EBOOK

# Engineer's Guide to Simulating Marine Engineering

by Cadence

# Introduction

Marine engineering is a discipline focused on the design, construction, operation, and maintenance of marine vessels and structures. The marine industry currently faces numerous challenges, such as achieving environmental sustainability, complying with evolving regulations, and enhancing vessel efficiency. Computational fluid dynamics (CFD) has increasingly become critical in addressing these challenges. Among the myriad tools available to engineers, Cadence's Fidelity CFD platform stands out. It offers accuracy, automation, acceleration. and artificial intelligence integration, setting a new standard in computer-aided engineering.

The Fidelity CFD platform features Fidelity Fine Marine, advanced CFD software specifically designed to address the unique challenges faced by marine engineers. This tailored solution provides a virtual environment for simulating fluid flow around marine vessels, from yachts and sailboats to commercial ships and tankers. With advanced automation features, high-performance computing (HPC) capabilities, and dedicated workflows, Fidelity Fine Marine enables marine engineers to overcome the limitations of conventional design methods.

The importance of CFD simulation in modern marine engineering cannot be overstated. Fidelity Fine Marine allows for the detailed analysis of propulsion, resistance, seakeeping, and maneuvering, directly impacting vessel performance, safety, and environmental footprint. By replacing experimental facilities with numerical sea trials, it empowers engineers to design and optimize ships at full scale, under a wide range of conditions, with an unprecedented level of accuracy.

This guide showcases the marine simulation process using Fidelity Fine Marine. We will explore the key features and applications of the software for marine design and analysis. Whether you are optimizing the hull form for reduced drag and fuel consumption, analyzing the hydrodynamic performance of a new hydrofoil design, or ensuring compliance with international emission standards, Fidelity Fine Marine offers the capabilities necessary to excel in today's competitive marine industry.

# Fundamentals of Marine Engineering

Before we delve into the features of Fidelity Fine Marine, let's review the basic principles of marine engineering. These principles are essential for effectively simulating real-world marine applications. This section explores the primary aspects of the field that are vital for any practitioner.

## Hydrodynamics

Hydrodynamics, the study of fluids in motion, forms the basis of marine engineering. This subject is essential for understanding the water flow around various marine structures, including ships, submarines, and offshore platforms. Hydrodynamics plays a crucial role in the design and analysis of these structures, ensuring they can move efficiently through water, withstand ocean currents, and maintain stability in various sea conditions. The principles of hydrodynamics are applied to calculate resistance, propulsion efficiency, and the effects of waves on vessels. This involves concepts such as buoyancy, which determines whether a vessel will float or sink; drag, which affects speed and fuel efficiency; and lift, which can be harnessed in designs like hydrofoils to reduce drag.

## Types of Marine Vessels and Key Components

Building on the principles of hydrodynamics, let's explore the diverse array of marine vessels and their primary components. There are many types of marine vessels, from gigantic oil tankers and container ships to sophisticated high-speed boats; each is designed for specific purposes and environments. Marine vehicles designed for voyaging across seas are generally classified into two broad groups: those purposed for transport, which includes cargo ships, container ships, and passenger liners, and those intended for non-transport functions such as fishing boats, support vessels like tugs and supply ships, and military naval ships. Figure 1 illustrates the terms used to describe boat geometry, which are defined below.

- Deadrise: The angle between the bottom of a boat or ship and a horizontal plane, indicating the hull's shape. A higher deadrise angle improves wave-cutting capabilities, leading to a smoother ride in rough waters, but can reduce stability at rest.
- Keel: The central structural backbone of a vessel's hull, running along the bottom from the bow (i.e., front) to the stern (i.e., back). It provides stability, helps the boat maintain a straight path through the water, and is the reference point for measuring the draft.
- Beam: The vessel's width measured from its widest point, which is typically the hull. A wider beam increases stability and interior space but can affect a boat's handling and speed.
- Draft: The vertical distance between the waterline and the lowest point of the hull (bottom of the keel). It indicates the minimum depth of water a vessel needs to float and is crucial for navigation in shallow waters.
- Freeboard: The vertical distance between the waterline and the upper deck level, measured at the lowest point where water can enter. It represents a vessel's safety margin, indicating how high the sides are above the water and its capacity to handle waves without taking water on board.

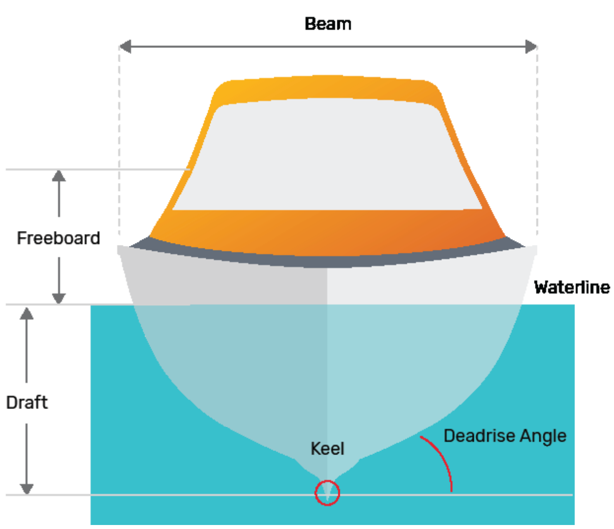



*Figure 1: Terminology describing a boat's geometry*

# Hull Design

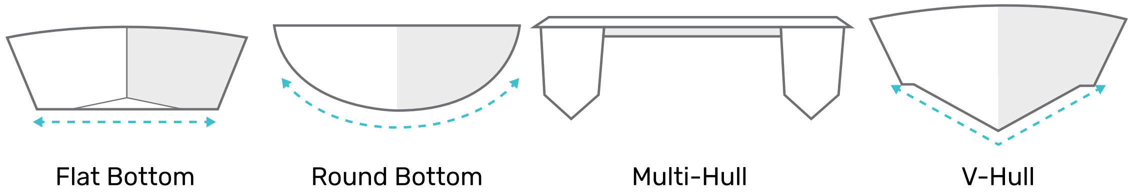



*Figure 2: Types of hull shapes*

The shape of a ship's hull is one of the most significant factors in its overall performance. The hull design determines a vessel's load-carrying capacity, resistance to water, and speed. There are three categories of hulls: displacement, planing, and semi-displacement.

- Displacement Hull: These hulls are designed to move through the water by displacing a volume of water equal to the vessel's weight. Displacement hulls can be found on large, ocean-going vessels such as cargo ships and tankers. They typically have a deep round bottom hull that provides stability and can carry heavy loads. These vessels are not designed to plane and generally have a maximum speed determined by their hull length; this is known as the hull speed.
- Planing Hull: Planing hulls are designed to rise and glide on top of the water at higher speeds rather than push through it. This is achieved using hydrodynamic lift to raise the hull out of the water, reducing contact and drag. These hulls are often flat-bottomed and are used in smaller, faster boats such as speedboats and motorboats. A planing hull allows for higher speeds but generally provides less stability at lower speeds compared to displacement hulls.
- Semi-Displacement Hull: Semi-displacement hulls are commonly v-shaped and combine features of both displacement and planing hulls. They can displace water like a displacement hull at slower speeds, providing stability and efficiency for cruising. As speed increases, they can partially rise out of the water, reducing drag and allowing for higher velocities than a pure displacement hull. These hulls offer a compromise between the carrying capacity and efficiency of displacement hulls with some of the speed capabilities of planing hulls, making them suitable for a variety of medium-sized vessels such as fishing boats and ferries.

Different hull shapes (Figure 2), such as flat bottom, round bottom, v-bottom, and multi-hull designs, are suitable for different purposes and water conditions. For instance, a flat-bottom hull is optimal for calm waters and shallow drafts, while a V-shaped hull is better for rough seas. Multi-hull designs, like catamarans, offer high stability and speed. Therefore, hull design is a complex optimization challenge that balances many different performance aspects.

- Flat Bottom Hull: These hulls have a flat surface along their base and are characterized by their shallow draft, which allows them to navigate in shallow waters easily. They provide a stable platform when stationary or at low speeds, making them ideal for activities like fishing in calm waters. However, their reduced ability to cut through waves can lead to rough rides in choppy conditions.
- Round Bottom Hull: Round bottom hulls, often found on sailboats and larger vessels, can cut through water and offer a smoother ride in rough conditions. They typically have a more pronounced displacement and are known for their superior seakeeping abilities. The drawback is that they can be less stable when at rest and often require a keel or other stabilizing mechanisms.
- Multi-Hull: This category includes catamarans (two hulls) and trimarans (three hulls), which provide high stability and speed. The wider beam created by the separate hulls offers a more stable platform and can achieve greater speeds by reducing the overall hydrodynamic resistance. Multi-hulls are often more efficient, with less drag at higher speeds, but can be more complex to maneuver in tight spaces.
- V-Hull: V-hulls are characterized by the V-shaped cross-section from the bow to the stern. This design allows the hull to cut through water efficiently, resulting in a smoother ride in rough sea conditions. However, v-hulls typically require more power than other hull types to achieve planing and maintain speed.

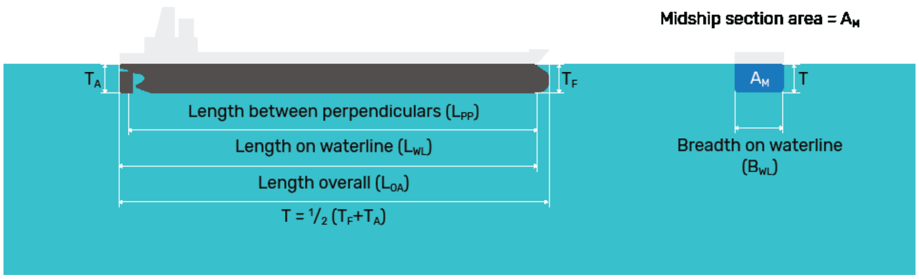



*Figure 3: Hull dimensions*

The hull dimensions and coefficients directly impact the hydrodynamic performance of marine vessels. As depicted in Figure 3, the hull dimensions typically include the following:

- Length Between Perpendiculars ($L_{PP}$): The distance measured along the waterline from the forward perpendicular, typically at the bow, to the aft perpendicular, usually at the rudder post or the stern. It represents the effective length of the ship's hull.
- Length on Waterline ($L_{WL}$): The distance measured along the waterline from the forwardmost to the aftmost point of the hull. This dimension signifies the length of the ship's hull at the water's surface.
- Length Overall ($L_{OA}$): The maximum length of a ship measured from the forwardmost point of the bow to the aftmost point of the stern, including any overhanging parts.
- Breadth on Waterline ($B_{WL}$): The maximum width of a ship's hull measured at the waterline. It indicates the widest horizontal distance across the hull where it meets the water.
- Draft (T): The vertical distance between the waterline and the bottom of the hull (keel), with the average draft taken as the mean of the forward ($T_F$) and aft ($T_A$) drafts.

The hull coefficients, illustrated in Figure 4, quantify the fullness of the hull form in various dimensions. These coefficients affect design considerations for stability, speed, and fuel efficiency.

- Volume of Displacement (∇): The volume of water displaced by the submerged portion of a ship's hull when it is floating, which is directly related to the ship's buoyancy and weight.
- Block Coefficient ($C_B$): The ratio of the displacement volume to the product of $L_{PP}$, $B_{WL}$, and T. This dimensionless ratio measures how full or "block-like" a ship's underwater hull is by comparing the actual displaced volume of the hull to the volume of an imaginary rectangular block with the same length, breadth, and draft as the ship.
- Waterline Area ($A_{WL}$): The horizontal cross-sectional area of a ship's hull at the waterline. It signifies the area of the hull that is in contact with the water when the ship is afloat.
- Waterplane Area Coefficient ($C_{WL}$): The ratio of the waterplane area ($A_{WL}$) to the product of $L_{WL}$ and $B_{WL}$. It represents the fullness of the waterplane area.
- Midship Section Area ($A_M$): The vertical cross-sectional area of a ship's hull at its widest point, typically at the midpoint of the vessel's length. It symbolizes the largest vertical slice of the submerged hull.
- Midship Section Coefficient ($C_M$): This coefficient characterizes the fullness of a ship's midship section. It is defined as the ratio of the hull's cross-sectional area at midship ($A_M$) to the product of $B_{WL}$ and T.
- Longitudinal Prismatic Coefficient ($C_P$): The ratio of displacement volume to the product of $A_M$ and $L_{WL}$. It compares the volume of a ship's displacement to the volume of a prism with the same length as the ship and a cross-sectional area equal to $A_M$. This coefficient indicates how the hull's volume is distributed along its length.

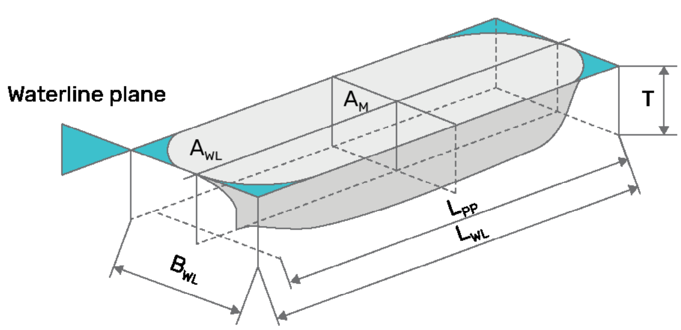



*Figure 4: Hull coefficients*

# Marine Propulsion Systems

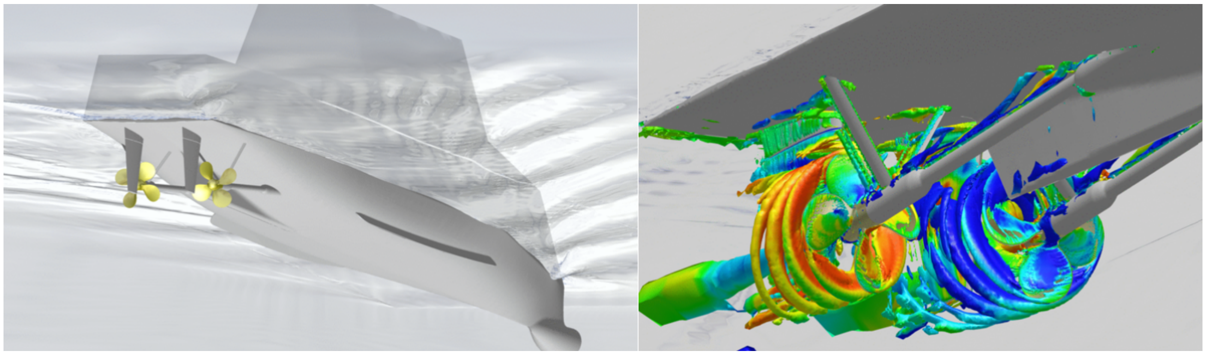


*Figure 5: Self-propulsion system of Office of Naval Research (ONR) tumblehome ship*

Marine propulsion systems are the heart of maritime vessels, providing the necessary force to move a vessel through water. These systems encompass all the components that work together to generate thrust, allowing a vessel to maneuver and maintain its course. Propulsion systems have evolved significantly over time, from the use of sails and oars to sophisticated engines and propulsion technologies. As shown in Figure 5, the choice of propulsion system in marine engineering is dictated by the vessel's intended use, size, speed requirements, and operational environment. The most common types of marine propulsion systems include internal combustion engines, electric propulsion, and renewable energy propulsion. Furthermore, advanced technologies like azimuth thrusters and waterjets provide enhanced maneuverability and speed, catering to the specific needs of different vessel types.

## Internal Combustion Engines

Marine vessels have traditionally relied on internal combustion engines for propulsion due to their high efficiency and robust power output. Diesel engines are the mainstay in commercial shipping and military vessels and are valued for their durability and power density. Gas turbines are another kind of internal combustion engine with high power output suitable for fast ships and naval vessels. However, they generally have higher fuel consumption than diesel engines. A novel approach to marine propulsion is the use of dual-fuel engines, which can switch between gas and diesel. They provide operational flexibility and the potential for reduced emissions through cleaner fuel options.

## Electric and Hybrid Propulsion

The push towards sustainability has spurred the development of electric and hybrid propulsion systems in marine applications. Electric propulsion relies on large battery banks to power electric motors. These systems emit zero direct emissions and greatly reduce noise, making them well-suited for short-range operations, especially in ecologically sensitive areas. Additionally, hybrid systems represent a bridge between traditional and contemporary technologies, combining the reliability of internal combustion engines with the efficiency and quietness of electric motors. For example, stand-alone electric motors can be operated independently from the main power source (e.g., engine or generator). This setup enables flexible operation with the ability to switch power sources based on operational needs.

## Renewable Propulsion

Renewable energy sources are also harnessed for marine propulsion to decrease reliance on fossil fuels and minimize the ecological impact of maritime operations. Wind power, utilized through sails, is a time-honored method now enhanced with modern technology, such as automated sail systems for cargo ships. Solar power is also being adopted, with photovoltaic panels installed on vessels to generate electricity for propulsion and onboard power needs, which is particularly useful for smaller vessels. Additionally, wave and tidal power technologies capture the kinetic energy from natural water movements, converting it into mechanical or electrical energy for propulsion. These renewable sources represent the forefront of eco-friendly marine propulsion, offering clean, sustainable, and cost-effective alternatives to traditional systems. As environmental considerations become increasingly important, the development of more efficient and sustainable propulsion methods will continue to be a key trend in the industry.

# Environmental Considerations

The design and operation of marine vessels require a deep awareness of environmental conditions. Engineers should be able to design marine structures that can withstand nature's forces to ensure they are efficient and resilient against the unpredictability of the seas.

## Wind Effects

Wind forces play a major role in the stability, propulsion, and maneuverability of marine vessels. The marine design process must consider aerodynamics to predict how the wind will interact with the ship's superstructure, which is critical for vessels with high profiles, such as cruise ships, container ships, and sailboats (Figure 6). For sailing vessels, wind is the primary propulsion force and requires detailed aerodynamic analysis to optimize sail and hull design. Wind can also drive wave formation, further influencing vessel performance.

## Ocean Currents and Waves

The movement and behavior of ocean currents and waves considerably impact maritime operations. Currents can influence navigational routes, affecting the duration and the amount of fuel consumed during voyages. Waves, especially during storms or in rough seas, can impose dynamic loads on vessels, affecting their stability and structural integrity. Understanding these elements is vital for designing vessels that can withstand such forces and creating accurate stability analysis models that predict a vessel's performance in various sea conditions.

## Acoustics

The noise generated by ships, submarines, and other marine vessels can disrupt the natural behavior of marine life (e.g., fish and whales), affecting their communication, navigation, and mating patterns. To mitigate these effects, engineers must design vessels with quieter propulsion systems and incorporate noise-reducing technologies. Measures such as sound-dampening materials, optimizing hull shapes, and implementing advanced propeller designs can help reduce underwater noise pollution. By prioritizing the reduction of acoustic disturbances, marine engineers can contribute to preserving the health and balance of marine ecosystems.

## Salinity and Temperature

Variations in temperature and salinity generally have a negligible impact on marine simulations involving large bodies of water, such as those for ship motion and hydrodynamics in open seas. However, these factors can influence marine growth on hulls and accelerate corrosion processes in metallic structures. Thus, careful material selection and protective measures like coating and cathodic protection are necessary. Temperature variations can also affect the performance of mechanical systems and contribute to thermal stresses in materials.

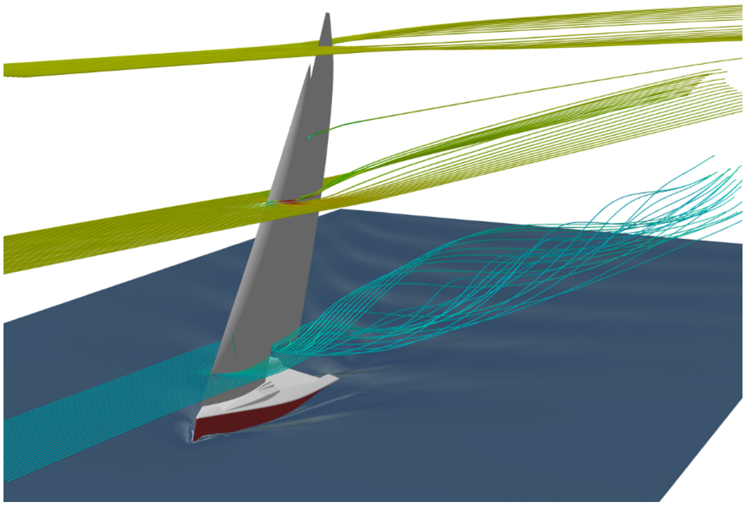


*Figure 6: Wind study of a sailboat*

# Fidelity Fine Marine for Marine Engineering Applications

Having introduced the fundamentals of marine engineering, let's explore why Fidelity Fine Marine is the tool of choice for simulating and predicting the performance of marine vessels. Fidelity Fine Marine by Cadence emerges as the leading marine engineering CFD software, with 15+ years of market experience, 25+ years of solver development, a worldwide community of hundreds of users, and teams of marine engineers dedicated to support and training. Its core functions as a virtual towing tank and wind tunnel, revolutionizing traditional design and testing methodologies. Fidelity Fine Marine allows for direct, full-scale ship design with sophisticated automation features designed for speed and user-friendliness. Its key features and capabilities are listed below.

- High-Performance Fluid Flow Simulations: This efficient, high-performance software is equipped with a marine-dedicated graphical user interface (GUI). It excels in simulating mono- and multi-fluid flows around a diverse array of marine vessels, from simple boats to complex yachts, as well as the intricate design of appendages.
- Innovative Design and Optimization: Leveraging decades of mesh generation and solver development experience, Fidelity Fine Marine provides dedicated CFD workflows that greatly enhance the design and optimization process of any vessel. The software's exceptional free surface flow simulation capabilities make it an invaluable asset for exploring the hydrodynamics of innovative hull designs and optimizing vessel performance under various sea conditions.
- Achieving Efficiency Targets: In line with the International Maritime Organization (IMO) standards, such as the Energy Efficiency Design Index (EEDI) and Energy Efficiency Existing Ship Index (EEXI), Fidelity Fine Marine plays a pivotal role in helping designers calculate these coefficients and meet and surpass energy efficiency and emissions reduction targets. Its capabilities enable thorough assessment and exploration of design alterations to improve efficiency and evaluate the cost implications of modifications before prototypes are built or retrofits are undertaken.
- Advanced Meshing and Automation: With the inclusion of the C-Wizard tool, users can automate the setup of calculation matrices to accommodate different speeds, angles, sea conditions, and geometrical variations. The automation extends to resistance calculations, seakeeping simulations, self-propulsion tests, and more, dramatically reducing engineering time and costs. The software supports a range of meshing strategies, including adaptive grid refinement, which is essential for capturing the dynamic interactions of moving bodies, free surfaces, and multiphysics phenomena.
- Broad Applications: Beyond standard design and analysis, Fidelity Fine Marine supports extensive R&D efforts, including foil development and seakeeping enhancements. Its wind study capabilities further allow marine engineers to forgo traditional wind tunnel testing and optimize designs for passenger comfort and vessel safety.

Fidelity Fine Marine unlocks new potential in marine engineering, enabling the industry to push the boundaries of performance, efficiency, and innovation. From achieving sleeker hull designs to enhancing propulsion systems, the software stands as a testament to Cadence's commitment to advancing marine engineering.

### Streamlining Marine Simulations with the C-Wizard

Fidelity Fine Marine's C-Wizard tool accelerates the setup process for marine flow simulations by automating the entire analysis. This powerful feature considerably reduces engineering time using scripts to automatically configure simulations for a diverse range of marine applications, including resistance evaluation at varying Froude numbers, seakeeping tests, propulsion assessments, and planing hull analyses. The C-Wizard supports simulations for all types of vessels, from tankers and container ships to highspeed crafts and luxury yachts. Users are only required to specify the physical inputs, simplifying the initial setup.

# Addressing Industry Challenges

The maritime industry faces many challenges, ranging from decreasing fuel consumption to regulatory compliance. Cadence's Fidelity CFD solutions are pivotal in addressing these challenges and provide a robust platform for innovation, optimization, and comprehensive analysis.

## Cavitation

Cavitation is a common phenomenon in marine engineering characterized by the formation of vapor bubbles when a liquid's local static pressure drops below its vapor pressure. As the pressure decreases, the bubbles expand. These bubbles collapse violently when they move into higher pressure areas, leading to localized damage to the surrounding surfaces. In marine applications, cavitation occurs primarily around propellers, pumps, hydrofoils (Figure 7), and other submerged components, impacting their performance and efficiency. The detrimental effects of cavitation include erosion of propeller blades, decreased propulsion efficiency, and increased noise levels. Fidelity Fine Marine allows engineers to optimize designs and minimize cavitation-induced damage.

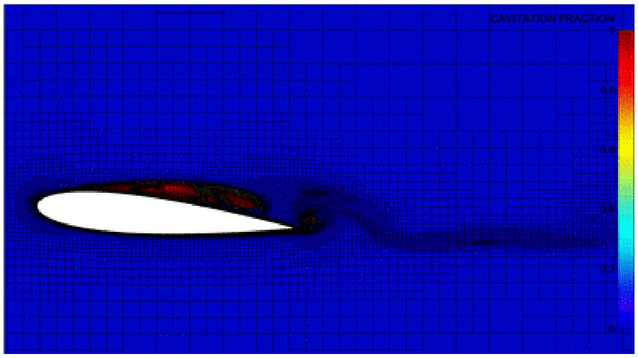



*Figure 7: Simulation of hydrofoil with cavitation pockets using adaptive grid refinement*

## Fuel Consumption Reduction

With rising fuel costs and growing environmental concerns, reducing fuel consumption has become a critical goal for the maritime industry. Fidelity Fine Marine offers sophisticated strategies for analyzing and optimizing vessel propulsion systems, including detailed simulations of propeller performance (Figure 8). These simulations allow for the precise modeling of propeller fluid dynamics, enabling the identification of design modifications that improve propulsion efficiency. The outcome is a reduction in fuel consumption, leading to cost savings and a lower environmental footprint. Through such accurate simulations, Cadence provides a pathway to more sustainable maritime operations.

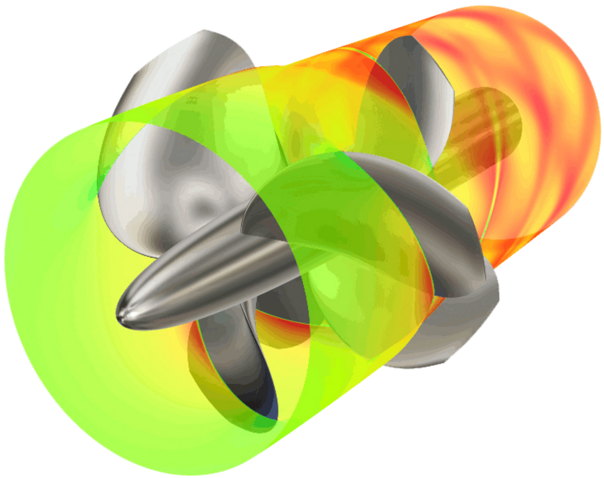


*Figure 8: Contra-rotating marine propeller simulation*

## Complex and Large-Scale Simulations

The complexity of maritime environments necessitates the consideration of numerous variables, from ocean currents and wave patterns to wind forces. Addressing this complexity efficiently requires the capability to perform automated, large-scale simulation runs. Fidelity Fine Marine excels in this area, offering automation features that streamline the simulation process. This automation enables the rapid analysis of design options under a wide range of environmental conditions, substantially decreasing the time and effort required for setup and analysis. As a result, engineers can swiftly iterate designs and conduct extensive optimization studies so that vessels perform optimally and reliably.

## Regulatory Compliance

In the pursuit of maritime advancement, regulatory compliance is a key consideration. Fidelity Fine Marine provides an essential framework to ensure vessel designs adhere to the latest maritime regulations and standards. This advanced tool allows engineers to validate and adjust their designs to meet stringent regulatory criteria. By integrating compliance into the core of the design process, Cadence ensures that vessels achieve peak performance in hydrodynamics and efficiency and fulfill their obligations towards environmental stewardship and passenger safety. The implementation of these regulations guides the path toward a more responsible and sustainable maritime future.

Cadence is reshaping marine engineering by addressing key challenges. Fidelity Fine Marine empowers engineers to design vessels that are economically viable, environmentally friendly, and equipped to thrive in the dynamic and demanding conditions of the world’s oceans. Through continuous innovation and the application of advanced simulation technologies, the maritime industry is poised to navigate the challenges of the 21st century with confidence and ingenuity.

## Setting Up Marine Simulations

The process of setting up CFD simulations for marine applications involves several key steps. Each step is designed to ensure that the simulation accurately captures the complexities of marine environments and delivers meaningful insights into the performance of marine vessels. This section outlines the key stages in setting up marine CFD simulations, from defining objectives to preparing the geometry, mesh generation and management, establishing boundary and initial conditions, and flow model selection.

### Introducing the Dynamic Velocity Prediction Program

Maximizing a sailing yacht’s performance across various wind conditions and angles is crucial, which is where Fidelity Fine Marine’s Velocity Prediction Program (VPP) comes into play. This program maps out a yacht’s performance by analyzing extensive aerodynamic and hydrodynamic data through iterative processing. The Dynamic VPP eliminates many issues found in traditional “static” VPP approaches. This innovative method allows for the optimization of yacht speed across five degrees of freedom in just one calculation. Central features of the Dynamic VPP include the optimization of boat speed using a fixed yaw angle, with rudder adjustments made to maintain yawing moment equilibrium. The outcome is optimized boat speed, sail power control, and detailed analysis results of efforts and motions.

## Defining Simulation Objectives

Before initiating the simulation, it's essential to define the objectives clearly. These objectives can vary widely in marine applications depending on the project's specific needs. Ship motion simulations are commonly performed to predict a ship's movements in response to various forces, such as waves and wind while navigating through water. One of the primary objectives is to assess the vessel's stability to ensure it can withstand rough seas and recover from tilts without jeopardizing safety. Below we describe typical ship motion studies available in Fidelity Fine Marine.

- Resistance: These studies investigate how water resistance impacts a vessel's hull during navigation. They help optimize hull shapes to reduce drag and improve fuel efficiency. Computations can be performed at a single speed, or a complete resistance curve can be generated at different speeds for various forms of resistance, such as frictional, form, wave-making, or air resistance.
- Seakeeping: Seakeeping analyses evaluate how waves affect a ship's motion and stability. They assess a vessel's ability to remain safe and comfortable in varying sea conditions. A single wave is simulated at various advancing speeds (i.e., the velocity at which a vessel or propeller moves through the water). Additionally, a specialized setup with an internal wave generator is available for scenarios where the body has zero advancing speed.
- Open Water: This study is dedicated to the analysis of propeller performance in open water. It allows for the generation of either a single advance ratio (i.e., the ratio of vessel speed to propeller rotational speed) or an entire performance curve describing how a propeller's performance metrics, such as thrust, torque, and efficiency, vary over a range of advance ratios. Furthermore, a bollard pull test can be automatically set up to measure the maximum pulling force that a tugboat or other vessel can exert while stationary. This test is used to evaluate a vessel's towing capability.
- Planing Regime: Planing regime studies involve analyzing the performance and behavior of planing hulls that operate at high speeds and rise and skim on the water surface (e.g., speedboats and racing yachts), which reduces their wetted area and hydrostatic drag. The Savitsky prediction method, a technique for calculating the resistance of prismatic planing hulls, is used to estimate a planing hull's final position for a given speed.
- Trim Optimization: Optimizing a vessel's trim, or longitudinal tilt, can enhance hydrodynamic performance and stability, reduce wear, and boost efficiency. The aim of trim optimization is to identify the ideal initial static trim that reduces drag to a minimum for a set displacement and speed of operation. This enables the operator to effectively arrange the ship's loads and ballasts (i.e., weight added to a vessel), thereby lowering fuel usage.
- Planar Motion Mechanism Maneuvers: The planar motion mechanism (PMM) is a technique used to study the hydrodynamic forces and moments acting on a ship or underwater vehicle during various maneuvers. The PMM maneuver involves controlled movements of the vessel in a towing tank or computational simulation to replicate real-world maneuvers such as zigzag tests, turning circles, and other controlled motions. The primary objective is to input CFD-based PMM maneuver results into a maneuvering model. This data is essential for understanding a vessel's maneuverability and stability characteristics. As shown in Figure 9, the available maneuvers include static drift, pure sway, pure yaw, yaw and drift, and roll decay.
- Self-Propulsion: These simulations are highly beneficial in determining ship power requirements. Since actual propeller thrust measurements for ships are seldom accessible, employing numerical self-propulsion simulations can facilitate a more accurate mapping of desired versus actual thrust more efficiently and cost-effectively. There are multiple ways of simulating self-propulsion depending on the known or unknown variables and how the propulsion system is modeled. Some simulation types involve modeling the real propeller geometry in a rotating domain. An actuator disk can also be used to model the propulsion system's effects on the surrounding fluid without the need for detailed modeling of the propeller blades.

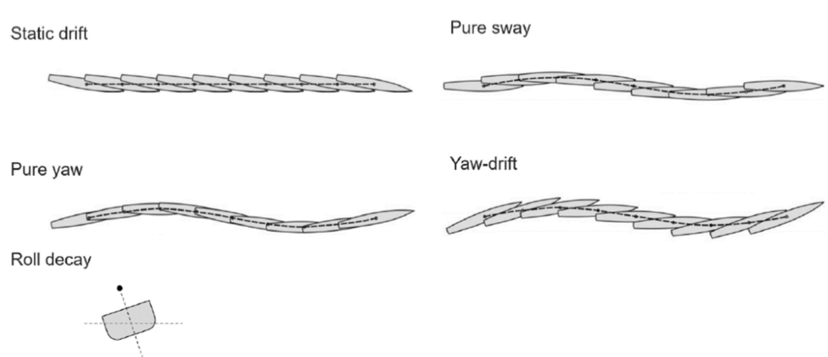



*Figure 9: Illustration of available maneuvers in Fidelity Fine Marine*

# Geometry Preparation

Once the objectives are defined, the vessel geometry is prepared for use within the simulation environment. The preprocessing stage is typically a manually intensive and time-consuming process that involves model creation and simplification. First, a virtual 3D representation of the vessel (Figure 10) is modeled or imported. Then, defeaturing is performed to remove any irrelevant features that do not considerably influence the fluid flow. Finally, holes are sealed to create a watertight geometry, and sharp angles are rounded to ensure optimal mesh quality and convergence. However, Fidelity CFD eliminates the need for manual geometry adjustments by providing various tools to automatically clean and prepare complex or defective geometry for the mesh generation phase.

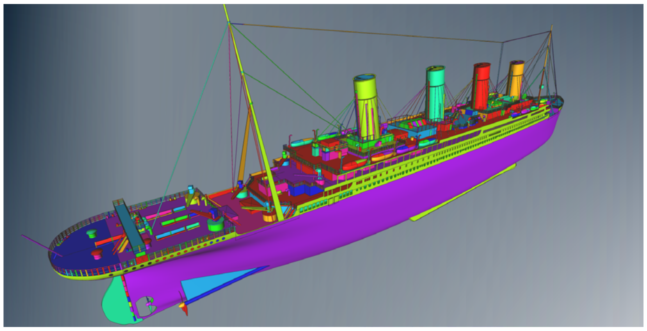


*Figure 10: Fidelity CFD is used to prepare this detailed ferry model*

## Meshing Techniques

After geometry preparation, mesh generation is a critical step in which the geometry is divided into small, discrete computational cells. Several meshing strategies exist for handling complicated geometries in marine applications, which are listed below. As presented in Figure 11, the choice between unstructured full-hexahedral (Figure 12), unstructured hexahedral-dominant (Figure 13), unstructured tetrahedral-dominant, or mixed element grids depends on the complexity of the geometry and the desired mesh quality.

### Accelerating Marine Geometry Preprocessing with Fidelity CFD

The Fidelity CFD platform offers powerful geometry preprocessing tools to simplify, automate, and accelerate the preparation of complex marine models for meshing. Its comprehensive features allow for the import of various CAD formats in parallel from local and remote machines, and advanced analysis to identify and rectify common geometry issues, such as free edges, non-manifold edges, and feature curves. Fidelity CFD ensures geometry conformality and facilitates the creation of watertight geometries by capping holes or gaps. The wrapping feature further aids in creating clean, tessellated surfaces, which is essential for efficient surface-to-volume meshing. Additionally, the platform's automatic grouping or splitting capability organizes the geometry into manageable segments based on surface characteristics, streamlining the workflow.

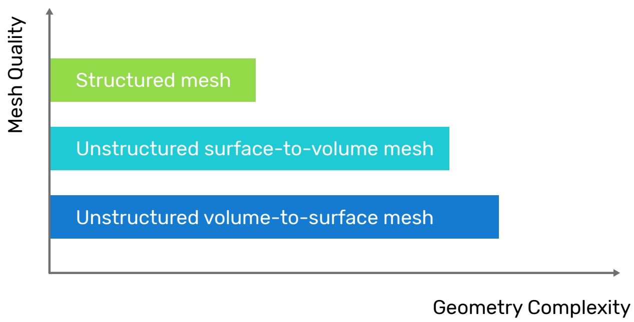



*Figure 11: As geometry complexity increases, unstructured volume-to-surface meshes can better accommodate intricate details compared to structured meshes but with lower mesh quality*

- Unstructured Full-Hexahedral Grids: Consist entirely of hexahedral elements arranged without a structured, regular pattern. Unlike structured grids, unstructured full-hexahedral grids do not adhere to a specific coordinate-based alignment, allowing for greater flexibility in representing complex geometries. They can adapt to intricate details and irregular surfaces, making them suitable for simulations that require high accuracy in specific areas. However, generating a fully unstructured hexahedral grid can be challenging and computationally intensive due to the need to maintain high-quality elements throughout the domain.
- Unstructured Hexahedral-Dominant Grids: Primarily composed of hexahedral elements but may also include other types of elements such as wedges (prisms), pyramids, and tetrahedrons. The hexahedral elements are used wherever possible due to their advantageous properties for numerical simulations, such as lower numerical diffusion and better convergence properties. The flexibility to include other elements allows these grids to adapt more easily to complex geometries while maintaining high-quality meshing. This approach balances the benefits of hexahedral elements with the practical need to mesh complicated shapes.

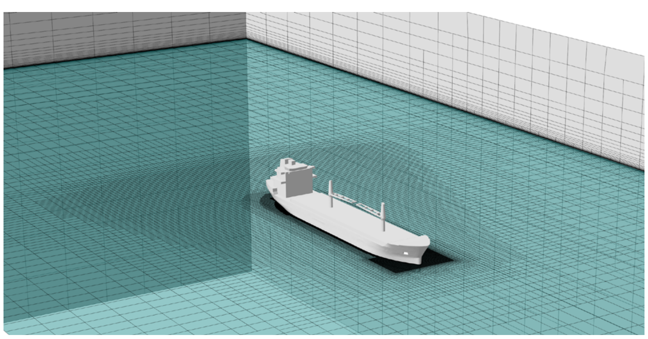


*Figure 12: Volume-to-surface, unstructured full-hexahedral mesh of ship*

- Unstructured Tetrahedral-Dominant Grids: Predominantly contain tetrahedral elements, which are simple to generate and adapt well to highly complex and irregular geometries. Tetrahedral elements can fill space efficiently and are commonly used in unstructured grids due to their flexibility. However, they may require finer meshing to achieve the same level of accuracy as hexahedral elements, potentially increasing computational costs. Tetrahedral-dominant grids are typically used when the geometry is too intricate to be adequately represented by hexahedrons or when rapid meshing is required.

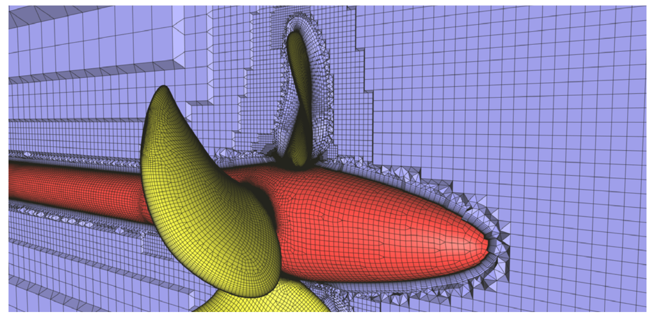


*Figure 13: Unstructured hexahedral-dominant mesh of Potsdam Propeller Test Case (PPTC) propeller*

After defining the mesh type, the appropriate level of fineness or coarseness for the mesh should be determined, which can affect both the accuracy of the simulation and the computational resources required. Fidelity Fine Marine’s adaptive grid refinement (AGR) feature can dynamically adjust the mesh in areas that need higher resolution, such as free surfaces, vortices, or boundary layers, as depicted in Figure 14. AGR saves users hours by automatically determining the optimal mesh density for various configurations.

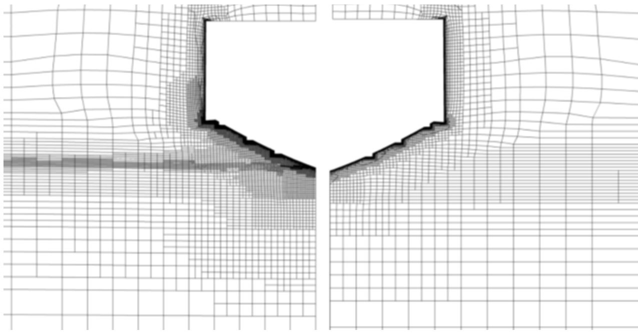


*Figure 14: Example of computation with adaptive grid refinement (left) and initial mesh (right)*

### Mesh Management

This section describes various methods of managing the mesh in Fidelity Fine Marine, from multidomain definitions to adaptive grid refinement.

### Multidomain Definitions

In multidomain projects, the computational domain is divided into separate regions, each of which can be manipulated independently to allow for completely different motions. This approach is useful in scenarios where components have relative motion, such as a ship moving through water with rotating propellers. To ensure the accuracy of the simulation, information about the flow must be transferred between these domains whenever they slide against each other. This transfer is achieved using nonconformal interfaces, which can be grouped together to create sliding patches. These sliding patches allow for the establishment of unique connections between domains.

### Sliding Patch Motion

Sliding patches facilitate the connection between moving and stationary parts of the mesh. The types of connections that can be defined include:

- Rigid: Maintains a fixed relationship between the moving and stationary domains.
- Roll: Allows for rotational motion around the longitudinal axis.
- Pitch: Permits rotational motion around the transverse axis.
- Yaw: Enables rotational motion around the vertical axis.
- Rotation: Accommodates general rotational motion.

These connection types are essential for precisely modeling the relative motion and interaction between different parts of the model.

## Advanced Marine Meshing with Fidelity Automesh

Fidelity Automesh transforms the meshing process with cutting-edge features. From volume-to-surface (V2S) meshing for complex and unclean geometries to surface-to-volume (S2V) meshing for obtaining high-quality surface grids and viscous layers, this software is equipped to handle the intricacies of marine geometries. It offers global and local refinement options to increase mesh resolution where it counts, and various viscous layer insertion techniques to capture the boundary layer physics accurately. With features like anisotropic and adaptive mesh refinement, Fidelity Automesh ensures efficient, high-fidelity simulations, making the journey from design to deployment faster and more reliable.

## Domain Mesh Management

Domain mesh management involves defining mesh displacement to represent the motion of the fluid and solid boundaries. This can include:

- Rigid Motion: The entire mesh moves as a solid body without deformation. This is suitable for translating or rotating objects.
- Weighted Deformations: Different parts of the mesh deform by varying amounts based on predefined weighting factors.

Mesh displacement must be carefully managed to ensure the fidelity of the simulation and to avoid numerical errors.

## Rotating Frame Method

The rotating frame method, also known as the frozen rotor approach, is used to model rotating machinery within the fluid domain. This method involves fixing the rotating frame relative to the rotating components, which simplifies the simulation by reducing the need to account for time-dependent changes in the rotor position. The frozen rotor approach assumes that the rotor's position is constant over a single time step, which is particularly useful for steady-state simulations of rotating machinery.

## Overset Grid Management

The overset grid technique handles complex configurations, particularly those involving large body motions. Known also as the overlapping grid or Chimera method, it allows for the intersection of multiple domains, with the solution being interpolated between them. This method ensures a high-quality mesh in each domain while enabling body movements of unlimited complexity. Its applications are vast and include, but are not limited to, large and intricate movements like ships navigating through waves, movement of appendages, the interaction between rudders and propellers, objects dropping (e.g., lifeboats), operations in confined spaces and during maneuvering situations (e.g., bank effects, shallow waters, rivers, and channels), and ships crossing paths or overtaking each other. Table 1 compares different mesh deformation methods, including the overset technique.

## Adaptive Grid Refinement

AGR is an iterative method that dynamically adjusts the mesh in CFD simulations based on the evolving flow physics. This versatile and automated tool is advantageous for a wide range of marine applications, allowing users to capture complex flow details such as waves, wake flows, boundary layers, and trailing vortices behind ships and propellers. By refining cells only where necessary, AGR speeds up the mesh generation process and significantly reduces the total number of cells compared to static meshes. The grid refinement can be isotropic (i.e., uniform in all directions) or directional, and existing cells are divided based on flow features. This process is repeated during the flow computation at specified intervals. AGR is particularly beneficial in unsteady flow simulations where the grid is adjusted as the flow evolves. This method simplifies the meshing process and ensures mesh continuity, good data transfer, and compatibility with overset grids.

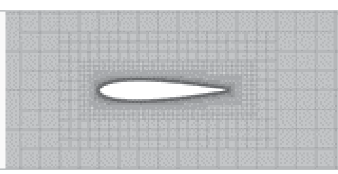
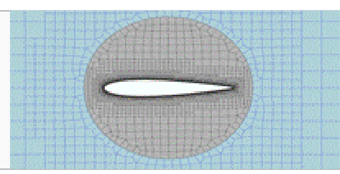
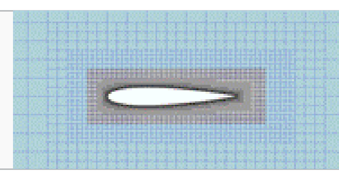
| Weighted Deformation | Sliding Grid | Overset Grid |
|---|---|---|
| • Fast method<br>• No interpolation<br>• Limited by the motions<br>• Recommended for ship trim and sink in calm sea | • Slower than single domain approach<br>• Interpolation: single cell layer at boundaries<br>• Almost limitless for motions<br>• Recommended for propeller rotation | • Slower than single domain approach<br>• Interpolation: multiple cell layers at boundaries<br>• Limitless for motions<br>• Recommended for large rudder executions |
|  |  |  |

*Table 1: Comparison of various mesh deformation methods*

# Boundary and Initial Conditions

Following the mesh generation and management steps, setting precise boundaries and initial conditions is a nuanced task that directly influences the accuracy of CFD simulations. For marine applications, these conditions must reflect the complex interplay of hydrodynamic forces acting on vessels or offshore structures. Below we describe the available boundary conditions in Fidelity Fine Marine.

## Solid Wall

The following solid wall boundary conditions are applied to surfaces that represent physical barriers in the fluid flow, such as a ship's hull.

- No-Slip Wall (zero shear stress): The fluid velocity relative to the wall is zero, meaning the fluid at the boundary sticks to the wall, which is typical for viscous flows.
- Slip Wall (wall-resolved turbulence model): The fluid is allowed to slide along the wall without any frictional resistance. This is used in scenarios where viscous effects are negligible. The wall-resolved approach with a finely meshed model fully resolves the entire boundary layer, from the wall to the turbulent core, providing high-fidelity results ideal for detailed analysis of flow dynamics around the hull.
- Wall Function: Simplified model used to estimate the velocity profile and shear stress within the boundary layer without fully resolving it. This method is suitable for high Reynolds number flows where computational efficiency is important. The corresponding wall roughness can be activated and defined with uniform or varying sand grain height.
- Synthetic Jet: This condition simulates the effect of fluid jets generated by oscillating membranes or diaphragms without the addition of net mass to the system. It's used to define an input flow and simulate active flow control mechanisms where the jet influences the flow dynamics, thereby enhancing mixing or delaying flow separation. The normal blowing direction (e.g., local, averaged, and user-defined) and kinematic law (e.g., constant, pulsed, and user-defined) must be defined.

## External Boundaries

External boundary conditions are applied at the edges of the computational domain where the fluid interacts with the surroundings or enters and exits the simulation area. The key types of external boundary conditions include:

- Far Field: Applied to simulate an open domain where the effects of boundaries are minimal. It is used to represent conditions far away from the object of interest.
- Prescribed Pressure: A fixed pressure value is specified at the boundary. It is commonly used to control the flow rate and ensure accurate pressure distribution within the domain.
- Zero Pressure Gradient: The pressure gradient normal to the boundary is assumed to be zero. This condition is often used at outflow boundaries to allow fluid to exit the domain without affecting the internal flow.
- Wave Generator: Used to simulate incoming waves. It is important for applications where wave interactions with structures are studied.
- Overset: Also known as chimera or overlapping grids, this condition allows the use of multiple overlapping mesh regions to handle complex geometries and moving objects within the domain. It does not influence the flow physics, but it facilitates the transfer of information among the various grids within the simulation.
- Synthetic Jet: The synthetic jet external boundary condition introduces controlled fluid jets into the external flow domain, influencing the surrounding flow dynamics to enhance mixing or control separation.

## Mirror Condition

The mirror boundary condition, also known as the symmetry condition, is used to simulate a plane of symmetry within the computational domain. This condition assumes that the flow on one side of the plane is a mirror image of the flow on the other side, effectively reducing the computational effort by modeling only half (or a portion) of the domain. It is particularly useful in symmetric geometries or flow conditions.

## Non-Conformal Interface

A non-conformal interface is used in simulations involving grids that do not match perfectly at the interface between different mesh regions. This condition allows for the exchange of information between non-matching grids, enabling complex simulations where various parts of the domain require different mesh resolutions. The non-conformal interface ensures continuity of flow variables across the interface while accommodating varying mesh densities and structures.

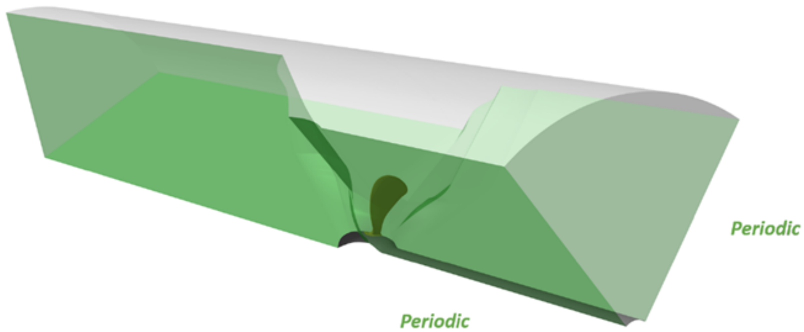



*Figure 15: Periodic boundary conditions (green) are used to simulate open-water propellers*

## Periodic Condition

The periodic boundary condition is employed in simulations where the flow pattern repeats itself at regular intervals. This condition allows the edge of the computational domain to connect seamlessly, effectively creating an infinite domain by repeating the pattern. Periodic conditions are useful in modeling flows in components like marine propellers (Figure 15) and offshore platform risers or any scenario where the flow is expected to be cyclic. Rotation and translation are the two periodicity types compatible with Fidelity Fine Marine.

## Boundary Conditions for Grid Deformation

Boundary conditions for grid deformation control how the mesh deforms independently from the fluid boundary conditions. These include:

- Free (Neumann) Condition: Allows the mesh to deform freely without any constraints.
- No-Slip (Dirichlet) Condition: Fixes the mesh to the fluid boundary, preventing any relative motion.
- Bottom (Shallow Water) Condition: The mesh deformation is evenly distributed between the ship and domain bottoms.
- Mirror Condition: Supports the mirroring of the mesh deformation across a symmetry plane.

## Initial Conditions

In addition to boundary conditions, initial conditions need to be defined, such as the ship's initial velocity components, turbulent quantities, and the interface position. The initial values can be based on parameters derived from previous computations or the uniform values listed below.

- Initial Velocity: The initial speed and direction of fluid flow are specified.
- Turbulent Quantities: The initial turbulent quantities are automatically computed in Fidelity Fine Marine but can also be initialized with custom values. They depend on the fluid parameters and the flow model characteristics. The following is a list of available initial turbulent quantities:
  - $K$ = turbulent kinetic energy ($K$-$\omega$ and $K$-$\varepsilon$ models only)
  - $\omega$ = turbulent frequency ($K$-$\omega$ model only)
  - $\nu_t$ = turbulent viscosity ($K$-$\omega$ and $K$-$\varepsilon$ models only)
  - $\varepsilon$ = turbulent dissipation ($K$-$\varepsilon$ model only)

  The turbulence level (a) is also used to define the turbulence velocity scale ($V_s$), and the turbulence length ($L_s$) corresponds to the reference length for the turbulence initialization. It is recommended to increase the turbulence level for internal flows, but not for external flows.
- Interface Position: For multi-fluid computations, the interface position between the various fluids must be defined. The initial free surface is applied along the Z-axis for 3D simulations and the Y-axis for 2D simulations. Users can customize the location of the free surface by altering a user-defined FORTRAN program in Fidelity Fine Marine.

Various scenarios may require a user to restart a simulation: when it halts unexpectedly, when the duration needs to be extended beyond the original timeframe, or to evaluate new or different physical aspects than those previously considered. Fidelity Fine Marine allows users to initiate a new simulation using data from a different project. By selecting this feature, the solution generated during an initial calculation on a designated grid is transferred to the current grid through interpolation. This transferred solution is then used as the starting point for the ongoing calculation. Numerous situations are ideal for employing a multigrid configuration, including calculations of standard resistance, evaluations of self-propulsion that incorporate sliding grids for rotating propellers, as well as bare and appended versions of identical geometries. If a considerable portion of the computation is performed on a coarse grid with fewer cells, it can lead to a 30% to 60% reduction in CPU time, depending on the specific application.

By thoroughly addressing each step of the simulation setup-from defining clear objectives and preparing geometric models to applying sophisticated meshing techniques and realistic boundary conditions-marine engineers can perform robust CFD simulations to deliver insightful data that drive the optimization of marine vessel design and operation.

# Time Configuration

Once the boundary and initial conditions are set, it is important to choose the appropriate time configuration based on the analysis type. For simulations involving mono-fluid systems where the flow conditions are constant over time (Figure 16), a steady time configuration should be used. If the simulation involves multi-fluid dynamics, a time-marching method can be employed to iteratively reach a steady-state solution where the flow properties no longer change with time. This method treats the steady problem as pseudo-time-dependent to enhance the stability and convergence of the solution. For fully unsteady scenarios, where the flow properties substantially vary over time and cannot be approximated as steady, an unsteady time configuration is necessary. The fully unsteady setup provides access to additional unsteady simulation parameters with first or second-order accuracy in time.

## Time Accurate Computations

Achieving accurate time-resolved computations in marine simulations involves a methodical approach, starting with the initial setup. There are three main approaches to initiate time-accurate computations: begin the simulation from a steady-state solution, start from a previous unsteady solution, or initialize the simulation with constant values.

## Steady-State Solution

Beginning the simulation from a steady-state solution is useful for establishing a baseline before introducing time-dependent variables. A preliminary steady-state computation can be used to capture the system's natural unsteady behavior or obtain an approximate solution. The steady solution can serve as the initial condition for unsteady time-dependent computations with periodic behavior.

## Unsteady Solution

Starting from a previous time-accurate solution is beneficial for continuing or refining previous transient analyses. As a result, the flow solver can utilize the previous unsteady solution values to perform a second-order accurate simulation. For unsteady simulations involving weighted deformations, first fix all degrees of freedom to stabilize the initial computation. Once steady-state conditions are reached, restart the simulation using the parameters determined from the previous computation and free the degrees of freedom to accurately reflect the flow dynamics. Studies can also be performed without blocking deformations, but it is recommended to free up to two degrees of freedom.

## Constant Solution

Initializing the simulation with constant values is suitable for scenarios where the initial conditions are uniform. When beginning from a constant solution, a second-order time-accurate computation is recommended following a first-order calculation to enhance the precision of results.

Additionally, the time step selection should match the expected frequency of the phenomena being modeled. Approximately 100 steps per period are sufficient to fully resolve the cycles of interest. By adhering to these best practices, engineers can ensure their marine simulations reflect real-world dynamics, thereby enhancing the reliability of their analyses.

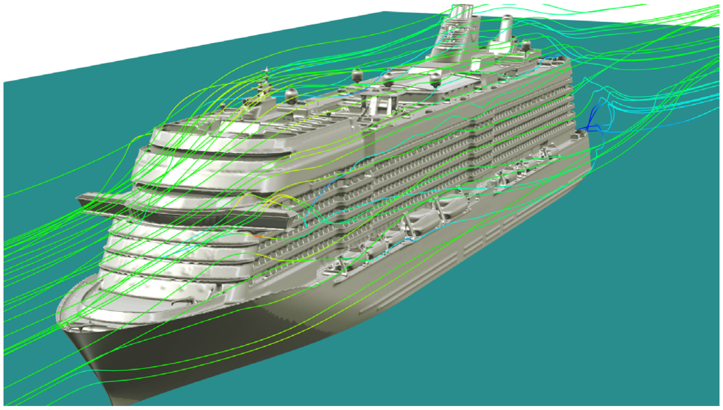


*Figure 16: Streamlines demonstrating the airflow around a cruise ship during constant wind conditions*

# Fluid Model

There are three fluid models to choose from in Fidelity Fine Marine: mono-fluid, multi-fluid, and passive scalar. The user must select at least one fluid whose corresponding properties can be defined using the default fluid properties or by creating new fluids with specific parameters. Presently, Fidelity Fine Marine supports Newtonian liquids with uniform density.

## Mono-Fluid

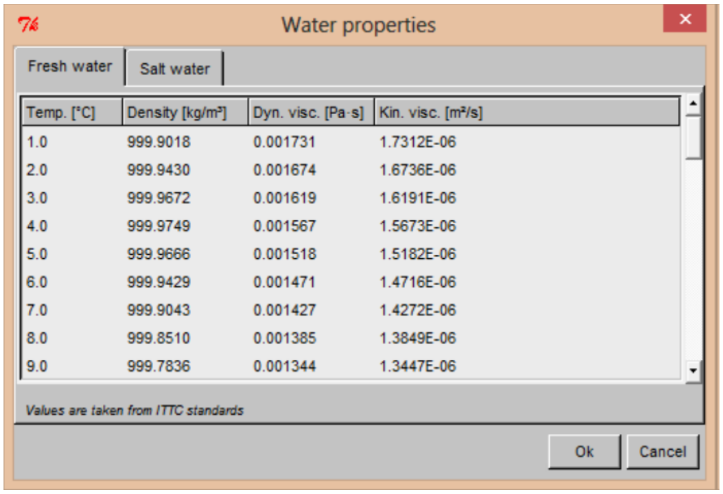
A single fluid is modeled in mono-fluid projects. Users can define the fluid type as freshwater or saltwater and edit fluid properties such as temperature, density, dynamic viscosity, and kinematic viscosity. These parameters are selected from a provided database (Figure 17) whose values are derived from the International Towing Tank Conference (ITTC) standards.

## Multi-Fluid

Multi-fluid projects involve the interaction between two or more fluids. When activating the multi-fluid model, free surface capabilities become available, enabling the simulation of interfaces between different fluids. The default fluid types for multi-fluid flow are saltwater and atmospheric air. In multi-fluid setups, the first fluid is always defined below the second fluid along the direction of gravity. The default free surface location value is set to zero [m] along the Z-axis, but this value can be modified. Users can also choose from a selection of mass fraction discretization schemes, including the Bounded, Robust, Implicit, Coupled Scheme (BRICS), which is the default numerical model. BRICS is used to solve the transport equations for mass fractions, particularly in multi-phase and reacting flow simulations.

## Passive Scalar

Both mono- and multi-fluid configurations are possible in passive scalar projects. A passive scalar in fluid dynamics is a quantity, such as temperature, concentration of a contaminant, or dye, that is carried by the fluid flow but does not influence the flow itself. It is characterized by its ability to diffuse within the fluid and be transported by the fluid's velocity field without affecting the fluid's motion or properties. In marine simulations, passive scalars are used to track how substances disperse and mix in the flow, providing insights into processes like pollution dispersion, heat transfer, or the spread of substances within a fluid system.




| Temp. [°C] | Density [kg/m³] | Dyn. visc. [Pa·s] | Kin. visc. [m²/s] |
|---|---|---|---|
| 1.0 | 999.9018 | 0.001731 | 1.7312E-06 |
| 2.0 | 999.9430 | 0.001674 | 1.6736E-06 |
| 3.0 | 999.9672 | 0.001619 | 1.6191E-06 |
| 4.0 | 999.9749 | 0.001567 | 1.5673E-06 |
| 5.0 | 999.9666 | 0.001518 | 1.5182E-06 |
| 6.0 | 999.9429 | 0.001471 | 1.4716E-06 |
| 7.0 | 999.9043 | 0.001427 | 1.4272E-06 |
| 8.0 | 999.8510 | 0.001385 | 1.3849E-06 |
| 9.0 | 999.7836 | 0.001344 | 1.3447E-06 |



*Figure 17: Fidelity Fine Marine GUI displaying the physical properties of freshwater and saltwater at various temperatures*

# Flow Model

After choosing the fluid model, several factors, including the nature of the flow (e.g., laminar or turbulent), the intensity of gravity, and dimensionless numbers, such as Reynolds and Froude numbers, play a major role in the flow model selection.

## Mathematical Model

Fidelity Fine Marine offers various laminar and turbulent models. Laminar flow is characterized by smooth, orderly fluid motion where layers of fluid slide past one another without mixing. This type of flow typically occurs at low velocities and in fluid with low Reynolds numbers. Laminar flow models are simpler and computationally less intensive, making them suitable for scenarios where the flow remains stable and predictable, such as in small-scale systems or fluids with high viscosity. In Fidelity Fine Marine, non-turbulent models include the laminar and Euler models. The laminar model considers the effects of viscosity, which is dependent on the fluid type (e.g., Newtonian or non-Newtonian). In marine applications, Newtonian fluids such as water and most oils are typically analyzed. The dynamic viscosity can vary for Newtonian fluids depending on the temperature or pressure, but it has a constant value with changing stress within the fluid. In contrast, the dynamic viscosity is set to zero in the Euler model, and all solid boundaries are set to a slip wall condition. As a result, viscous effects are neglected, and the fluid slides along the surface without friction.

Turbulent flow is chaotic and characterized by eddies, vortices, and apparent randomness. This type of flow occurs at high velocities and high Reynolds numbers. Turbulent flow models are more complex and computationally demanding. Advanced techniques like Reynolds-Averaged Navier-Stokes (RANS) equations, Large Eddy Simulation (LES), or Direct Numerical Simulation (DNS) are required to predict turbulent behavior. As displayed in Figure 18, the primary types of turbulence models available in Fidelity Fine Marine are one-equation models (e.g., Spalart-Allmaras), two-equation models (e.g., $k$-$\varepsilon$ and $k$-$\omega$), Explicit Reynolds Stress Models (e.g., EASM, EASM-BSL, EASM-SBSL, and EASM-SSC), and Detached Eddy Simulation models (e.g., DES, DDES, and IDDES).

For basic hydrodynamic simulations, the $k$-$\omega$ (SST-Menter) model is the standard and recommended option. Alternatively, the EASM turbulence model offers slightly improved outcomes with moderately higher CPU usage. It is particularly useful for vessels with a high block coefficient ($C_b$) that produce large longitudinal vortices from the hull sides. However, caution is advised in situations of extensive flow detachment, which is common with flat wet transoms. The transom is the section at the back of a boat that seals off the hull to prevent water from entering and provides structural support. EASM tends to predict higher turbulence levels in flat wet transom simulations. This is not an issue with dry transoms, as air effects are minimal in multi-fluid simulations, though they become significant in wind studies.

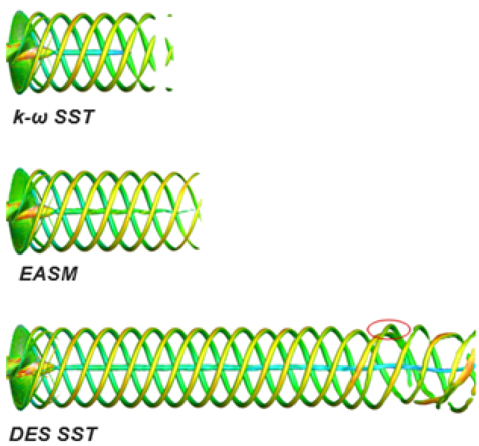



*Figure 18: The DES SST turbulence model offers the most detailed depiction of the propeller tip vortex compared to the EASM and $k$-$\omega$ SST turbulence models*

### Gravity Intensity

The intensity of gravity is another essential factor in flow modeling, especially in applications involving buoyancy-driven or free-surface flows, such as in marine simulations. Gravity influences the pressure distribution within the fluid and affects the flow patterns. It is usually modeled as a body force acting on the fluid. However, the gravity intensity is set to zero in mono-fluid computations in Fidelity Fine Marine when gravitational effects are insignificant in the flow behavior being studied. This simplification is typically applied when the primary forces driving the fluid motion are inertial, pressure, or viscous forces rather than buoyancy or other gravity-related effects. By neglecting gravity, the computational model becomes less complex, reducing the resources required and allowing the user to focus on the dominant flow characteristics.

### Reynolds and Froude Numbers

The Reynolds number is a dimensionless quantity that helps predict flow patterns in different fluid flow situations. It is defined as the ratio of inertial forces to viscous forces. A low Reynolds number indicates laminar flow, while a high Reynolds number suggests turbulent flow. The Reynolds number is key in determining the appropriate flow model and ensuring precise simulation results. Moreover, the Froude number is another dimensionless parameter that compares the flow inertia to the gravitational forces. It is particularly relevant in free-surface flows. The Froude number helps assess the wave-making resistance and the behavior of waves around bodies moving through a fluid. It is essential in naval architecture and hydrodynamics, where predicting wave patterns and resistance is crucial. Reference parameters such as the reference length and velocity are required to compute the Reynolds and Froude numbers.

## Body Motion

Coordinate frames are important for simulating the motion and interactions of vessels with their environment. Fidelity Fine Marine utilizes global, body, and user-defined reference frames. The global frame serves as a fixed reference point and is typically aligned with the Earth’s surface or a stationary reference. It is used to describe the overall motion of the fluid(s) and vessel. Cardan angles, also known as Euler angles, are used to describe the orientation of the vessel relative to the global frame. These angles define the vessel’s rotation about its principal axes and are critical for managing the complex, multi-axis movements typical in marine environments. The body frame is attached to the vessel itself, moving and rotating with it, which is essential for analyzing forces and dynamics from the vessel’s perspective. User-defined frames offer flexibility by allowing custom reference points tailored to specific parts of the simulation, such as rotating machinery or localized phenomena. As shown in Figure 19, ship motion studies analyze the behavior of ships across six degrees of freedom:

- **Surge:** Forward and backward movement along the ship’s longitudinal axis.
- **Sway:** Side-to-side movement perpendicular to the longitudinal axis.
- **Heave:** Vertical movement.
- **Roll:** Rotation about the longitudinal axis.
- **Pitch:** Rotation about the transverse axis.
- **Yaw:** Rotation about the vertical axis.

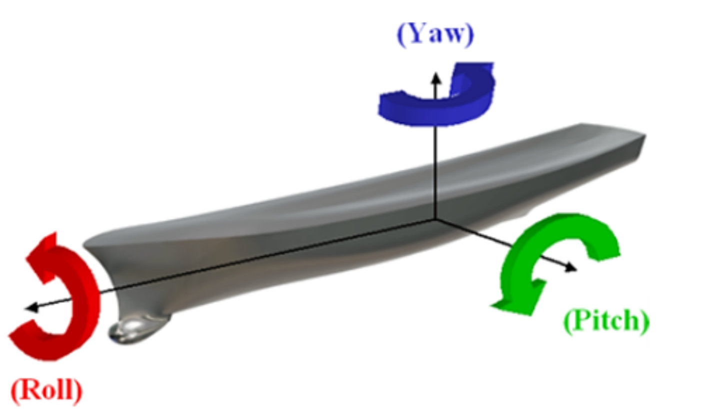



*Figure 19: Cardan angles represented on the David Taylor Model Basin (DTMB) geometry*

# Additional Models

After preparing the physical configuration, additional models are available in Fidelity Fine Marine for specialized analyses. Below are brief descriptions of some commonly used models.

- Actuator Disk: The actuator disk model (Figure 20) simplifies the representation of propellers or rotors by simulating them as permeable disks that impart a pressure jump or momentum change to the fluid passing through them. This approach allows for the efficient simulation of the thrust and drag effects of propulsion systems without needing to model individual blades.

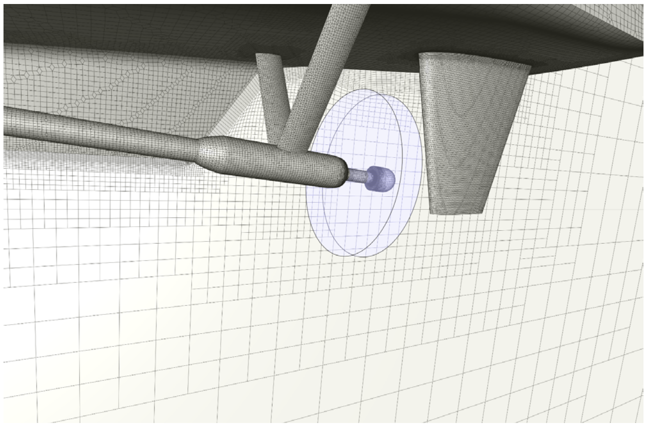


*Figure 20: Ship geometry grid featuring an actuator disk*

- Cavitation: The Merkle, Sauer, and Kunzsimulate cavitation models are used to simulate the formation and collapse of vapor bubbles in a liquid flow due to pressure fluctuations. This phenomenon is significant in marine propellers, pumps, and hydrofoils as it can cause damage, noise, and performance degradation.
- Temperature: The effect of temperature on the fluid flow can be modeled using the Boussinesq approximation, which simplifies the equations of motion by assuming that density variations are negligible except in the buoyancy term, thus capturing natural convection phenomena. Additionally, a solute model can be employed to account for variations in solute concentration, which can affect fluid density and flow. These methods can be used to simulate fume exhausts.
- Non-Deterministic Data: Non-deterministic data models incorporate uncertainties and variations into simulations to reflect real-world conditions where exact values are not always known. These models use statistical methods to analyze the impact of variability on the simulation outcomes, thereby enhancing reliability.
- Modal Approach: The modal approach is used to simulate fluid-structure interaction (FSI) by decomposing the structural response into a series of mode shapes and eigenfrequencies. This method is efficient for analyzing vibrations and structural dynamics, particularly in cases where FSI is significant, such as on ship hulls, propellers, rudders, hydrofoils, underwater vehicles, and offshore platforms.
- Laminar Transition Model: The shift from laminar to turbulent flow is important for certain applications operating at low or moderate Reynolds numbers, such as unmanned maritime vehicles, small submarines, or hydrofoils under specific conditions. Although the RANS modeling approach is commonly used owing to its lower computational demands, it does not capture the nuances of flow transition. Transition models, which typically employ the Local Correlation-based Transition Modeling (LCTM) strategy, can capture flow instabilities and the onset of turbulence.
- Internal Wave Generator: An internal wave generator simulates the generation and propagation of waves within the fluid domain. These models are essential for studying wave dynamics and interactions in coastal and offshore engineering. Waves are produced within the domain through a momentum source term added to the Navier-Stokes equations. This method is particularly useful for stationary objects such as platforms, where waves reflected off the object might reach the domain's upstream boundary. Placing the generator inside the domain rather than on a boundary and using it alongside sponge layers ensures realistic wave behavior.
- Wave Damping/Wave Forcing: Wave damping models use sponge layers to dampen or reduce the amplitude of free surface waves and avoid undesired reflections at the domain boundaries. The damping is automatically adjusted based on the scale of the problem to ensure efficient wave simulations of all types. Wave damping can be combined with wave forcing to enforce the wave signal and prevent artificial damping and reflections.

By incorporating these additional models, marine simulations can be applied to capture a wide range of physical phenomena. These models allow engineers to better understand and predict the behavior of complex systems in various applications.

# Analysis and Optimization

Fidelity Fine Marine serves as more than just a predictive tool; it also guides the iterative process of refining designs and enhancing performance. Let's take a detailed look at the importance of analysis and optimization processes in marine engineering simulations.

## Post-Simulation Analysis

Once a simulation is complete, the analysis phase begins. This involves data interpretation, where engineers examine the simulation outputs (Figure 21), such as free surface elevation, wetted area, wave elevation, streamlines in an upright orientation, forces by section, wake flow, and towing tank lines. Then key performance indicators, such as resistance, propulsion efficiency, and seakeeping qualities, are evaluated against design objectives. The simulation results may reveal areas of suboptimal design where flow separation, excessive drag, or instability occurs.

### Grid Convergence Study

The discretization error within a CFD simulation can be identified by analyzing spatial convergence. This approach requires conducting the simulation on at least three progressively refined grids, as depicted in Figure 22. As grids become finer, the spatial discretization errors asymptotically diminish, except for errors due to computer round-off. This method can estimate the error margin and uncertainty range in the results from a particular CFD project. When the results stabilize and show minimal variation (typically under 1-2%) with further refinement, the grid can be considered converged.

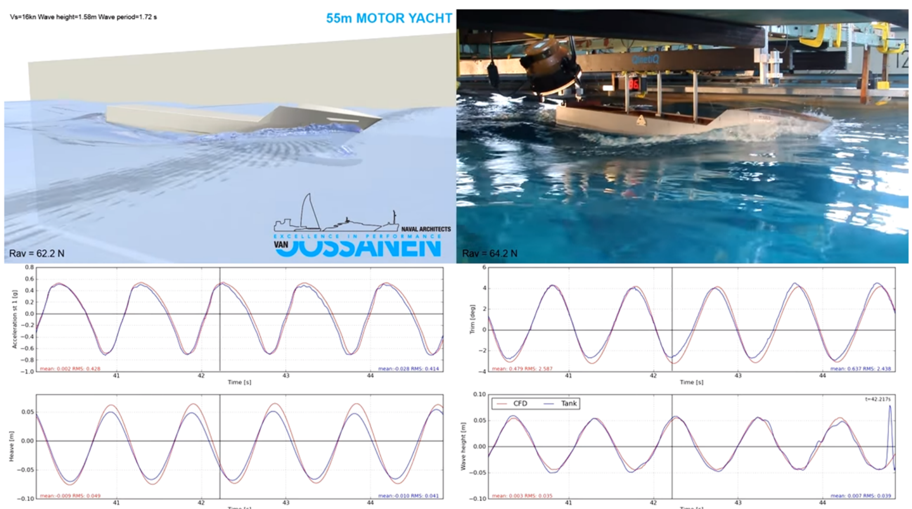



*Figure 21: Comparison of CFD and experimental towing tank measurements for the Van Oossanen motor yacht*

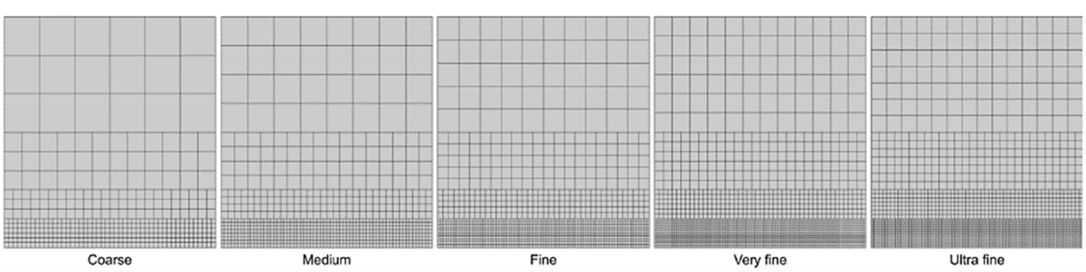



*Figure 22: Grids can be refined from coarse to ultra-fine to reduce spatial discretization errors*

# Optimization Strategies

The insights gained from the post-simulation analysis are used to optimize the design and operation of marine vessels. Based on the analysis, the design of the hull, propeller, or other components may be iterated to address inefficiencies or performance issues. Marine engineering often requires multi-objective optimization or balancing conflicting objectives, such as maximizing speed while minimizing fuel consumption and weight. Moreover, real-world conditions are never exactly as simulated, so designs must be robust and optimized under uncertainty. Below we define various optimization strategies from parametric studies to uncertainty quantification.

- Parametric Studies: Design parameters are varied within the simulation to explore a wide design space to find optimal configurations.
- Design of Experiments (DOE): A systematic approach used to plan, conduct, and analyze experiments to understand the relationships between multiple factors and their effects on outcomes.
- Pareto Frontiers: Utilized in decision-making to identify designs that offer the best trade-off between competing objectives.
- Sensitivity Analysis: Used to understand how sensitive a design is to changes in certain parameters, which helps prioritize the aspects of a design to focus on.
- Surrogate-Based Optimization: Surrogate models are approximations of an actual system that are often created using techniques like artificial neural networks, Kriging, or radial basis functions. They are used to predict the outcomes of different design configurations with relatively low computational costs.
- Robust Design Optimization (RDO): A technique that considers the variability in operating conditions or manufacturing tolerances to ensure the design performs well under a range of scenarios.
- Uncertainty Quantification: Systematically assess and reduce the impact of uncertainties in input parameters, model assumptions, and environmental factors on the predicted outcomes.

In the quest for superior maritime performance, optimization and innovation are paramount. Cadence stands at the forefront of this endeavor, offering advanced optimization capabilities to refine vessel designs to adapt to the ever-changing conditions of the sea, transport demands, and loading scenarios. These tools enable the exploration of novel design concepts and the fine-tuning of existing models to achieve optimal performance in hydrodynamics, stability, and operational efficiency.

### ML-Driven Optimization with Fidelity Fine Design3D

Fidelity Fine Design3D is a sophisticated CFD optimization software that streamlines the discovery of optimal designs through advanced machine learning and automated design space exploration. It excels in multidisciplinary optimization, addressing factors like off-design conditions and manufacturing costs with robust evaluators and solvers. The software's data mining algorithms offer deep insights into the design space, highlighting key variables and speeding up the optimization process. Key applications include uncertainty quantification, which ensures robust performance by assessing variability in manufacturing and operational conditions. It also features advanced design of experiment (DOE) techniques for exploring high-dimensional design spaces and surrogate modeling for rapid solution approximation using methods like artificial neural networks and Kriging.

# Real-World Case Studies

The following section presents multiple case studies illuminating the practical application of simulation-based design using Fidelity CFD platform tools. These real-world examples illustrate the challenges encountered in the marine field, the innovative solutions implemented, and promising results that highlight the value of simulation. From optimizing the hydrodynamics of a yacht to implementing cutting-edge propulsion systems, each case study offers a unique glimpse into the maritime industry and the critical role of simulation in advancing marine engineering.

## Finot-Conq Designs Award-Winning Yachts

Over the last five decades, Finot-Conq has been a leader in yacht design, producing over 45,000 boats, including production boats, cruising vessels, and racing prototypes, as shown in Figure 23. Notably, their designs have garnered more European Yacht of the Year awards in the past 15 years than those of any other competitor, highlighting their commitment to excellence and innovation in the industry.

A critical element of Finot-Conq's success is its use of advanced simulation tools provided by Cadence, a long-term partner. The Velocity Prediction Program (VPP) integrated within Cadence's Fidelity Fine Marine software has revolutionized their design process. Typically, simulating one second of sailing can require a full day's computation on modern servers. However, the combination of Fidelity Fine Marine and Finot-Conq's VPP has drastically reduced the necessary calculations to evaluate a yacht's design from 100–200 to fewer than ten. This efficiency has substantially accelerated the development timeline for new yacht models, allowing Finot-Conq to bring innovations to the market quicker than ever.


*Figure 23: Finot-Conq designs fast sailing yachts*

The increasing reliance on CFD to address design challenges led Finot-Conq to expand its computational capacity by purchasing over 400,000 CPU hours from Cadence OnCloud to complement its existing CFD capabilities. Finot-Conq also boosted its processing power by adding a dual-socket server equipped with AMD EPYC 7773X CPUs, effectively doubling its simulation capacity. This enhancement enabled it to conduct twice as many simulations per month, marking a major improvement in its design process.

> Thanks to doing our simulations on Fine Marine and AMD EPYC processors, we are able to take our designs to the next level.
>
> David de Premorel,
> CEO, Finot-Conq

# Damen Shipyards Improves Vessel Stability Through CFD Wind Studies

Damen, a global shipbuilding conglomerate, operates more than 50 shipyards across over 120 countries. Leveraging Fidelity Fine Marine and Cadence's cloud solutions, Damen has integrated marine simulations across the entire design workflow instead of limiting its use to the final stages. Historically, their expertise has been in bare hull resistance analyses. However, they are venturing into more intricate simulations, including propulsion, maneuvering, and wind studies.

Wind loads can impact ship navigation and stability, especially for vessels operating in challenging environments, such as the open sea and harbors, where wind effects can substantially influence handling and safety. Traditionally, Damen used wind tunnel testing to study these effects, but this approach was prohibitively expensive and time-consuming. To address these challenges, Damen Shipyards, in collaboration with Cadence, developed an application within Fidelity Fine Marine to replace costly experimental tests.

The company encountered a specific issue with their DAMEN Fast Crew Support (FCS) 3307 vessel (Figure 24): a slender design that was prone to stability problems when sustaining severe side winds. According to the International Maritime Organization (IMO) regulation 749.18, vessels must demonstrate sufficient transversal stability to resist over-rolling in severe side wind conditions. This regulation is part of the IMO's broader efforts to ensure safety and efficiency in international maritime shipping, which accounts for over 80 percent of global trade. Given the stringent regulation, slender vessels like the FCS3307 often struggle to meet this standard due to costly and extensive physical testing requirements.

To mitigate these costs, Damen Shipyards and Cadence developed a technique to accurately simulate wind effects on the vessel. This approach not only reduced the need for physical wind tunnel tests but also accelerated the design and validation process. The methodology was rigorously validated against actual wind tunnel test results for the DAMEN FCS3307, ensuring its reliability. This validation campaign demonstrated that Fidelity Fine Marine could be used to effectively replicate the physical phenomena observed in traditional tests, providing a cost-effective and efficient alternative.

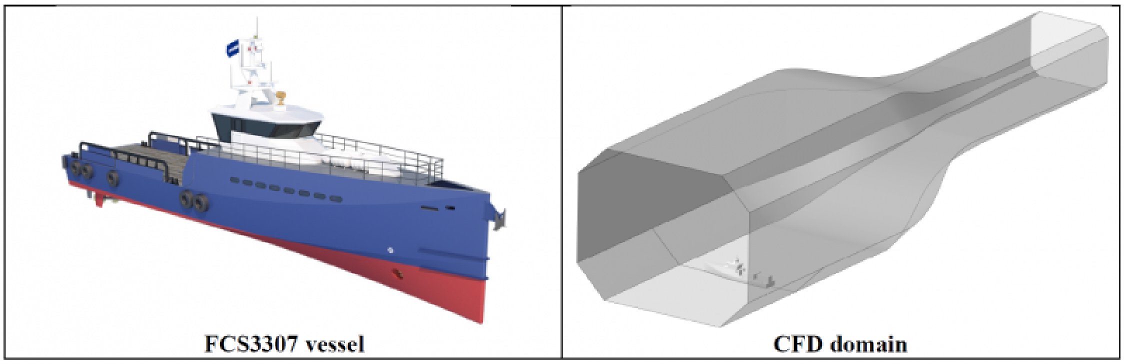



*Figure 24: The FCS3307 vessel is simulated within a computational wind tunnel domain to analyze severe side wind conditions*

# Seakeeping Analysis of SA Agulhas II

The marine industry is transforming as it shifts from traditional and costly towing tank tests to CFD adoption. This shift is exemplified in the seakeeping study of the SA Agulhas II, South Africa's icebreaking research vessel. The ship is owned by the South African Department of Environment, Forest, and Fisheries and was built by STX Finland in the Rauma shipyard in 2012. The SA Agulhas II (Figure 25), the successor to the SA Agulhas, measures 121.3 meters in length and 21.7 meters in width. It has advanced propulsion systems and accommodates 44 crew members and 100 passengers. The ship plays a critical role in supply and research missions in Antarctica, Gough Island, and Marion Island. Notably, it was involved in the discovery of the wreck of Shackelton's ship Endurance in early 2022. The ship's seakeeping properties were studied by the Sound and Vibration Research Group (SVRG) at Stellenbosch University using Fidelity Fine Marine.

The SVRG team used Fidelity CFD software to streamline their entire workflow, from geometry preparation to post-processing. They utilized Fidelity Automesh to create high-quality, unstructured hexahedral meshes with anisotropic cell refinement. This meshing capability, combined with the inflation technique, allowed for accurate boundary layer resolution, which was essential for effective turbulence modeling. Additionally, Fidelity Fine Marine's C-Wizard substantially reduced the engineering time required for setting up complete marine simulations. This automation provided a ready-to-go setup that included all necessary numerical parameters, thereby minimizing manual input and accelerating the simulation process.

The simulations conducted of the SA Agulhas II revealed detailed insights into the vessel's performance under design draft conditions, including visuals of bow spray at a speed of 12 km/h and a wetted surface area of 1671 m$^2$. Additionally, unsteady simulations demonstrated a wave period of 9 seconds and an encounter period of 6.3 seconds, with a wave height of approximately 1.6 meters. These findings highlighted regular wave responses and set the stage for further investigations into more complex conditions, such as irregular sea spectra and rough sea states.

Fidelity Fine Marine has proven to be an invaluable tool for both academic and commercial ventures in ship design. Its ability to handle complex simulations under strong wave conditions with minimal manual intervention showcases its potential to revolutionize seakeeping studies, enhancing safety, efficiency, and effectiveness in ship design and analysis.


*Figure 25: SAA II vessel during the 2015/2016 relief voyage to Antarctica*

# Conclusion

This comprehensive guide culminates with a reflection on the transformative impact of Fidelity Fine Marine on marine engineering. Throughout this journey, we have discussed the theoretical underpinnings of the field, the process of setting up simulations, and the detailed steps necessary to obtain reliable results.

We've learned the importance of defining objectives in the early design stages, where parameters like drag reduction, propulsion efficiency, and vessel stability are explored. Moreover, the importance of meticulous geometry preparation, precise meshing techniques, and the careful setting of boundaries and initial conditions have been highlighted to ensure the fidelity of simulations.

The application of Fidelity Fine Marine extends into the realm of post-processing, where it informs iterative design processes that push the envelope of innovation in vessel design and performance. Real-world case studies also illustrated the power of simulation in action, with Finot-Conq and Damen Shipyards leveraging Fidelity Fine Marine to cut costs and time while achieving superior design quality.

As we look to the future, it is evident that the role of simulation will only grow in the marine industry. It is not just a tool for validation but is pivotal in pioneering new advancements, whether in the pursuit of speed and agility in competitive sailing or in striving for sustainability and efficiency in commercial maritime operations.

This guide underscores a key message: embracing CFD simulation technologies is no longer optional but essential. Fidelity Fine Marine enables engineers to navigate the complexities of marine design and analysis with confidence. In an era where the maritime industry seeks greener horizons and smarter solutions, Fidelity Fine Marine is a beacon guiding the way forward.

## Want to Learn More?

Advance your marine engineering designs using Fidelity Fine Marine, which is meticulously engineered for intricate marine applications. With Cadence's robust tools, leverage precise simulations to enhance the design, analysis, and optimization of your maritime projects.

Request a Demo: For an in-depth exploration of how Fidelity Fine Marine can benefit your marine projects, request a personalized demonstration. Our team is ready to highlight the power and precision of this software, ensuring it aligns with your specific project goals. Get in touch today to see this technology in action and take the first step towards transforming your marine designs.