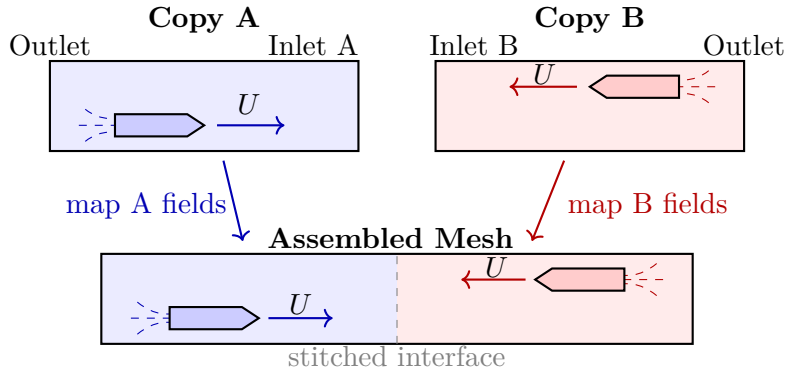


Figure 3.12: Illustration of the field mapping procedure.

The mapping successfully mapped the majority of the flow field, but failed to accurately map the regions directly surrounding the hulls.

3.7.3 Rigid Body Motion Setup

The final encounter case used different motion solvers for the two hulls. Hull A was treated as the vessel of interest and retained the five degrees-of-freedom setup with surge, sway, heave, pitch, and yaw active. This hull used the same `rigidBodyMotion` solver as previously. The steering methods were applied to this hull.

Hull B was assigned a prescribed straight trajectory and motion with constant velocity using OpenFOAM’s `solidBody` solver. This represents Hull B having ideal steering, which reduces the control problem to the trajectory response of Hull A, while still retaining the hydrodynamic interaction between the two moving vessels. This also entailed that Hull B was retained in its orientation throughout the simulation.

The initial surge velocity of Hull A was $U_A = 1.852$ m/s, while Hull B was prescribed the opposite velocity,

$$\mathbf{U}_B = -U\mathbf{e}_x. \quad (3.8)$$

The two hulls therefore approached each other with equal speed and opposite headings.

3.8 The Steering

The propulsion of the hull was implemented as an applied force at the stern, and steering was achieved by controlling the angle of the applied force. While ships are typically propelled by thrust generated by propellers, this simplification was made due to the limitations of this project, as outlined in Section 1.4. It should therefore be noted that the present propulsion model does not reproduce the flow field induced by a propeller. A propeller accelerates water in the aft direction, whereas an applied resultant force on the hull does not directly add momentum to the surrounding fluid. Likewise, steering is typically achieved using a rudder, which deflects the surrounding flow and thereby generates hydrodynamic forces and moments. The applied force model does not reproduce this flow deflection. However, controlling the angle of the applied force remains analogous to controlling the angle of a rudder.

Two methods of steering were evaluated, the Aim-point steering model, and a PID controller. In both versions, only the hull's sway and yaw were used as information for the steering methods.

3.8.1 Aim-Point Steering Model

The aim-point steering model corrects the trajectory based on the sway and yaw state, and the steering action is determined from the relation between the current heading and a point located ahead on the desired path [37, 35, 36].

The method is illustrated in Figure 3.13, and was implemented as follows. The lateral displacement of the hull center of gravity from its initial lateral position was denoted by l , and the yaw angle of the hull relative to its initial heading was denoted by ψ . An aim point was then placed on the desired trajectory, a streamwise distance L ahead of the hull center of gravity. The distance L , also called the *look-ahead* distance, determines how far downstream the controller aims. The steering error, φ , was defined as the signed angle between the hull's current heading and the line from the hull center of gravity to the aim point. It was calculated as

$$\varphi = \text{atan2}(-l, L) - \psi. \quad (3.9)$$

The commanded rudder angle was then calculated as

$$\delta = -k\varphi, \quad (3.10)$$

where k is a dimensionless gain. The gain controls the strength of the correction, while L controls the sensitivity to lateral displacement. For a given l , a larger L gives a smaller steering error and therefore a smoother steering response [38, 35].

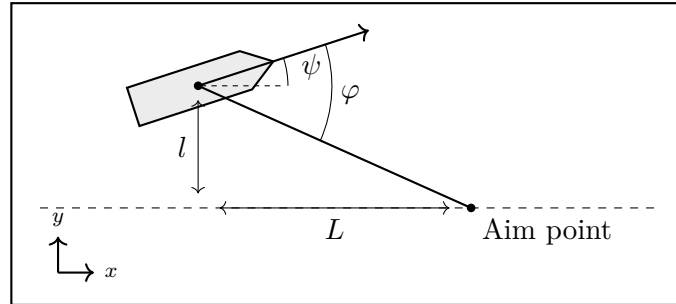


Figure 3.13: Illustration of the aim-point steering model. The hull center of gravity has swayed by l and yawed by ψ . The aim point is located on the desired trajectory a streamwise distance L ahead of the hull. The steering error is defined as φ .

The aim-point steering model was implemented in OpenFOAM as a custom restraint for the rigidBodyMotion solver, derived from the standard `externalForce` restraint included in the rigid body dynamics library. The modifications were limited to the parts of the source code directly affecting the application of the applied external force, while the surrounding rigid body framework was preserved.

The look-ahead distance was initially set to a distance of $L = 100$ m for the channel-fixed single-hull simulations in Stage 2. The evaluated gains were: $k = 1.5, 1.75, 2.0$,

2.25, and 2.5. After the findings from the results, the look-ahead distance was reduced to $L = 20$ for the subsequent full encounter simulation in Stage 3, keeping the same gains. Finally, a last combination using a look-ahead distance of $L = 8$, together with $k = 5$ was tested. The gains tested are summarized in Table 3.15.

Combination #	AIM1	AIM2	AIM3	AIM4	AIM5	AIM6
k	1.5	1.75	2.0	2.25	2.5	5

Table 3.15: Aim-point gain combinations used in the steering simulations. Combinations 1 to 5 were used with $L = 100$ and $L = 20$. Combination 6 used $L = 8$ m.

To prevent extreme rudder angles, the commanded angle δ was clamped to a maximum absolute value of 35° , and its rate of change was limited to $10^\circ/\text{s}$. The rudder angle limit follows standard practice, in which the maximum rudder angle of a comparable vessel is commonly limited to 35 degrees to each side [39].

The rudder angle was updated only once per timestep, at the beginning of the timestep, using the converged hull state from the end of the preceding timestep. This ensured that the rudder angle was not re-evaluated multiple times within a single timestep by the inner iterations of the rigid-body solver.

3.8.2 PID Controller

A PID controller was also implemented and used for steering. It was implemented in the same way as the aim-point steering model, as a custom restraint derived from the standard `externalForce` restraint, with the same rudder limits and update frequency applied. The difference is how the commanded rudder angle is calculated.

The PID controller treats the sway l and the yaw ψ as two independent error signals, each driven toward zero by a separate proportional and derivative action. An additional integral term acts on the sway alone, to remove any steady state lateral offset. The commanded rudder angle was calculated as

$$\delta = K_{P,l}l + K_{D,l}\dot{l} + K_{P,\psi}\psi + K_{D,\psi}\dot{\psi} + K_I \int_0^t l d\tau, \quad (3.11)$$

where $K_{P,l}$ and $K_{D,l}$ are the proportional and derivative gains acting on the sway, $K_{P,\psi}$ and $K_{D,\psi}$ are the corresponding gains acting on the yaw, and K_I is the integral gain acting on the accumulated sway.

The combinations of gains tested in the channel-fixed single-hull cases are listed in Table 3.16.

Combination #	$K_{P,1}$	$K_{D,1}$	$K_{P,\psi}$	$K_{D,\psi}$
PID1	3.0	1.0	2.0	1.0
PID2	0.5	0.3	1.5	0.7
PID3	0.2	0.3	1.0	0.7
PID4	0.8	0.7	2.0	1.0

Table 3.16: PID gain combinations used in the steering simulations. The integral gain K_I was set to zero in all cases.

In all cases the integral gain was set to zero ($K_I = 0$), effectively reducing the controller to a PD form.

For the final two-hull encounter, the time constraint of the project only allowed for PID combination 1 to be evaluated.

3.9 Computational Resources

All simulations were executed on Tetralith, a computing cluster operated by the National Supercomputer Center (NSC) at Linköping University and made available through the National Academic Infrastructure for Supercomputing in Sweden (NAISS). Each compute node comprises two Intel Xeon Gold 6130 processors (16 cores each), giving 32 cores per node.

The three simulation stages described in Section 3.1 were run in parallel on the cluster. The number of CPU cores, simulated physical time, and wall clock time for the main simulation stages are summarized in Table 3.17. All decompositions followed a $n/2 \times 2 \times 1$ hierarchical split in the streamwise, lateral, and vertical directions, respectively, where n denotes the total number of CPU cores. This split reflects the elongated streamwise extent of the domain and limits the number of processor boundaries intersecting the overset mesh region.

Stage	CPU cores	Simulated time [s]	Wall clock time [h]
Stage 1	160	45	44
Stage 2	160	12.2	14–34
Stage 3	320	12.2–27	32–55

Table 3.17: Computational resources used for the main simulation stages. Wall clock times are approximate; ranges reflect variation across the individual simulations within each stage.

Within Stage 2 and Stage 3, aim-point and unsteered cases were consistently faster than PID controlled cases, and were similar in duration to one another. Within Stage 3, wall clock times also varied depending on whether a case was terminated early following a collision or continued for a longer portion of the encounter. An aim-point case that did not result in a collision ran 27 seconds of simulated time, with a corresponding wall clock time of approximately 41 hours.

4

Results and Discussion

This chapter presents the results obtained from the staged workflow described in Chapter 3. The results are presented in the same order as the workflow, starting with the hull-fixed simulations, followed by the channel-fixed single-hull cases and finally the two-hull encounter case. Throughout the chapter, the x , y , and z directions refer to the streamwise, lateral, and vertical directions in the global, channel-fixed coordinate system.

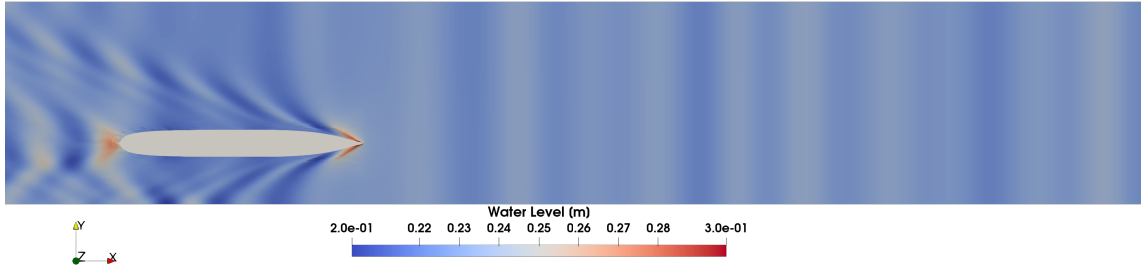
4.1 Hull-fixed Simulations

This section presents the results from the hull-fixed simulations in Stage 1. First, the free surface and the effect of the coarse inlet extension are presented, followed by a comparison between an offset and centered hull case.

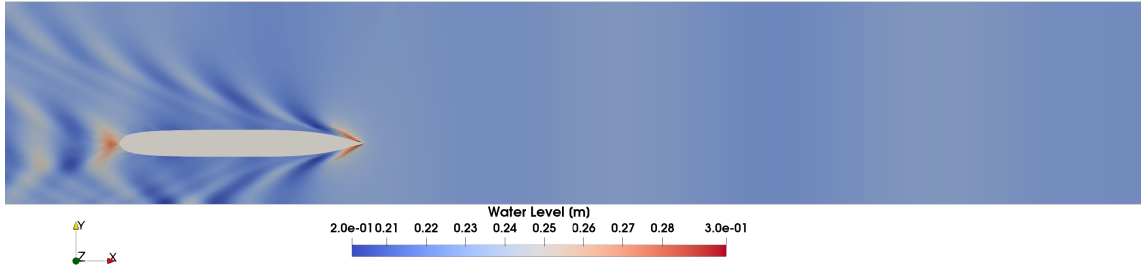
4.1.1 Inlet extension and wave reflections

Figure 4.1 shows the developed flow field for the offset and centered hull, and also illustrates the mitigating effect of the inlet extension on the wave reflections discussed in Section 3.3.1.1. Figure 4.1(a) shows the case without an inlet extension, where wave reflections can be observed on the free surface. The remaining subfigures, (b) and (c), show cases where the inlet extension was used. In these cases, the disturbances are significantly smaller, since the propagating waves dissipate in the coarse extension before reaching and reflecting from the inlet. The underlying source of the waves is still present, and propagating waves can still be observed, although the magnitude of the disturbance is reduced.

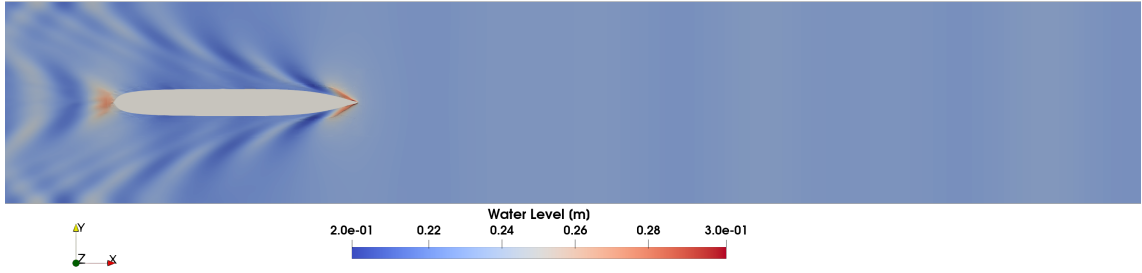
Figure 4.2 shows the Kelvin wake structure around the hulls, which has an approximate Kelvin wake angle Φ_k and transverse wavelength λ_k of around 22° and 2.18 m, comparable to the theoretical constant Kelvin wake angle 19° and wavelength $\lambda_w = 2\pi U^2/g$ [20]. The bank's effect on the wake structure is also observable. In the offset case, the waves are reflected on the starboard side bank with the reflection angle equal to the angle of incidence, as expected from theory.



(a) Offset, no inlet extension.



(b) Offset, inlet extension, (cropped image).



(c) Centered, inlet extension, (cropped image).

Figure 4.1: Top views of the fixed-hull simulation domain after 30 seconds, showing the hull and water level. The undisturbed water level is at 0.225 m.

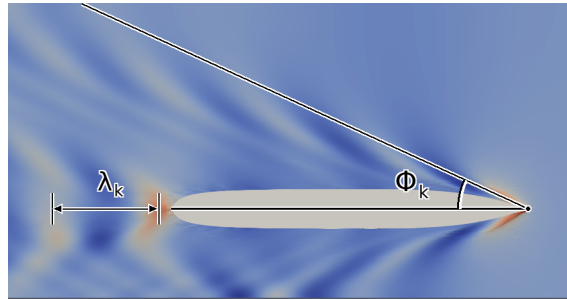


Figure 4.2: Kelvin wake angle $\Phi_k \approx 22^\circ$. Transverse wavelength $\lambda_k \approx 2.18$ m.

4.1.2 Offset vs. Centered Hull

4.1.2.1 Pressure distribution

The pressure distribution over the offset and centered hulls is shown in Figure 4.3. The pressure distribution is noticeably asymmetric along the length of the offset hull, compared to the centered hull. At the stern and mid-body, the pressure between the offset hull and the starboard bank is lower than on the unrestricted port side of the hull, as expected from the theory in Section 2.8. The same regions on the centered hull show no significant asymmetry. For both cases, the pressure distribution in the bow region of the hull appears symmetric about the hull centerline.

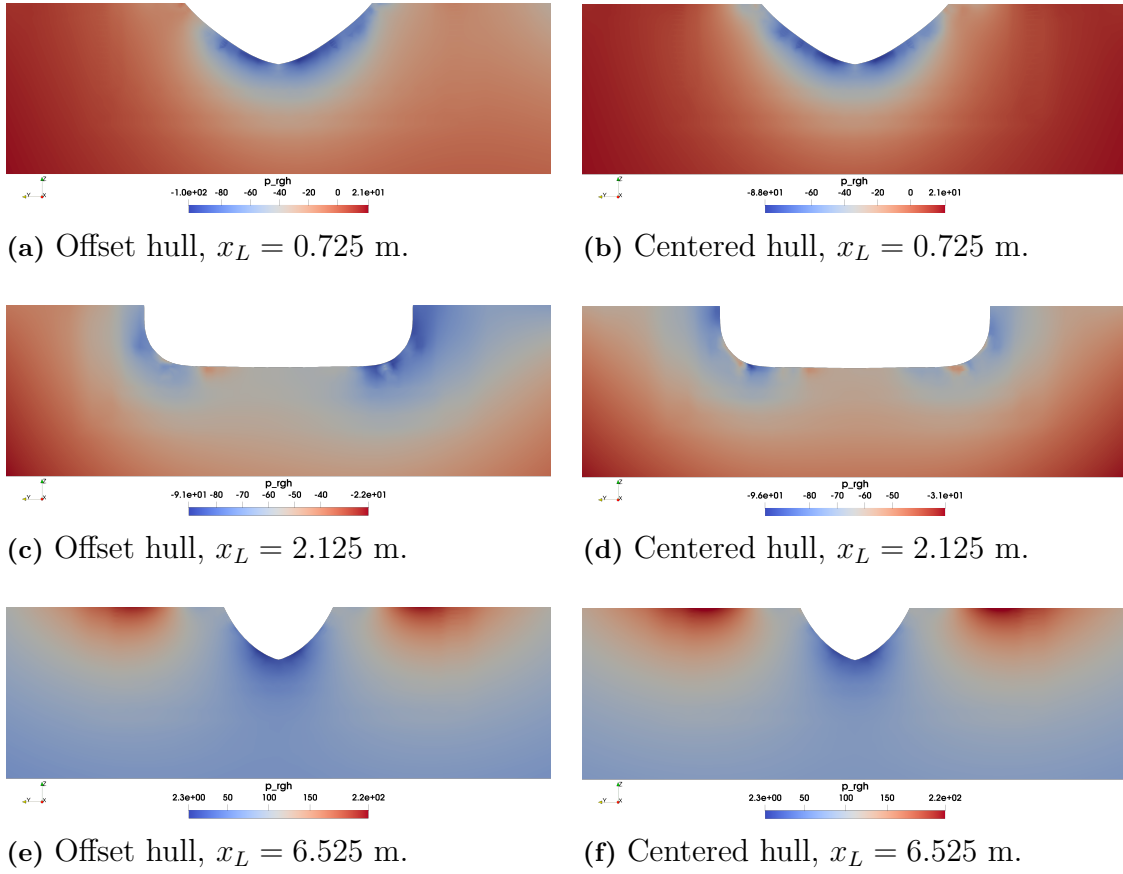


Figure 4.3: Streamwise-normal slices showing the p_{rgh} distribution for the offset and centered hull cases. The slices have vertical extent $-0.3 < z < 0.22$ m and lateral extent ± 0.8 m from the hull centerline. The stern is located at $x_L = 0$, and the starboard side is shown to the right.

4.1.2.2 Hydrodynamic Forces and Moments

The total forces and moments acting on the hull are presented in Figure 4.4. Since the resolved force and moment signals contain high frequency oscillations, a moving time average was included to make the overall trends easier to compare. The faint lines show the raw resolved data, while the darker lines show the moving time average. The moving average was computed using a sliding 0.5 s time window, where each plotted value represents the average of the data in the surrounding 0.5 s interval. The plotted values start from

$t = 20$ s to exclude unrepresentative large transient values from the unphysical initialization. Figure 4.4 shows that the force and moment signals oscillate at a frequency similar to that of the hull's pitch, which is plotted in Figure 4.5, indicating that the oscillations are driven by the resolved heave and pitch motion of the hull.

The force and moment components are calculated using OpenFOAM's `forces` function object and are reported in the global, channel-fixed coordinate system, with moments taken about the hull's center of rotation. Since the only rotational motion for the hull-fixed cases is pitch, and the pitch angle remains small, within approximately $\pm 0.3^\circ$, as shown in Figure 4.5, the misalignment between the global and hull-fixed x axes is itself small, as the resulting difference in F_x scales with the sine of the pitch angle. The reported F_x is therefore considered sufficiently representative of the hull for assessing the development of the solution and for selecting the prescribed thrust force used in later stages. The lateral force F_y is unaffected by pitch and is therefore identical in the global and local frame for the hull-fixed case. The same small pitch argument applies to M_x and M_z , which are not identical to the hull-fixed roll and yaw moments when pitch is present, but where the difference scales with the pitch angle. In the later moving hull stages, where yaw motion is also allowed, M_z remains a suitable measure of the yaw loading, since a yaw rotation alone does not change the z component of a moment vector, and the remaining difference is again associated with the small pitch motion.

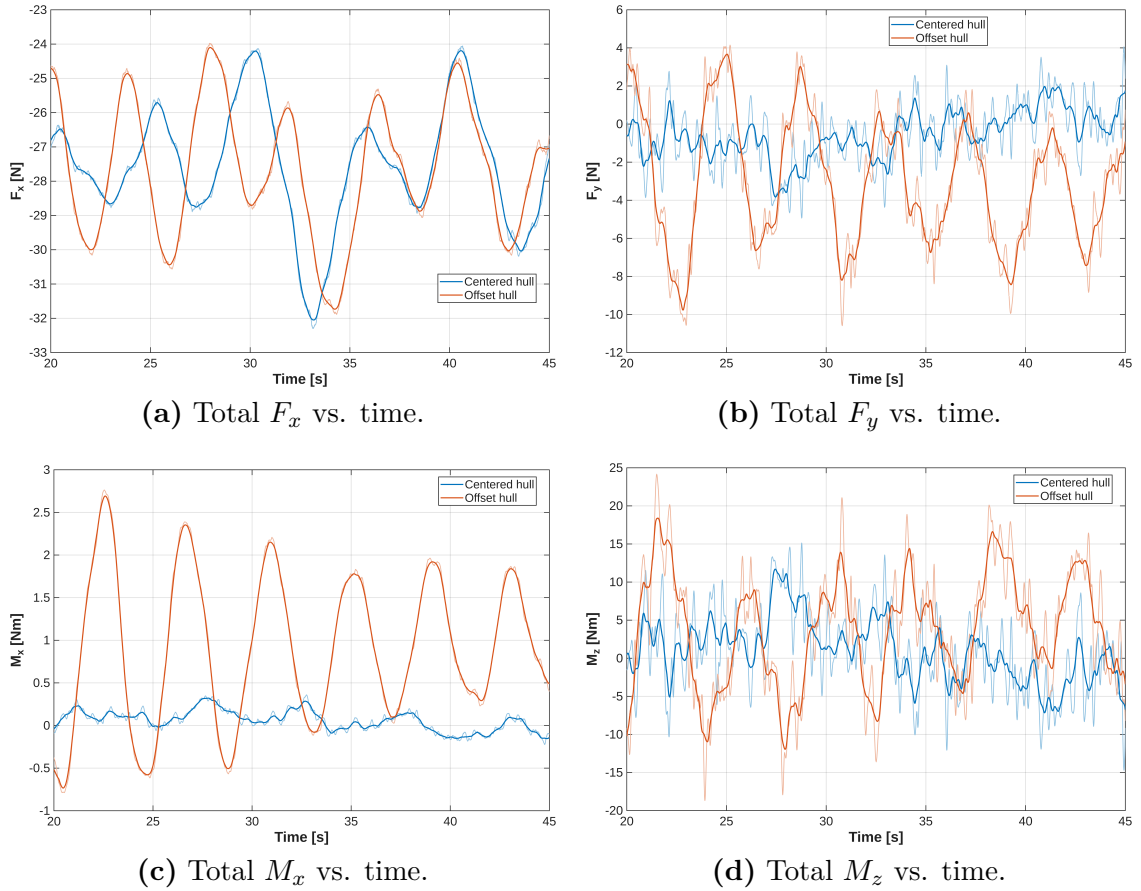


Figure 4.4: Forces and moments vs. time. Faint lines: raw resolved signals. Darker lines: moving time averaged signals. Red: offset hull. Blue: centered hull.

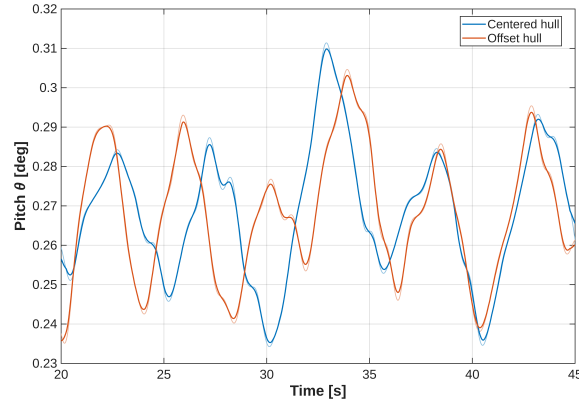


Figure 4.5: Pitch angle vs. time. Faint lines: raw resolved signals. Darker lines: moving time averaged signals using a 0.5 s sliding time window.

The force and moments in Figures 4.4(b), (c) and (d) show larger amplitudes for the offset hull. However, due to oscillations and overall noise in the figures, comparing the cases is difficult. The running means of the signals are therefore plotted in Figure 4.6. The running mean is different from the moving time average used in Figure 4.4. While the moving time average is a local smoothing operation, the running mean is a cumulative time average. At each time instant, the running mean is calculated by averaging all previous data points from the start of the averaging interval up to the present time. Consequently, the final value of each curve is of primary interest, since it represents the mean value over the full averaging interval, $t = 20\text{--}45$ s.

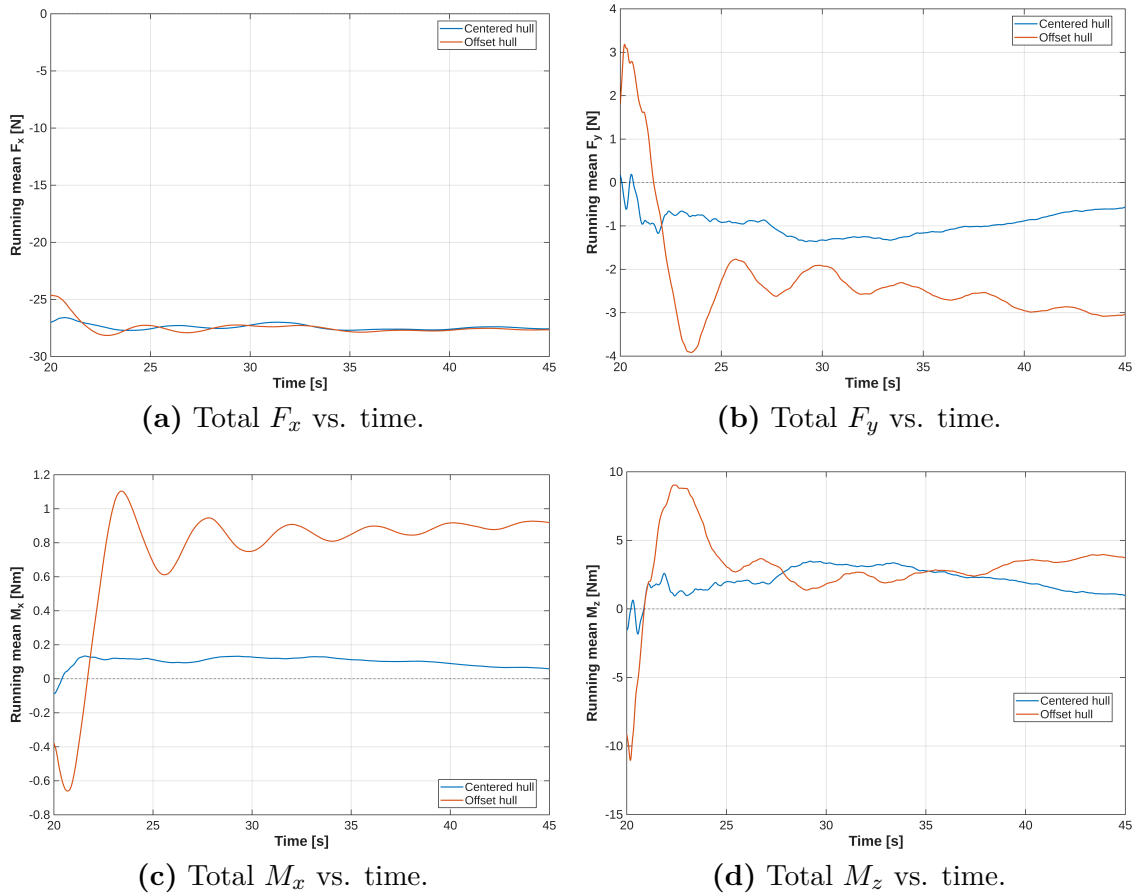


Figure 4.6: Running mean forces and moments vs. time. Red: offset hull. Blue: centered hull.

Although some of the running mean curves still vary slightly near the end of the interval, they reveal a clearer difference between the cases, particularly for the sway force and the moments which are all greater in magnitude for the offset hull. The larger, positive roll moment conforms with the low pressure regions on the starboard bottom side shown in Figure 4.3. Likewise, low pressure regions around the starboard midbody and stern are in line with both the greater negative sway force and positive yaw moment shown in Figure 4.6. The streamwise force curves remain close to each other, indicating that the difference in total streamwise force between the two cases is small.

For a numerical comparison, the final values of the running mean curves in Figure 4.6 are listed in Table 4.1. These values are taken at $t = 45$ s and correspond to the mean loads over the interval $t = 20$ – 45 s.

Quantity	Centered hull	Offset hull
F_x [N]	-27.57	-27.65
F_y [N]	-0.57	-3.04
M_x [N m]	0.06	0.92
M_z [N m]	0.96	3.72

Table 4.1: Final values of the running mean curves in Figure 4.6, evaluated at $t = 45$ s. Averaging interval: $t = 20$ – 45 s.

The tabulated values confirm that the streamwise force is negligibly affected by the lateral offset, changing from -27.57 N for the centered hull to -27.65 N for the offset hull. The main differences are instead found in the lateral force and moments. The lateral force changes from -0.57 N to -3.04 N, while the roll and yaw moments increase from 0.06 Nm to 0.92 Nm and from 0.96 Nm to 3.72 Nm, respectively.

4.2 Channel-Fixed Simulations (Single Hull)

This section presents the results obtained from the steering calibration considered in Stage 2. The cases were transformed to the channel-fixed reference frame using the flow field obtained at $t = 30$ s from the preceding parent simulation. The magnitude of the thrusting force was set to 28 N in all cases, to match the magnitude in Table 4.1. All simulations last 12.2 s, after which the hull reaches the inlet.

In the following figures, time is reported relative to the start of the Stage 2 simulations. The gain settings for the active steering cases are given in Section 3.8.

4.2.1 No Active Steering

The following result corresponds to no active steering. The rudder angle was maintained at 0° relative to the hull's heading throughout the simulation.

The resulting yaw, sway, rudder, and yaw-moment histories are presented in Figure 4.7, which shows that the hull starts to displace at the start of the simulation, as soon as the restrictions on the degrees of freedom are lifted. Furthermore, the yaw dynamics are illustrated clearly. The yaw angle increases during the initial part of the simulation, while the yaw moment is predominantly positive. Between approximately $t = 2$ s and $t = 10$ s, the yaw angle continues to increase with a relatively constant slope, while the yaw moment oscillates around zero. After $t = 10$ s, the yaw angle begins to decrease, coinciding with a predominantly negative yaw moment.

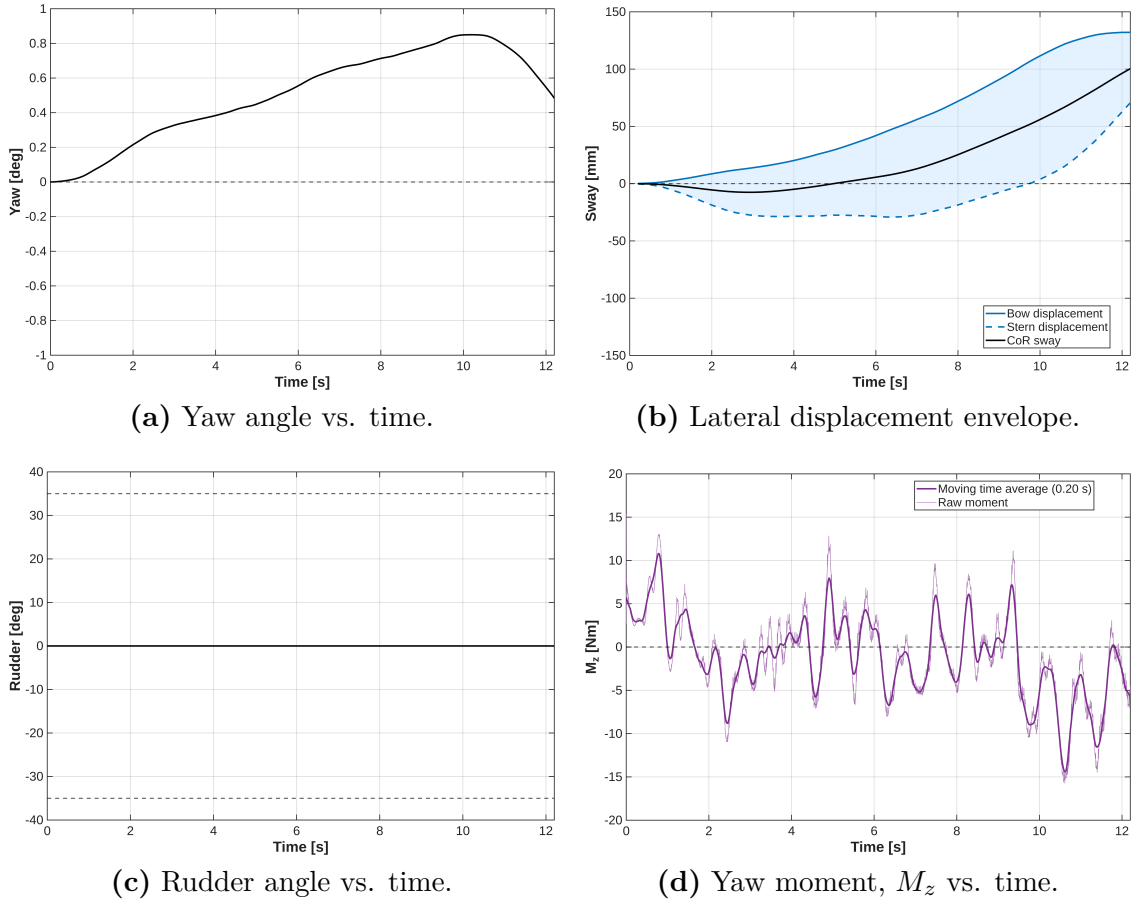


Figure 4.7: Summary of steering related quantities, (no active steering).

The direction of the initial yaw moment agrees with the mean bank-induced yaw moment found for the offset hull in the hull-fixed simulations (Table 4.1), confirming that the disturbance identified in Stage 1 drives the free response observed here. Since the rudder is fixed at 0° , no rudder moment acts on the hull, and the plotted moment is also the net moment.

Figure 4.7(b) shows the lateral displacement of the hull. The bow displacement increases as the yaw angle increases. In Figure 4.8, the position of the hull's center of rotation, along with its heading is plotted to show the trajectory.

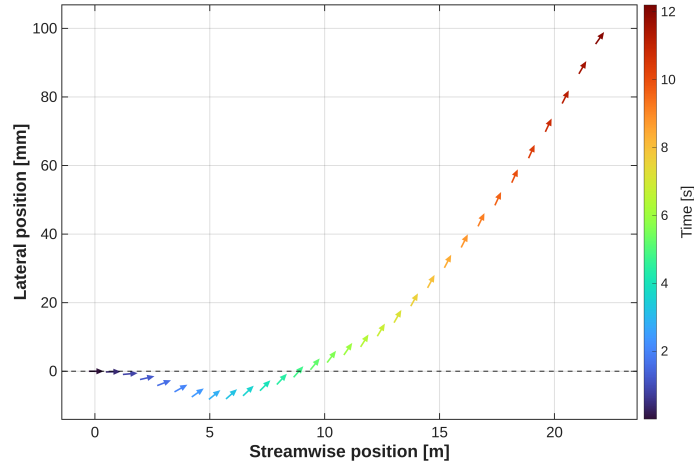


Figure 4.8: Hull trajectory. Initial position of the hull is defined as (0,0).

The center of rotation initially remains close to the centerline, swaying slightly to starboard while the bank-induced moment rotates the hull. As the yaw angle grows, the thrust, which acts along the hull heading, carries the hull steadily towards port, consistent with the expected bank effect behavior.

The results are summarized in Table 4.2.

Quantity	Max	Min	t_{max}	t_{min}
Yaw [deg]	0.85	0.00	10.183	0.00
CoR sway [mm]	100.385	-7.556	12.20	2.935
Bow sway [mm]	132.076	-6.3	12.20	0.00
Stern sway [mm]	70.596	-29.249	12.20	6.40

Table 4.2: Extreme values of the hull motions over $t = 0$ –12.20 s, together with the times at which they occur.

The maximum yaw angle is 0.85° , occurring at $t = 10.183$ s. The largest lateral displacements occur at the final time, $t = 12.20$ s, where the CoR, bow, and stern displacements are 100.385 mm, 132.076 mm, and 70.596 mm, respectively. This shows that the lateral motion has not reached a bounded response within the simulated interval, but continues to increase in the absence of steering. The larger bow displacement compared with the stern displacement is consistent with the positive yaw angle shown in Figure 4.7(a).

4.2.2 Aim-Point Steering Model

Figure 4.9 presents steering related quantities and shows that all aim-point calibrations outperform the unsteered case in terms of maximum lateral displacement, and that the differences between the calibrations themselves are small. The rudder angle counteracts the bank-induced drift throughout the run, and towards the end of the simulations the hull yaws back towards the target trajectory after having swayed to port, as reflected in the bow and stern displacements.

4. Results and Discussion

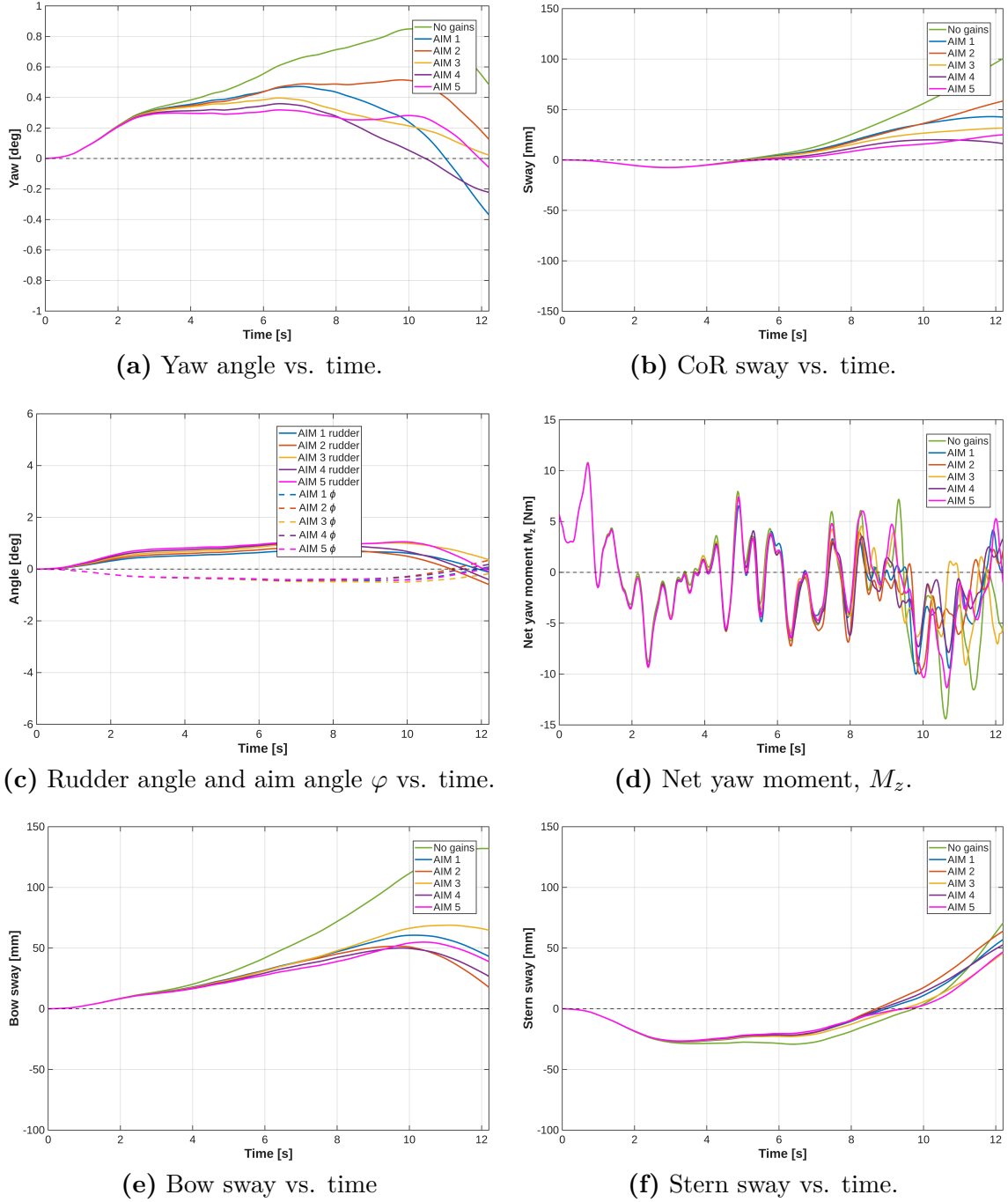


Figure 4.9: Summary of steering related quantities for aim-point cases 1 to 5 with $L = 100$.

Note that the yaw moment in Figure 4.9(d) is the net moment driving the yaw motion, which consists of the sum of the hydrodynamic moment integrated over the hull surface and the rudder moment, which is applied directly in the rigid-body equations of motion and is computed from the applied thrust and its lever arm about the center of rotation. The two contributions are shown separately in Figure 4.10. The rudder moment in subfigure (a) opposes the hydrodynamic moment in subfigure (b), counteracting the bank induced yaw.

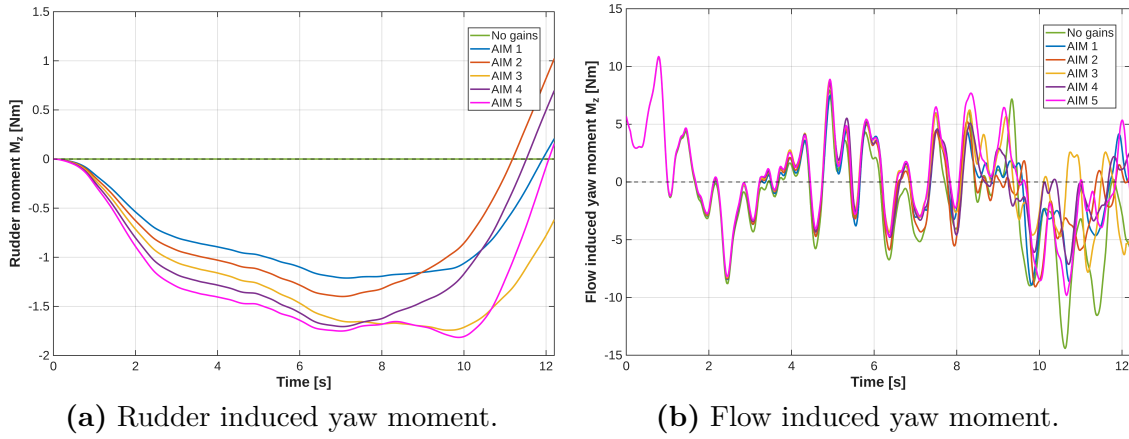


Figure 4.10: Rudder induced yaw moment, and flow induced (hull) yaw moment.

The extreme values for the aim-point cases are summarized in Table 4.3.

Quantity	AIM 1	AIM 2	AIM 3	AIM 4	AIM 5
Max. CoR sway [mm]	50.3	41.7	55.1	40.1	43.0
Max. bow sway [mm]	60.6	51.5	68.9	50.0	54.9
Max. stern sway [mm]	57.1	63.9	46.1	52.6	46.8
Max. rudder angle [deg]	0.70	0.81	1.01	0.99	1.05
Min. rudder angle [deg]	-0.12	-0.59	0.00	-0.40	-0.09
Max. yaw angle [deg]	0.46	0.46	0.49	0.44	0.41
Max. φ angle [deg]	0.47	0.46	0.51	0.44	0.42

Table 4.3: Summary of steering extrema for the aim-point cases over $t = 0-12.2$ s. Sway, yaw and φ maxima are taken of absolute values.

4.2.3 PID

The corresponding time histories for the active PID cases are shown in Figure 4.11. Similar to the Aim-point steering model, Figure 4.11 shows that all tested PID calibrations outperform the case without steering in terms of maximum lateral displacement.

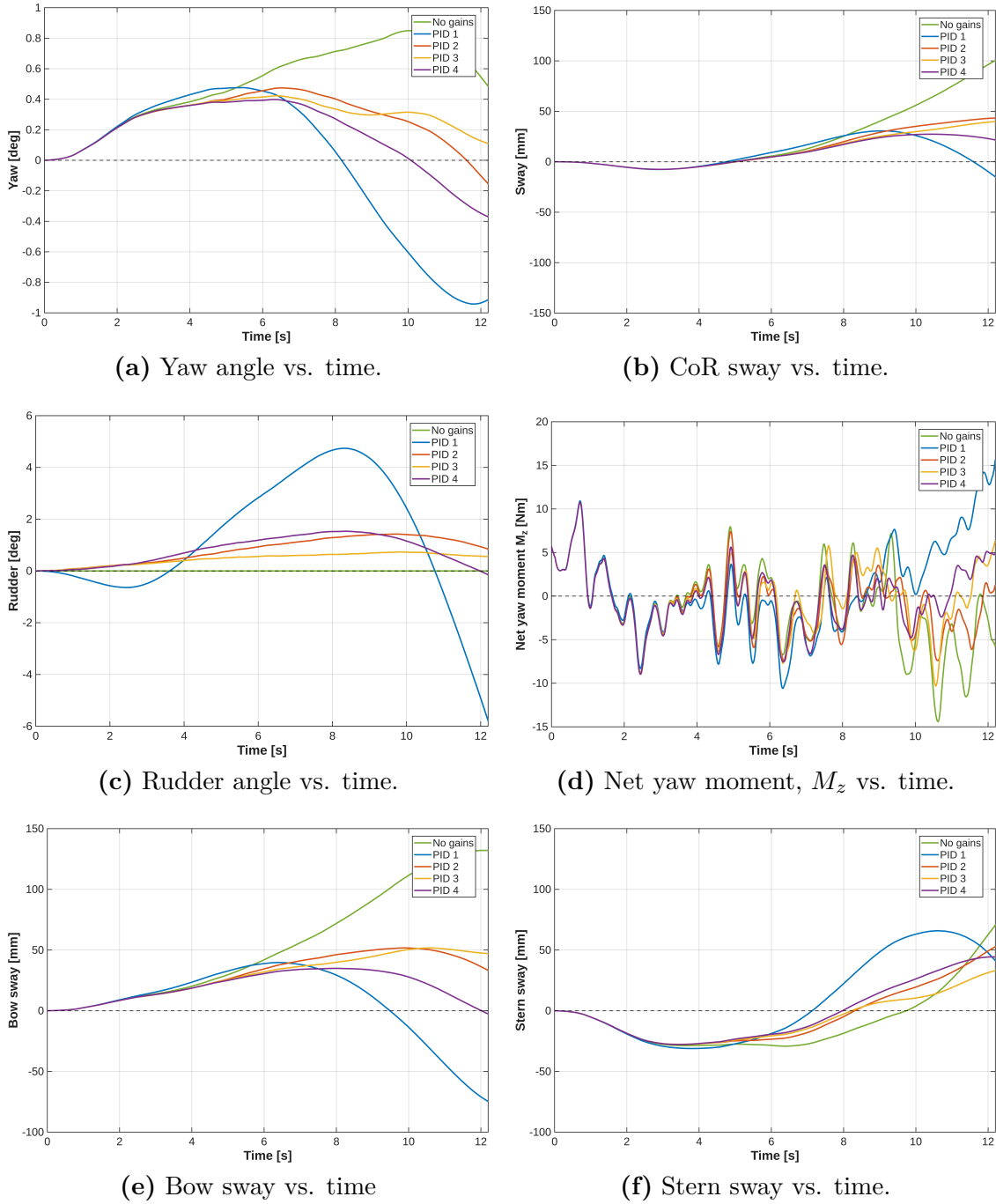


Figure 4.11: Summary of steering related quantities for the PID cases.

Compared to the aim-point cases, both the yaw response and the rudder activity vary noticeably between the PID calibrations, with PID 1 standing out. This is reflected in the net yaw moment, which is largest for PID 1 towards the end of the simulation as its rudder deflection rapidly decreases; the coupling between rudder angle, net moment and yaw is evident, with the large negative rudder deflection corresponding to the large positive net moment, which slows the decrease in yaw angle around $t \approx 12$ s. Although the maximum center-of-rotation sway is comparable to the aim-point cases, the rudder and yaw histories show oscillatory behavior and overcorrection, particularly for PID 1.

The PID 1 case also illustrates the yaw dynamics clearly. Its yaw angle decreases after the maximum at $t \approx 5$ s while the net moment has been predominantly negative during the climb, since the net moment governs the yaw acceleration. It first decelerates and then reverses the yaw rate, and the yaw angle continues to fall until the rate changes sign again at $t \approx 12$ s, which happens when the yaw moment becomes positive as a result of the negative rudder angle.

The extreme values for the PID cases are summarized in Table 4.4.

Quantity	PID 1	PID 2	PID 3	PID 4
Max. CoR sway [mm]	30.6	43.3	39.9	27.3
Max. bow sway [mm]	74.8	51.7	51.8	34.9
Max. stern sway [mm]	65.9	52.8	33.1	44.4
Max. rudder angle [deg]	4.74	1.43	0.73	1.53
Min. rudder angle [deg]	-5.79	0.00	0.00	-0.15
Max. yaw angle [deg]	0.94	0.47	0.42	0.40

Table 4.4: Summary of steering extrema for the PID calibration cases over $t = 0$ – 12.2 s. Sway and yaw maxima are taken of absolute values.

The smallest maximum CoR sway is obtained with PID 4 (27.3 mm) and the largest with PID 2 (43.3 mm). Notably, PID 1 achieves the second-smallest CoR sway (30.6 mm) while simultaneously showing the largest bow sway (74.8 mm), the largest rudder activity (-5.79° to $+4.74^\circ$) and a maximum yaw angle (0.94°) exceeding even the unsteered case. The ranking thus depends on the chosen metric: PID 4 gives the smallest CoR sway, bow sway and yaw angle, whereas PID 3 shows the smallest stern excursion and the least rudder activity.

Although the PID calibrations give better results on average than the aim-point cases for the given setup, the variation in their rudder response to the four gain parameters is harder to predict than for the aim-point model, where the relationship between L , k , the yaw and the sway is more straightforward. The PID cases also show signs of instability.

4.3 Channel-Fixed Simulations (Full Encounter)

This section presents the results from the full encounter simulations in Stage 3. The section first discusses the mapping of the two single-hull solutions and the resulting initial flow field. The hull motion is then evaluated for the case with no active steering, followed by the results obtained with the aim point and PID steering models.

4.3.1 Mapping

Images of the full encounter initialization are shown in Figure 4.12. The areas away from the hulls were mapped successfully and retained the solution from their respective parent cases. However, Figure 4.12(b) and (c) shows that the area around the immediate vicinity of the hulls did not get mapped successfully, and had incorrect values.

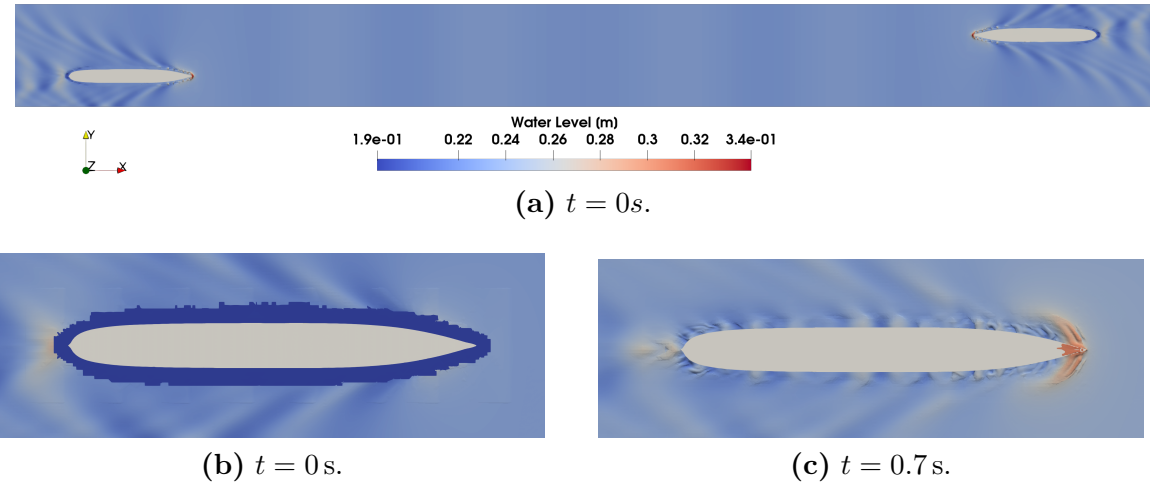


Figure 4.12: Top view of the final encounter case. Top: full domain. Bottom: zoom on hull A.

The flow field eventually recovers before the encounter. After around 6 seconds, most of the disturbances around the hulls have disappeared, as shown in Figure 4.13, where both hulls have fully prescribed motion. The motions for Hull A still used the `rigidBodyMotion` solver as described in Section 3.6.4; however, an under-relaxation factor of zero applied on the acceleration effectively locked the trajectory.

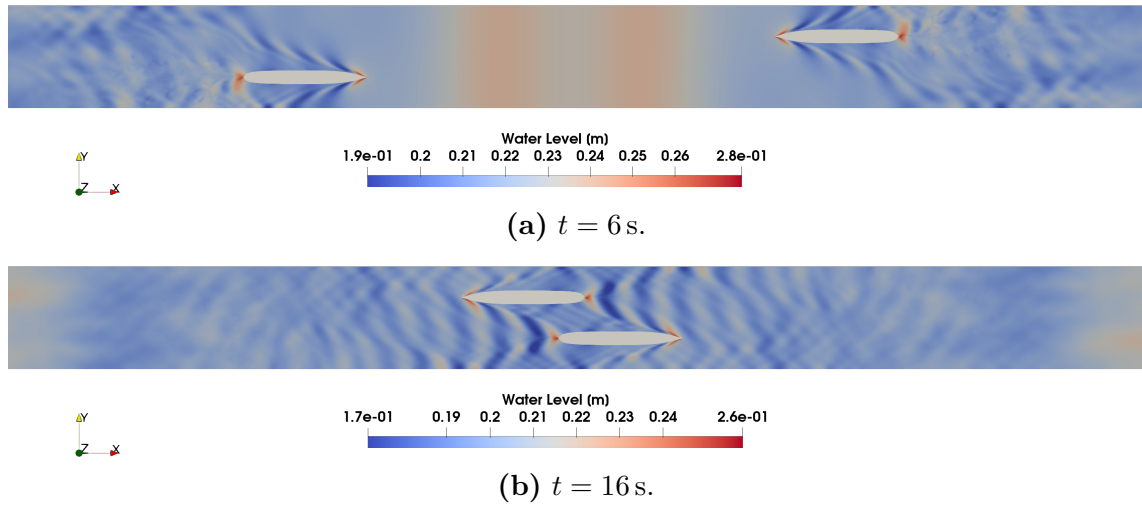


Figure 4.13: Top view of the final encounter case at two different time instances. Both hulls have fully prescribed motion to illustrate the recovering flow field.

Figure 4.13(a) shows that two noticeable waves appear in the middle of the channel. These are the result of the propagating disturbances from the hull-fixed parent simulation. The reference frame transformation eliminated the source of the disturbances and in the channel-fixed single-hull case, these propagating waves could freely exit the domain through the open boundary conditions. The same is true for the current case, although there are now waves propagating towards each other from either side of the channel. Once the waves reach their respective outlet, however, they fully exit.

Although locking both hulls shows that the flow recovers within roughly 6 s, the steering simulations in the following sections were initialized directly from the mapped field at $t = 0$ s rather than from the recovered field. They are therefore affected by the initial disturbance throughout.

4.3.2 No Active Steering

Despite the flow field recovering, the initial disturbance lingers long enough to generate large forces and moments that affect the hull's trajectory dramatically. This can be seen in Figure 4.14.

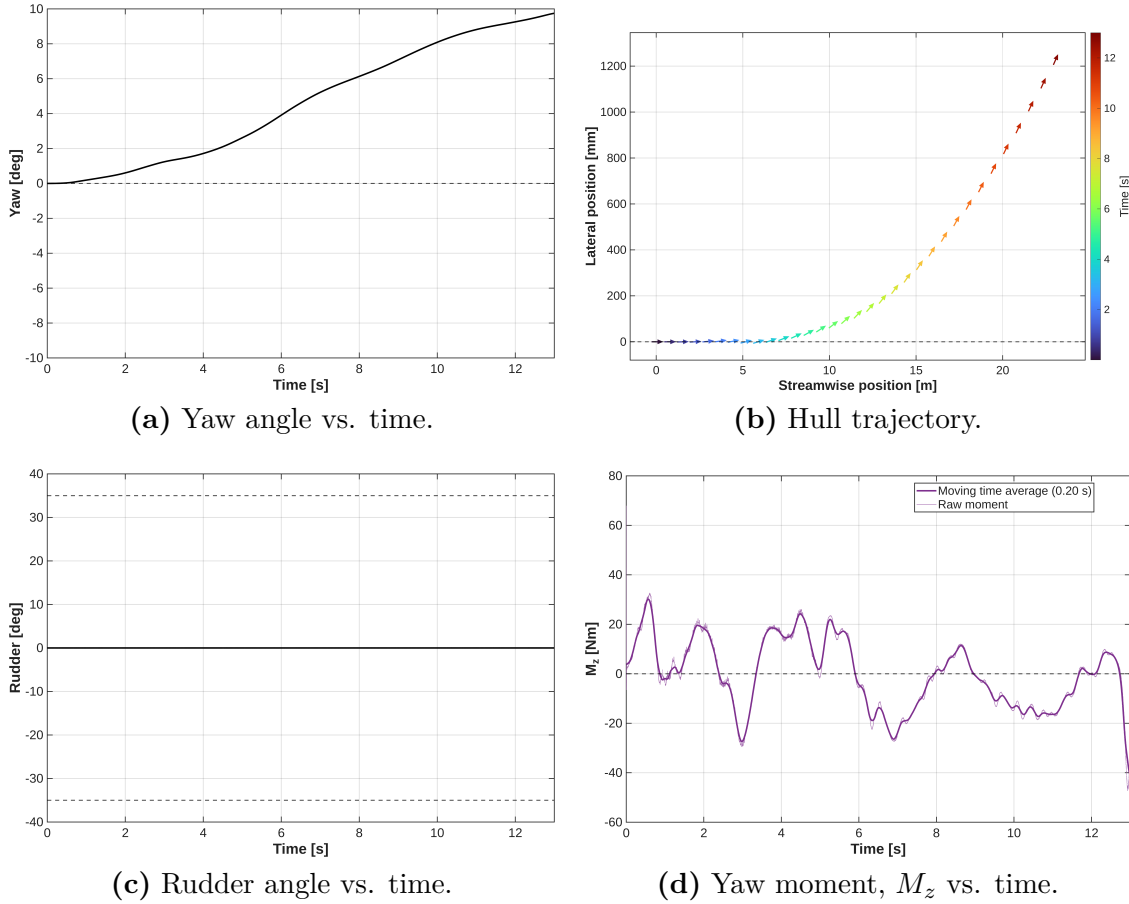


Figure 4.14: Summary of steering related quantities.

Both yaw and sway are significantly larger for this case compared to its single-hull counterpart. The yaw angle reached its maximum value of 9.75° , and the maximum displacement of the center of rotation was 1269.234 mm, after which it collided with the second hull. The moments leading up to the collision are also possibly indicated in the yaw moment, which is predominantly negative as the hulls' respective bows approach each other, although it could also be a normal oscillation.

4.3.3 Aim-point Steering Model

Following the single-hull results, the look-ahead distance was reduced from $L = 100$ to $L = 20$ for the encounter to obtain a stronger steering response. The result is shown

in Figure 4.15. Similarly to the single-hull case, all aim-point cases outperform the case with no active steering. The effect of the disturbance caused by the mapping can be seen in the presented yaw moment in the figures, which differ significantly in magnitude from the corresponding single-hull versions. Furthermore, the rudder activation is also greater, as expected from the decrease in look-ahead distance, although the large yaw moment is likely also a key contributor for the rudder activation.

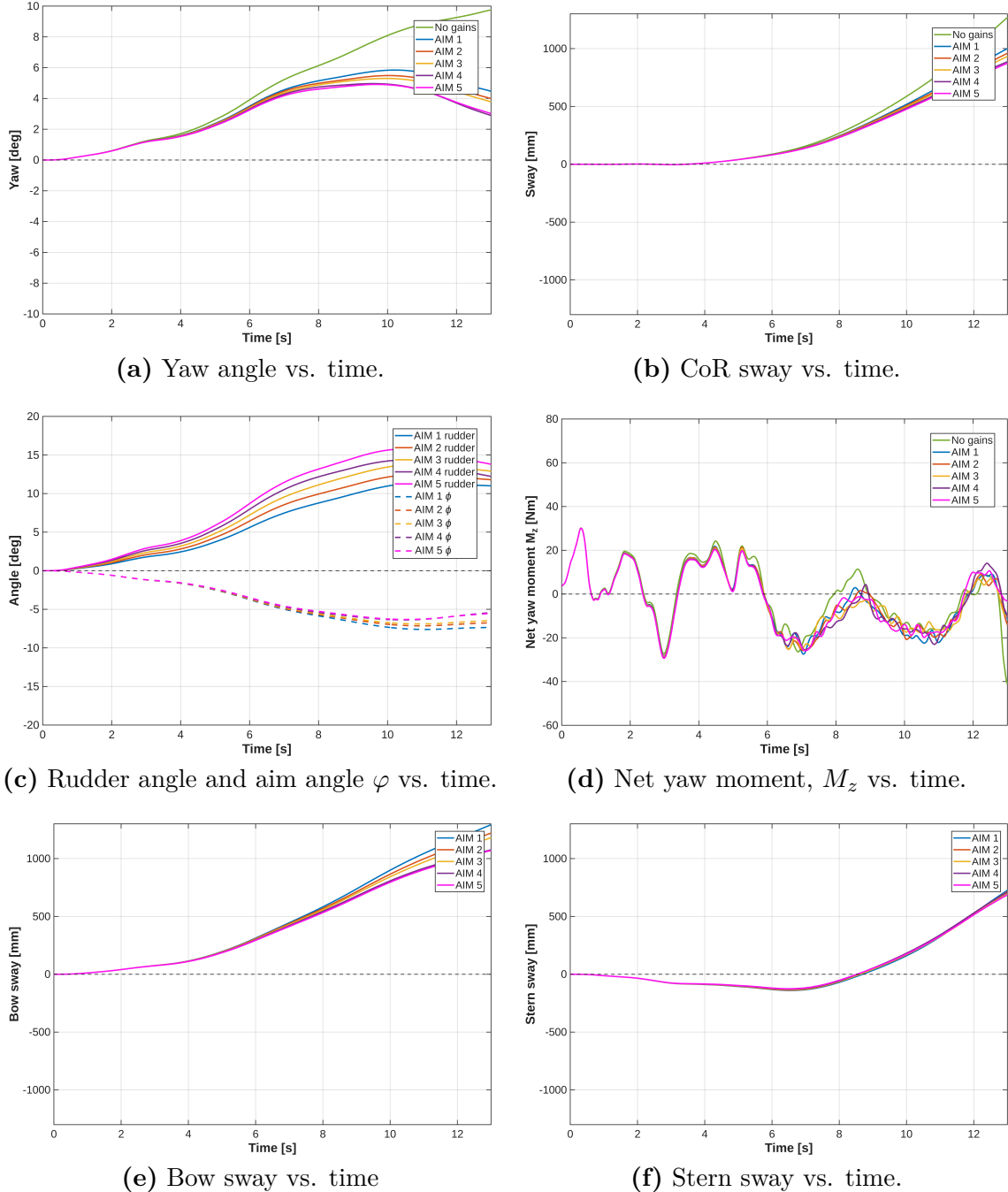


Figure 4.15: Summary of steering related quantities for aim-point cases 1 to 5 with $L = 20$.

Figure 4.15 also indicates that the rudder activation is not large enough to compensate

for the large yaw moment caused by the disturbance, causing a collision in all the cases, at $t \approx 13$ s, although aim-point configuration AIM 5 appears to perform the best in terms of minimum distance between the two hulls. The results are summarized in Table 4.5.

Quantity	AIM 1	AIM 2	AIM 3	AIM 4	AIM 5
Max. CoR sway [mm]	1000.1	959.3	932.6	886.1	872.5
Max. bow sway [mm]	1292.2	1220.1	1179.7	1075.2	1070.1
Max. stern sway [mm]	725.5	714.1	700.3	708.4	686.8
Max. rudder angle [deg]	11.41	12.50	13.79	14.34	15.79
Min. rudder angle [deg]	0.00	0.00	0.00	0.00	0.00
Max. yaw angle [deg]	5.84	5.49	5.30	4.96	4.91
Max. φ angle [deg]	7.61	7.14	6.89	6.37	6.32

Table 4.5: Summary of steering extrema for the aim-point cases with $L = 20$ m over $t = 0$ –13 s. Sway, yaw and φ maxima are taken of absolute values.

Shortening the look-ahead distance further causes the rudder activation to increase even more, as shown in Figure 4.16, which shows the results for steering with look-ahead distance of $L = 8$ m.

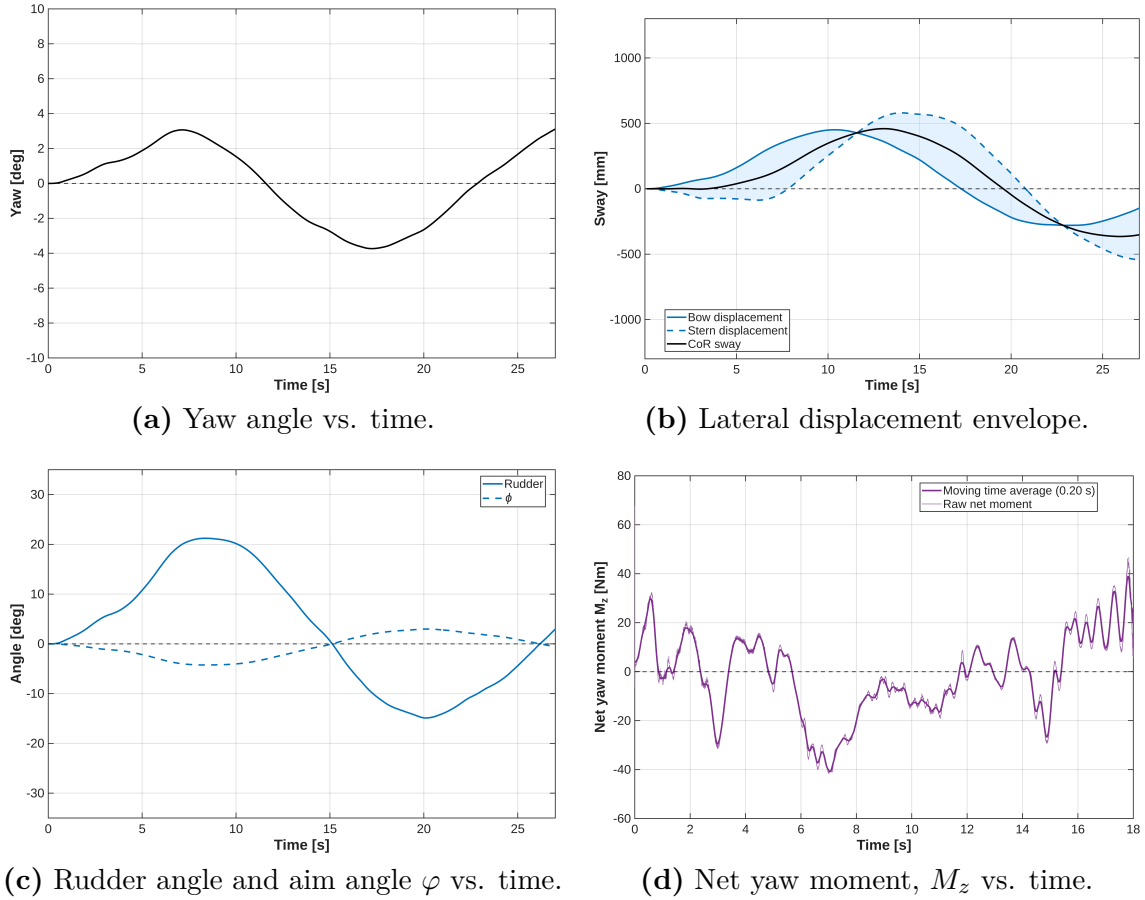


Figure 4.16: Summary of steering related quantities, AIM6 ($k = 5$, $L = 8$).

Figure 4.17 shows the results of the AIM6 configuration. With the shorter look-ahead distance of $L = 8$ and the higher gain $k = 5$, the performance of the aim-point steering model increases noticeably. This combination successfully counters the initial yawing moments and corrects the trajectory, despite the large disturbance from the mapping failure, and safely guides the hull through the encounter.

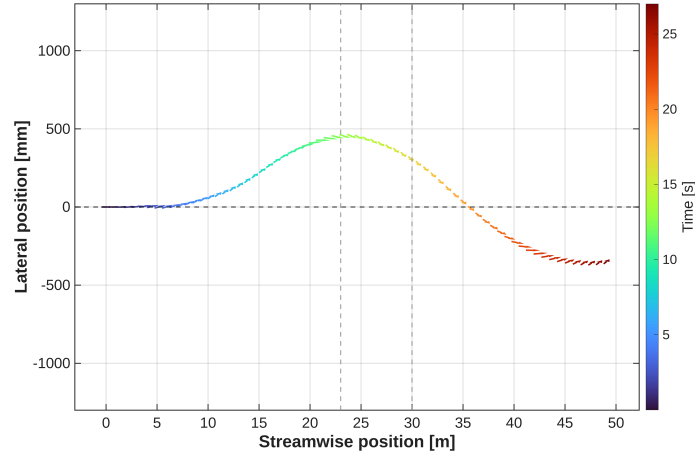


Figure 4.17: Trajectory plot, $k = 5$, $L = 8$. Left vertical line indicates when the two hull bows coincide in streamwise position, right line indicates when the two hull sterns coincide in the streamwise position.

The maximum values are listed in Table 4.6. The lower magnitude of the values shows that the current aim-point calibration outperforms the ones listed in Table 4.5.

Quantity	Max	Min	t_{max}	t_{min}
Yaw [deg]	3.112	-3.735	27.000	17.238
CoR sway [mm]	459.414	-364.818	13.042	25.931
Bow sway [mm]	450.969	-279.859	10.40	23.000
Stern sway [mm]	580.118	-542.837	14.000	27.000

Table 4.6: Extreme values of the hull motion over $t = 0-27$ s, together with the times at which they occur. ($L = 8$, $k = 5$).

4.3.4 PID

Of the four PID calibrations, only PID1 was evaluated for the encounter due to time restrictions. Figure 4.18 shows that the selected PID settings are unstable and cause oversteering. Out of all evaluated steering methods in Stage 3, PID1 shows the largest rudder activation, which gets clamped by the 35° -limiter for approximately 7 seconds, causing the hull to avoid the second hull, but to later collide with the starboard channel bank after around 18 seconds. The collision with the wall is also reflected in the net yaw moment plot, where it can be seen that the hull experiences a strong positive yaw moment as the bow approaches the starboard bank.

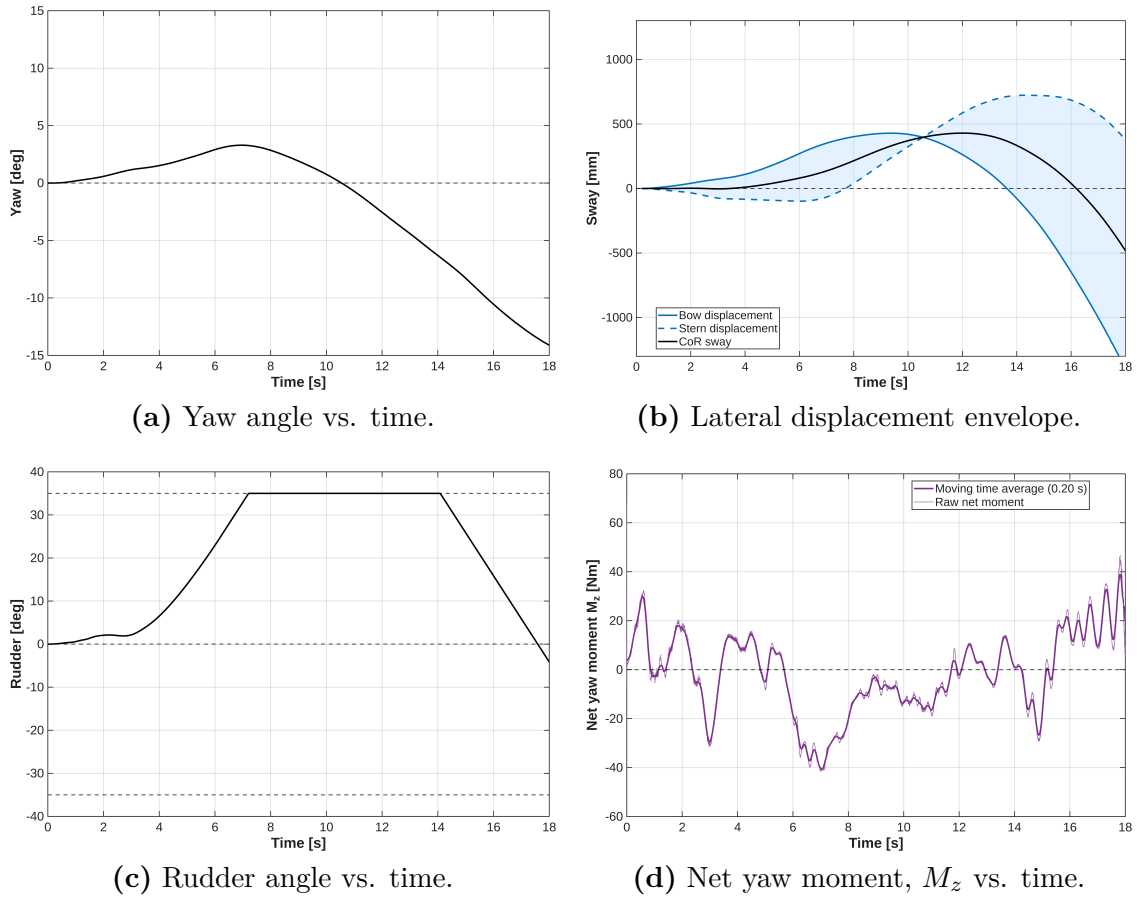


Figure 4.18: Summary of steering related quantities, PID1.

The trajectory is presented in Figure 4.19, which shows how the hull steers back towards the centerline after around $t = 20s$. The hull's sway in the positive lateral direction is less than half the sway in the case with without steering, however the negative sway towards the end of the simulation is larger.

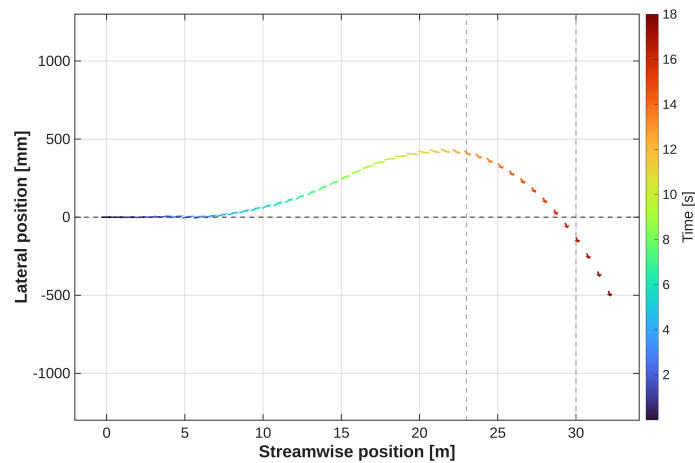


Figure 4.19: Trajectory plot, PID1. Left vertical line indicates when the two hull bows coincide in streamwise position, right line indicates when the two hull sterns coincide in the streamwise position.

5

Conclusion

The goal of this project was to construct a framework, based on OpenFOAM together with the overset mesh method, capable of capturing hydrodynamic interaction effects in confined waterways and of evaluating steering strategies. The workflow was divided into three main stages: a hull-fixed reference frame simulation, a channel-fixed single-hull simulation, and a final channel-fixed encounter simulation. Each stage confirmed a different aspect of the framework.

Stage 1 simulations showed a clear difference in the hull forces, hull moments, and the pressure field around the hull when the hull was centered in the symmetric channel, compared to when the hull was positioned closer to one channel bank. The results also match the expected overall behavior of bank suction effects from the theory, although the accuracy and magnitude of the effects remain unvalidated.

The results from Stage 2 showed that the reference frame shift worked smoothly, and that the presence of the bank causes the hull to displace in accordance with the theory of bank suction. The computational domain was however not long enough to clearly distinguish between the steering calibrations, although the implemented steering influenced the hull in a physically consistent manner.

In the third and final stage, unsuccessful mapping caused an initial disturbance around the hulls, leading to large yawing moments which significantly altered the trajectory of the hull. Since the major contribution to this displacement is due to faulty flow field initialization, the bank's effect on the hull is hard to evaluate. However, the steering results clearly show the differences between the steering methods and make it possible to evaluate them. The aim-point steering model outperforms no active steering and is preferable to the PID in terms of predictability and stability. The results are also consistent, enabling the user to adjust parameters iteratively depending on previous results, as shown by aim-point steering combination AIM6, which successfully passed the other hull.

Furthermore, all cases confirm that the free surface treatment was good enough to capture the expected Kelvin wake shape generated by the hulls, and the wave behavior is consistent with theory, acting as a form of validation for the physics of the setup.

5. Conclusion

The following conclusions can therefore be made:

1. The developed framework is capable of capturing hydrodynamic interaction effects from banks.
2. The developed framework is capable of assessing steering applications.

The accuracy of the results is however not validated, and no conclusions can be made on the reasonability of the magnitudes of the effects. Furthermore, not enough information is available to confidently conclude whether the constructed workflow captures hydrodynamic interaction effects between the two hulls.

In order to investigate whether the framework can capture the interaction effects between two hulls, further development and trials are needed. The channel geometry could be explored as a parametric study, varying the lateral confinement and the water depth to compare deep and shallow water regimes. The overset mesh could be optimized and made narrower in the transverse direction, allowing the two hulls to be placed closer to each other and to banks with reduced overlap between boundaries, increasing interaction effects. A mesh convergence study and more accurate solver settings would also be needed to assess the accuracy of the solution. Additionally, different hull geometries should be evaluated, since hydrodynamic interaction effects depend on how the hulls displace water.

Disturbances in the hull-fixed simulations caused unwanted wave reflection, which contaminated the results and made assembling the final encounter difficult. In order to mitigate this, a coarse extension was therefore added to the inlet, which numerically dampened the propagating disturbances before they could reflect at the inlet. The underlying source of the disturbance was however not eliminated and the propagating waves remain in the final encounter case. Resolving the underlying source remains a possibility for future work.

The biggest flaw of the staged workflow was the mapping of the flow field. The poorly mapped regions near the hulls caused large disturbances and yaw moments on the hulls, causing them to displace. The flow field does recover after a while, however, so an additional intermediate stage could be introduced in which both hulls follow a prescribed straight trajectory until the flow field recovers, with the recovered field then used as the initial condition for the encounter. This would require a longer computational domain and was not done due to the time restriction of this project.

Overall, the workflow and methods used in this project show promising results as a proof of concept and have achieved two out of the three goals of the project. However, future work is needed in order to increase the robustness enough for any practical use.

Bibliography

- [1] Ferziger, J. H., Perić, M., & Street, R. L. (2020). *Computational methods for fluid dynamics* (4th ed.). Springer. <https://doi.org/10.1007/978-3-319-99693-6>
- [2] Davidson, L. (2025). *Fluid mechanics, turbulent flow and turbulence modeling* [Course material]. Chalmers University of Technology, Gothenburg, Sweden. https://www.tfd.chalmers.se/~lada/postscript_files/solids-and-fluids_turbulent-flow_turbulence-modelling.pdf
- [3] Salehi, S. (2022). *PIMPLE algorithm and pimpleFoam solver* [Lecture notes]. CFD with OpenSource Software, Chalmers University of Technology, Gothenburg, Sweden. https://www.tfd.chalmers.se/~hani/kurser/OS_CFD_2022/lectureNotes/PIMPLE.pdf
- [4] Hirt, C. W., & Nichols, B. D. (1981). Volume of fluid (VOF) method for the dynamics of free boundaries. *Journal of Computational Physics*, 39(1). [https://doi.org/10.1016/0021-9991\(81\)90145-5](https://doi.org/10.1016/0021-9991(81)90145-5)
- [5] CFD Direct. (2020). *Interface capturing in OpenFOAM*. Retrieved June 2, 2026, from <https://cfd.direct/openfoam/free-software/multiphase-interface-capturing/>
- [6] OpenCFD Ltd. (n.d.). *Breaking of a dam*. OpenFOAM Tutorial Guide. Retrieved June 2, 2026, from <https://www.openfoam.com/documentation/tutorial-guide/4-multiphase-flow/4.1-breaking-of-a-dam>
- [7] CFD Direct. (2025). *MULES in OpenFOAM*. Retrieved June 2, 2026, from <https://cfd.direct/openfoam/free-software/mules/>
- [8] WOLF Dynamics. (2021). *Multiphase flows modeling in OpenFOAM: Guided tutorials*. Retrieved June 2, 2026, from https://www.wolfdynamics.com/training/mpphase/OF2021/mpphase_2021_OF8_guided_tutorials.pdf
- [9] Almeland, S. K. (2018). *Implementation of an air-entrainment model in interFoam*. In H. Nilsson (Ed.), *Proceedings of CFD with OpenSource Software*. Chalmers University of Technology. https://www.tfd.chalmers.se/~hani/kurser/OS_CFD_2018/SiljeKrekenAlmeland/report_silje.pdf
- [10] Steger, J. L., Dougherty, F. C., & Benek, J. A. (1983). A Chimera grid scheme. In K. N. Ghia & U. Ghia (Eds.), *Advances in Grid Generation* (FED-Vol. 5). American Society of Mechanical Engineers.
- [11] Tisovska, P. (2019). *Description of the overset mesh approach in ESI version of OpenFOAM*. In H. Nilsson (Ed.), *Proceedings of CFD with OpenSource Software*. Chalmers

- University of Technology. https://www.tfd.chalmers.se/~hani/kurser/OS_CFD_2019/Tisovska_Petra/report_TisovskaPetra.pdf
- [12] OpenCFD Ltd. (2023). *Overset*. OpenFOAM Documentation, Version 2312. Retrieved June 2, 2026, from <https://doc.openfoam.com/2312/tools/processing/numerics/overset/>
 - [13] MDO Lab. (n.d.). *Tips and troubleshooting: Overset meshes*. MACH-Aero Framework Documentation. Retrieved June 2, 2026, from https://mdolab-mach-aero.readthedocs-hosted.com/en/latest/machAeroTutorials/overset_tips.html
 - [14] Volupe. (2021). *Overset mesh in Simcenter STAR-CCM+*. Volupe Support Blog. Retrieved June 2, 2026, from <https://volute.com/simcenter-star-ccm/overset-mesh-in-simcenter-star-ccm/>
 - [15] Chan, W. M., Gomez, R. J., Rogers, S. E., & Buning, P. G. (2002). *Best practices in overset grid generation* (AIAA Paper 2002-3191). 32nd AIAA Fluid Dynamics Conference and Exhibit, St. Louis, Missouri. https://www.nas.nasa.gov/assets/nas/pdf/staff/Rogers_S_Best_Practices_in_Overset_Grid_Generation.pdf
 - [16] OpenCFD Ltd. (2022). *OpenFOAM v2212: New and improved numerics*. OpenFOAM News. Retrieved June 2, 2026, from <https://www.openfoam.com/news/main-news/openfoam-v2212/numerics>
 - [17] Shen, Z., Wan, D., & Carrica, P. M. (2015). Dynamic overset grids in OpenFOAM with application to KCS self-propulsion and maneuvering. *Ocean Engineering*, 108. <https://doi.org/10.1016/j.oceaneng.2015.07.035>
 - [18] OpenCFD Ltd. (2023). *interFoam*. OpenFOAM v2306 documentation. <https://doc.openfoam.com/2306/tools/processing/solvers/rtm/multiphase/interFoam/>
 - [19] SimFlow Technologies. (n.d.). *overInterDyMFoam – OpenFOAM solver*. SimFlow CFD documentation. <https://help.sim-flow.com/solvers/over-inter-dym-foam>
 - [20] Rabaud, M., & Moisy, F. (2013). Ship wakes: Kelvin or Mach angle? *Physical Review Letters*, 110(21), 214503. <https://doi.org/10.1103/PhysRevLett.110.214503>
 - [21] Wrzask, K. (2023). *On the vessel energy requirement prediction from the acceleration stage towing experiments on models* [Preprint]. arXiv. <https://arxiv.org/abs/2308.04200>
 - [22] International Towing Tank Conference. (2002). *Free-sailing model test procedure* (ITTC Recommended Procedures and Guidelines No. 7.5-02-06-01). <https://ittc.info/media/2081/75-02-06-01.pdf>
 - [23] Lataire, E., Vantorre, M., & Delefortrie, G. (2018). The influence of the ship's speed and distance to an arbitrarily shaped bank on bank effects. *Journal of Offshore Mechanics and Arctic Engineering*, 140(2), 021301. <https://doi.org/10.1115/1.4038804>
 - [24] Delefortrie, G., Verwilligen, J., Eloot, K., & Lataire, E. (2024). *Bank interaction effects on ships in 6 DOF* [Manuscript].

-
- [25] Lataire, E., & Vantorre, M. (2008). Ship-bank interaction induced by irregular bank geometries. In *Proceedings of the 27th Symposium on Naval Hydrodynamics*, Seoul, Korea, 5–10 October 2008.
- [26] De Marco Muscat-Fenech, C., Sant, T., Zheku, V. V., Villa, D., & Martelli, M. (2022). A review of ship-to-ship interactions in calm waters. *Journal of Marine Science and Engineering*, 10(12), 1856. <https://doi.org/10.3390/jmse10121856>
- [27] Zou, L., & Larsson, L. (2013). Numerical predictions of ship-to-ship interaction in shallow water. *Ocean Engineering*, 72. <https://doi.org/10.1016/j.oceaneng.2013.06.015>
- [28] Denny, M. E. (1951). B.S.R.A. resistance experiments on the Lucy Ashton: Part I – Full-scale measurements. *Transactions of the Institution of Naval Architects*, 93.
- [29] Conn, J. F. C., Lackenby, H., & Walker, W. P. (1953). B.S.R.A. resistance experiments on the Lucy Ashton: Part II – The ship-model correlation for the naked-hull conditions. *Transactions of the Institution of Naval Architects*, 95.
- [30] Lopes, R., Eslamdoost, A., Bensow, R. E., Ponkratov, D., Kömpe, A., Geremia, P., Pekküçük, Ç. B., Aydın, C., Villa, D., Ntouras, D., Seo, D. C., Rosano, G., Vesting, F., Bigini, G., Chillice, G., Kaufmann, J., Lin, J., Muralha, J., Dott, L., ... Li, Z.-H. (2026). A summary of the Lucy Ashton resistance prediction workshop. *Ocean Engineering*, 343, 122951. <https://doi.org/10.1016/j.oceaneng.2025.122951>
- [31] Göta kanal. (n.d.). *Travel with pleasure boat on the Göta Canal*. Retrieved June 2, 2026, from <https://www.gotakanal.se/en/boatingholiday>
- [32] International Towing Tank Conference. (2014). *Practical guidelines for ship CFD applications* (ITTC Recommended Procedures and Guidelines No. 7.5-03-02-03, Revision 01). <https://ittc.info/media/4196/75-03-02-03.pdf>
- [33] International Towing Tank Conference. (2011). *Practical guidelines for ship CFD applications* (ITTC Recommended Procedures and Guidelines No. 7.5-03-02-03, Revision 00). <https://ittc.info/media/1357/75-03-02-03.pdf>
- [34] OpenCFD Ltd. (n.d.). *Mesh generation with the snappyHexMesh utility* (User Guide, Section 4.4). OpenFOAM Documentation. Retrieved June 5, 2026, from <https://www.openfoam.com/documentation/user-guide/4-mesh-generation-and-conversion/4.4-mesh-generation-with-the-snappyhexmesh-utility>
- [35] Lekkas, A. M., & Fossen, T. I. (2013). Line-of-sight guidance for path following of marine vehicles. In O. Gal (Ed.), *Advances in Marine Robotics*. LAP LAMBERT Academic Publishing. https://www.fossen.biz/php/research/path_following.php
- [36] Han, B., Duan, Z., Peng, Z., & Chen, Y. (2024). A ship path tracking control method using a fuzzy control integrated line-of-sight guidance law. *Journal of Marine Science and Engineering*, 12(4), 586. <https://doi.org/10.3390/jmse12040586>
- [37] Benderius, O. (2012). *Driver modeling: Data collection, model analysis, and optimization* [Licentiate thesis]. Chalmers University of Technology, Gothenburg, Sweden. <https://publications.lib.chalmers.se/records/fulltext/157634.pdf>

- [38] Coulter, R. C. (1992). *Implementation of the pure pursuit path tracking algorithm* (Tech. Rep. CMU-RI-TR-92-01). Robotics Institute, Carnegie Mellon University. https://www.ri.cmu.edu/pub_files/pub3/coulter_r_craig_1992_1/coulter_r_craig_1992_1.pdf
- [39] Kpower. (2026). *How many degrees can a marine rudder turn? (Standard rotation angles and practical limits)*. Kpower Industry Insights. Retrieved June 5, 2026, from https://www.kpower.com/insight_servo/7398.html



CHALMERS
UNIVERSITY OF TECHNOLOGY