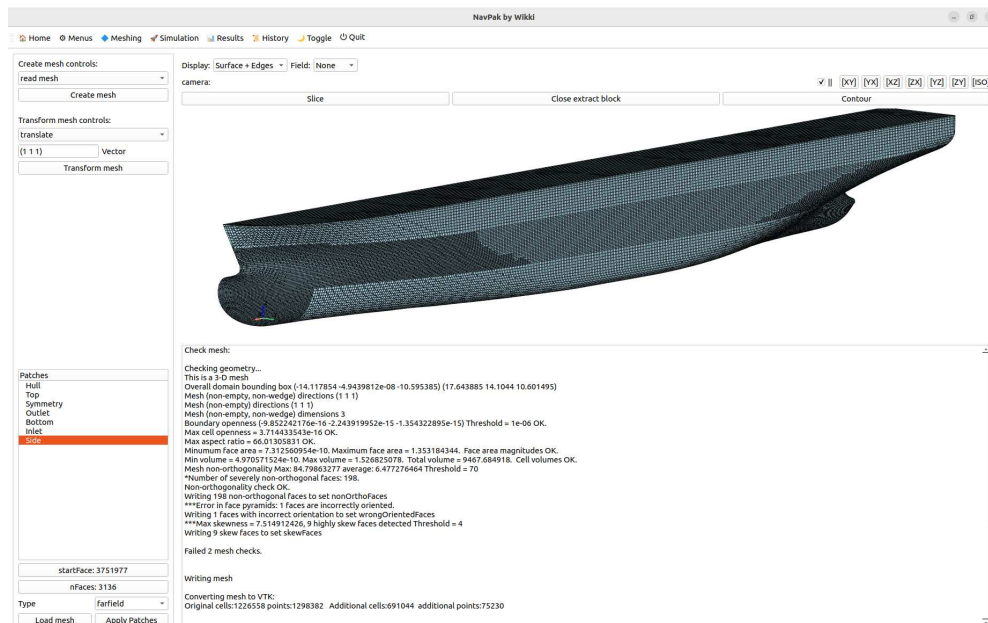


NavPak: Naval Hydrodynamics CFD Package

User Manual

Hrvoje Jasak, Marko Horvat
Wikki Ltd.

Version: NP/UG/09-26



Executive Summary

NavPak is a dedicated GUI toolbox for CFD simulations in marine hydrodynamics. It provides an integrated environment, preparing simulation cases, configuring physical and boundary conditions, running simulations, and analysing the results. Whenever possible, numerical settings are fully automated.

The toolbox supports self-propulsion, sea-keeping, and steady-resistance simulations. Built-in visualisation tools allow users to examine forces, residuals, flow fields, and the free surface.

It enables non-expert users to quickly set up high-accuracy simulations while still providing experienced users with control over the simulation parameters. It supports self-propulsion, sea-keeping, and steady-resistance simulations, with integrated tools for monitoring and visualising the results.

Contents

Contents	2
List of Figures	4
1 Introduction and Overview	5
1.1 What to Expect?	5
1.2 Mesh Import or Conversion	5
1.3 Automatic Interface	5
1.4 External Meshing Tools	5
2 Mesh Preparation and Import	6
2.1 Model Size and Orientation	6
2.2 Mesh Structure and Refinement	7
2.3 Actuator Disk	9
2.4 Mesh Deformation for Sinkage and Trim	9
2.5 Modelling Far Field Conditions	11
3 Types of Simulation	13
3.1 Modelling Approach: Ship, Fluid, Far-Field, Propeller	13
3.2 Steady Resistance Simulations	14
3.3 Sea-Keeping Simulations	14
3.4 Self-Propulsion Simulations	15
3.5 More Complex Cases	15
3.6 Manoeuvring Cases	16
4 Installation and Running	17
4.1 Preparing the Simulation Environment	17
4.2 Creating a Case Directory	17
5 Starting the NavPak GUI	18
6 Configuring the Simulation	19
6.1 Configuring Physical Properties	19
6.2 Mesh Functionality	20
7 Running the Solver	22
7.1 Field Initialisation	22
7.2 Simulation Controls	22
8 Results and Simulation Monitoring	23
8.1 Accessing Simulation Results	23
8.2 VTK Post-Processing	23

9	Advanced User Interface Functionality	25
9.1	Logs, Scripting and Parametric Studies	25
9.2	Saving and Exchanging Simulation Cases	25
9.3	Command-Line Dictionary Export	25
9.4	GUI Interface for Advanced Users	25
10	In Case of Problems	26

List of Figures

2.1	Ship hull orientation in space: x-axis in motion direction, y-axis to starboard, z-axis up.	6
2.2	Domain size guidance.	7
2.3	Mesh structure and refinement.	8
2.4	Kelvin angle free surface refinement.	8
2.5	Polyhedral mesh.	9
2.6	Actuator disk patch.	10
2.7	Motion mask function m	10
2.8	Relaxation zone blending function w . Note change in scale.	11
3.1	Steady resistance simulation: convergence of force and motions.	14
3.2	Sea-keeping simulation: convergence of force and motions.	15
3.3	Self-propulsion simulation.	16
6.1	Menus tab.	19
6.2	Meshing tab.	20
7.1	Simulation tab.	22
8.1	Results tab.	23

1 Introduction and Overview

Naval hydrodynamics CFD is a demanding simulation environment, with many geometric, flow and numerics parameters. The objective of **NavPak** is to encapsulate the professional knowledge and experience of expert practitioners into a ready-to-use rapid and accurate simulation environment for practical naval hydrodynamics use.

1.1 What to Expect?

A regular user should expect to set up the simulation in minutes, follow the GUI-guided work-flow and generate robust and accurate results without needing multiple attempts or changing opaque discretisation and numerics settings.

Simulation software has been validated in public benchmarks over the last 15 years. The GUI encapsulates the experience and best-practice guidelines in a simple-to-understand manner.

The software is designed for naval architects: input data is defined in terms of ship, flow, far-field and propeller parameters. It can deal with model-scale and full-scale simulations, internally adjusting the parameters as needed. Forces, motions and residuals are monitored interactively during the simulation.

1.2 Mesh Import or Conversion

NavPak will use ship hull meshes prepared in external meshing software, following detailed orientation guidelines given below. In order to allow rapid simulation setup, the user is encouraged to comply with hull orientation and domain size recommendations. Doing so will produce stress-free and error-resistant setup.

1.3 Automatic Interface

In most cases, the user is expected to rely on the automatic interface. When necessary, the GUI may expose the full set of fluid, equation sets, discretisation, linear solver and 6-DOF parameters. This, however, is intended for experts. When such interventions are needed, please contact Wikki's specialists for further guidance (under your support contract).

1.4 External Meshing Tools

In this release, **NavPak** does not provide integrated geometry analysis, clean-up or meshing tools. The user is expected to provide meshes with correct orientation and adequate resolution, which are then imported into the GUI.

2 Mesh Preparation and Import

NavPak is designed for simulation of single ships under varying operating conditions. You are expected to use **external mesh generation tools** and provide the mesh either in Star-CCM+ or OpenFOAM format.

2.1 Model Size and Orientation

Mesh hull shall be oriented to point in the positive x-direction, with the y-axis pointing to port and z-axis pointing upwards, Fig. 2.1

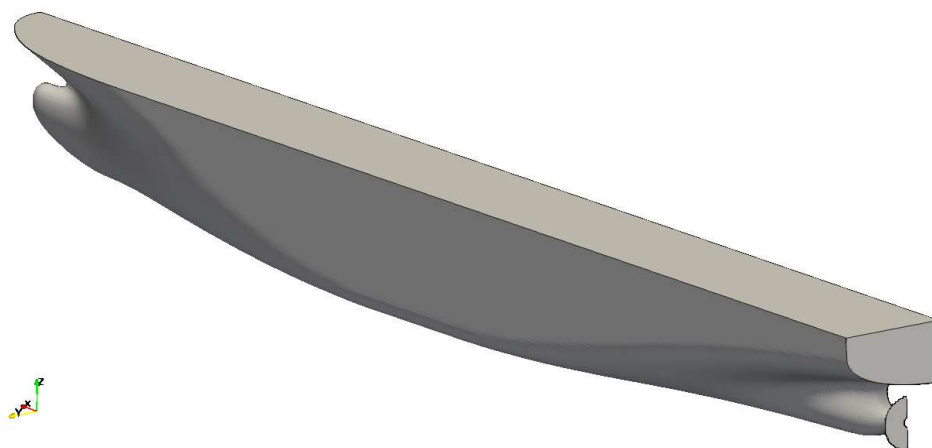


Figure 2.1: Ship hull orientation in space: x-axis in motion direction, y-axis to starboard, z-axis up.

The model can refer to half-ship, with a symmetry plane at $y = 0$ when case setup allows for it. In cases of oblique waves, free sailing with a single propeller *etc.* full ship hull should be meshed.

Important: In the z-direction, the hull should be located such that the plane $z = 0$ represents the reference free surface. For simulations with varying draft, please adjust the mesh z-coordinate as needed.

Important: In the x-direction, the hull should be located such that the vertical perpendicular is at $x = 0$, with positive x coordinate pointing in the direction of ship motion.

Guidance for the size of the computational domain around the ship is given in Fig. 2.2. This is the minimal size, governed by the far-field relaxation zones. All sizes are defined in terms of ship hull Length Between Perpendiculars, LPP .

As a guidance, aim to provide a far-field mesh of at least $2 LPP$ in front of the ship, 3 to 4 LPP behind and 2 LPP sideways. In the vertical direction, 1.5 to 2 LPP of height and depth should be provided.

In simulations with long waves, distances need to be scaled by the longest wave-length λ in the simulation, assuming it is larger than LPP .

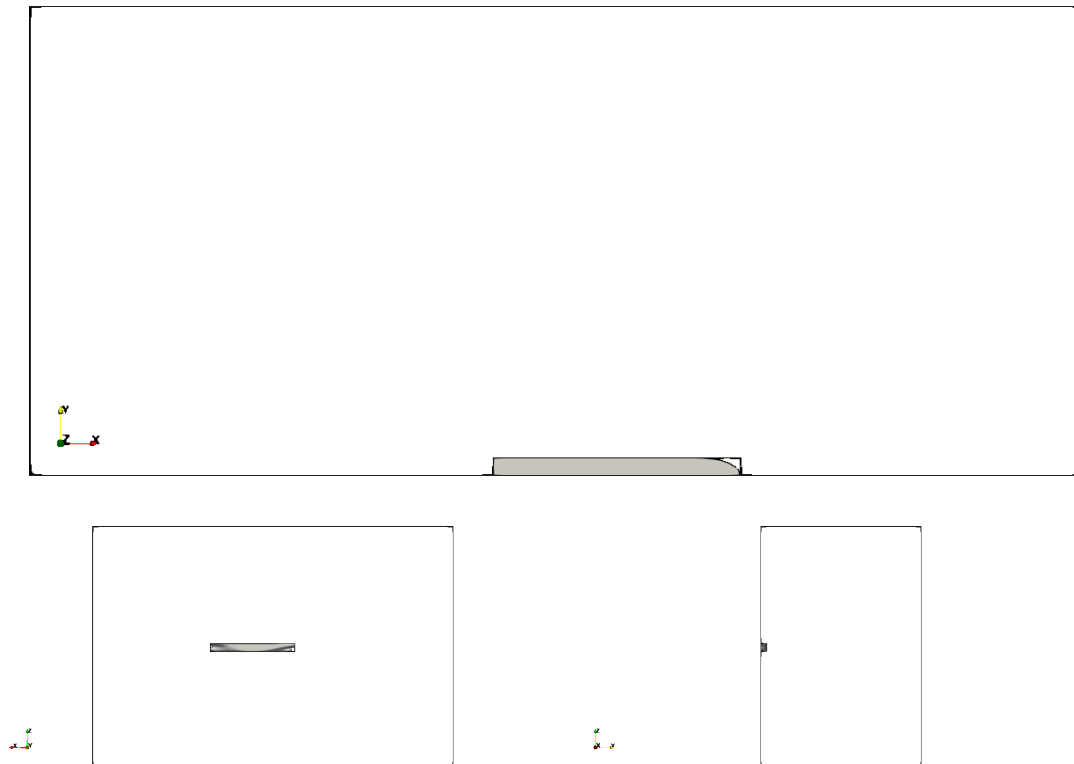


Figure 2.2: Domain size guidance.

2.2 Mesh Structure and Refinement

Far-field mesh is expected to be significantly coarser than the near-field mesh, with mesh resolution concentrated around the hull, Fig. 2.3.

Ideally, a hull mesh shall consist of a number of prismatic layers that resolve the viscous boundary layer. 6 to 8 layers are required for good accuracy. Details of hull mesh resolution are left to the user, depending on the desired accuracy and details of the flow next to the hull. Refinement regions can be used at will.

For good accuracy, sufficient vertical mesh resolution is required at the free surface. In steady resistance or self-propulsion simulations 25 to 30 cells across the wave height are ideal.

In sea-keeping simulation, good mesh resolution consists of 30 cells in the vertical peak-to-trough direction with 100 to 200 cells per wave length. Depending on type of simulation, mesh resolution and available computing power, this can be adjusted.

For good representation of waves, it is recommended to include a Kelvin angle refinement region at the free surface, Fig. 2.4.

Note that the superior resolution of wave elevation in the Kelvin angle carries little or no improvement in accuracy in dynamic pressure forces on the hull; it does however generate good wave elevation solution away from the hull.

NavPak works with arbitrary polyhedral meshes, including 2-1 refinement or other unstructured features. An example of a good polyhedral-native mesh is shown in Fig. 2.5.

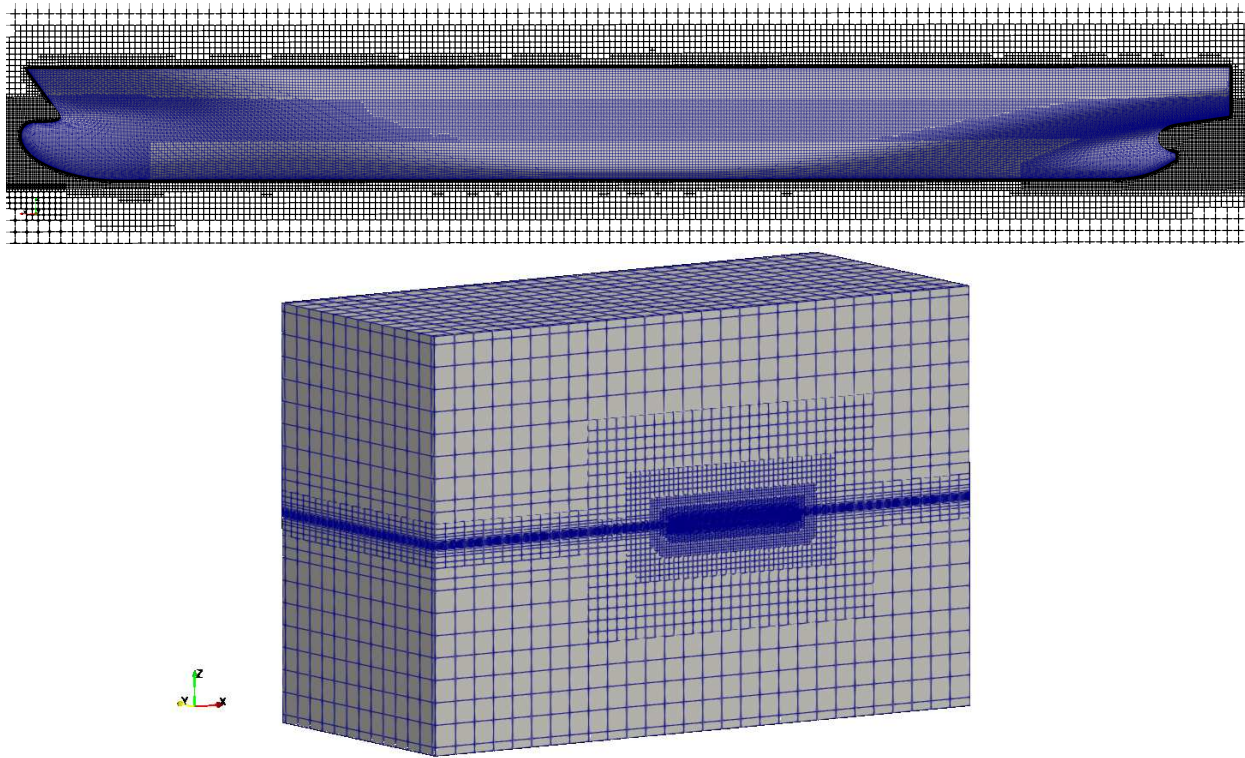


Figure 2.3: Mesh structure and refinement.

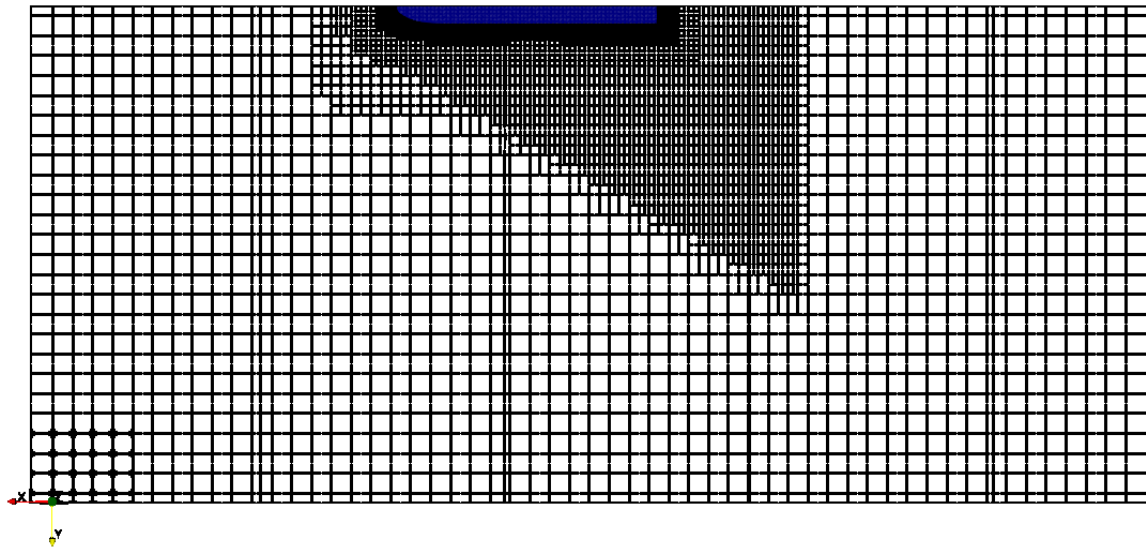


Figure 2.4: Kelvin angle free surface refinement.

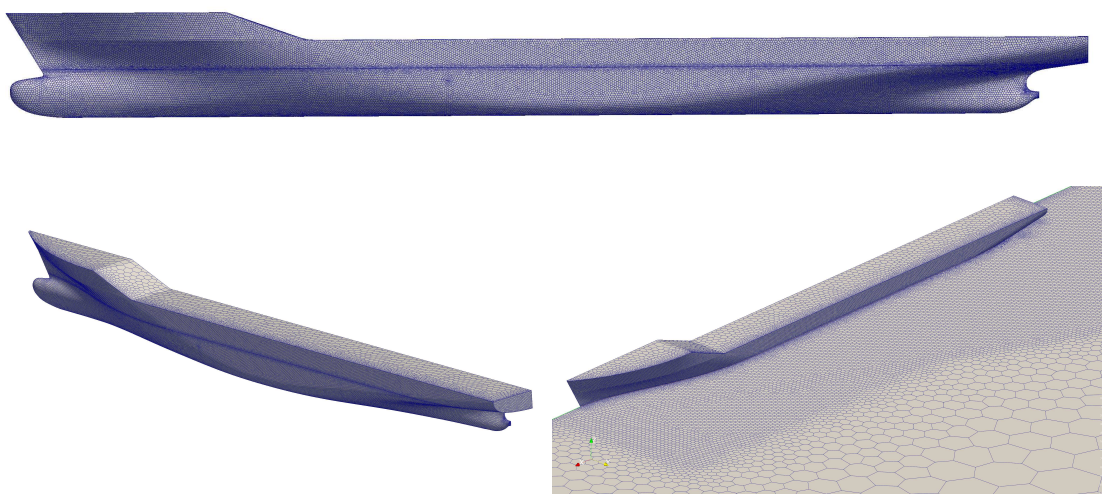


Figure 2.5: Polyhedral mesh.

2.3 Actuator Disk

The most efficient way of modelling self-propulsion uses the actuator disk approach. Here, a matching cyclic patch is located at the centre-plane of the propeller and it generates thrust and torque of propeller action without the need to resolve its geometric features. Actuator disk setup is shown in Fig. 2.6.

Actuator disk consists of two sets of matching faces, with the first set appearing upstream. Thrust model will create the pressure jump, positive from upstream half to downstream half of the patch. Thrust and torque are modelled based on $K - T/K - Q$ curves, accounting for non-uniform inflow conditions.

Actuator disk plane should point in the direction of thrust. Its external diameter should be equal to the propeller diameter, with an inner hole equal to the hub diameter.

Actuator disk model can be used both in symmetry plane mesh configurations and full ship hull meshes. Note that a symmetry plane actuator disk implies no torque in the flow, as this would break flow symmetry implied by the mesh.

Minor kinks or external edge irregularity in the actuator disk patch – such as quad cells describing a circle – do not result in loss of accuracy.

2.4 Mesh Deformation for Sinkage and Trim

In order to accommodate minor hull motion in *e.g.* steady resistance simulations, **NavPak** uses algebraic mesh deformation. Here, motion of the hull is calculated from 6-DOF motion equations and the mesh needs to be adjusted accordingly.

In order to preserve the alignment of the far-field mesh with the free surface, algebraic mesh deformation in **NavPak** uses a blending function for the motion. This function m , varying from 0 to 1 indicates which parts of the mesh move with the hull ($m = 1$) and which remain stationary in the far-field ($m = 0$). Between the two, motion mask function smoothly blends between 0 and 1 to preserve mesh quality. Function m is shown in Fig. 2.7

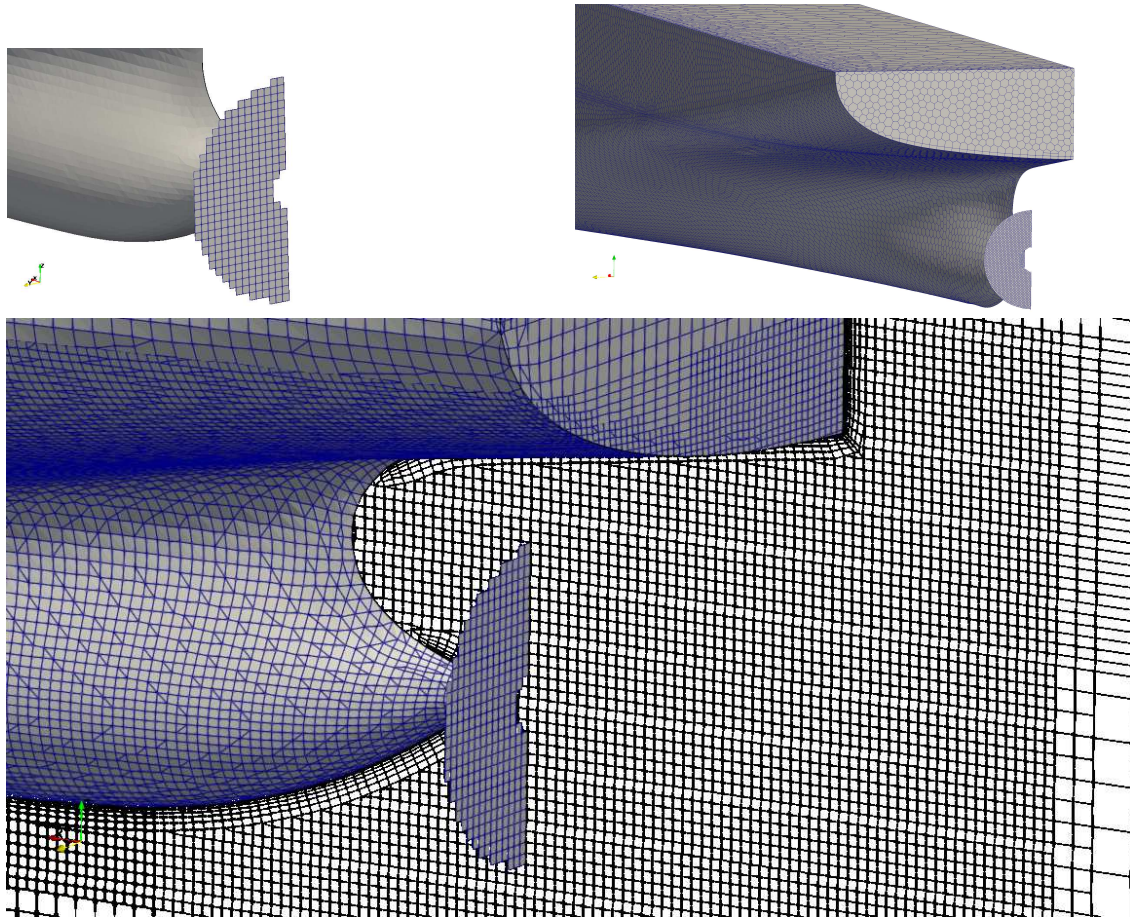


Figure 2.6: Actuator disk patch.

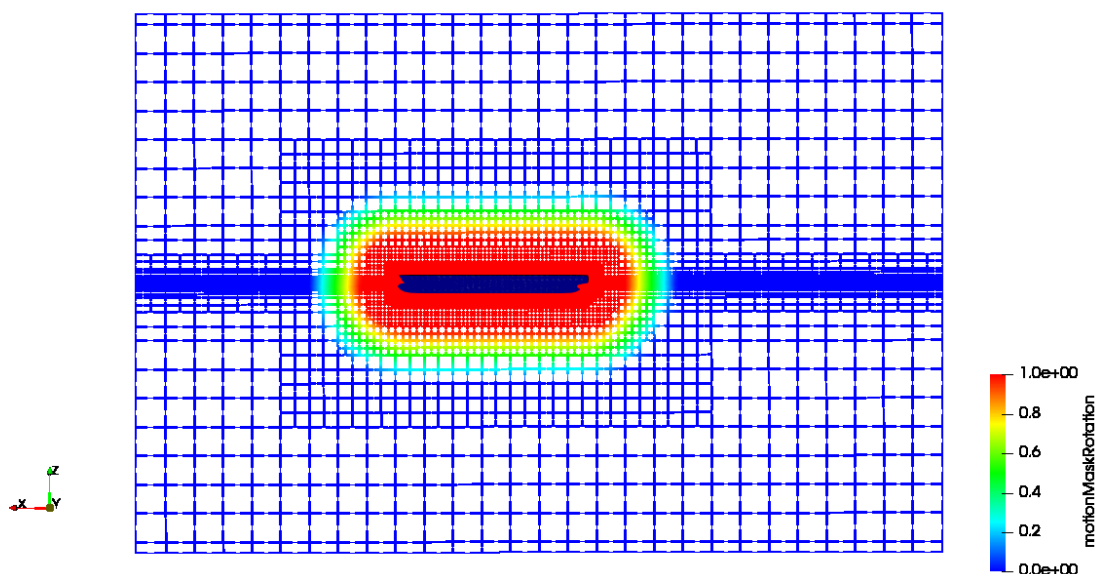


Figure 2.7: Motion mask function m .

2.5 Modelling Far Field Conditions

NavPak models far field conditions (calm seas, regular or irregular waves) using the implicit relaxation zone approach. In the far field of the domain, incoming and outgoing wave conditions are imposed on the solution and blended with the CFD solution in the near field. This approach guarantees no wave reflection problems and highly accurate introduction of incident waves onto the hull.

Far-field conditions are imposed via the weight function where $w = 1$ indicates that the flow conditions are imposed and $w = 0$ indicate fully non-linear Navier-Stokes solution via CFD. Function w is smoothly blended into the domain, with w close to unity in large cells away from the hull. This approach guarantees that even complex sea states defined by spectra and imposed via potential flow solution such as regular Stokes waves, stream functions or Higher Order Spectrum (HOS) non-linear potential flow solver is correctly imposed onto the CFD domain.

Fig. 2.8 shows the blending function w on the domain.

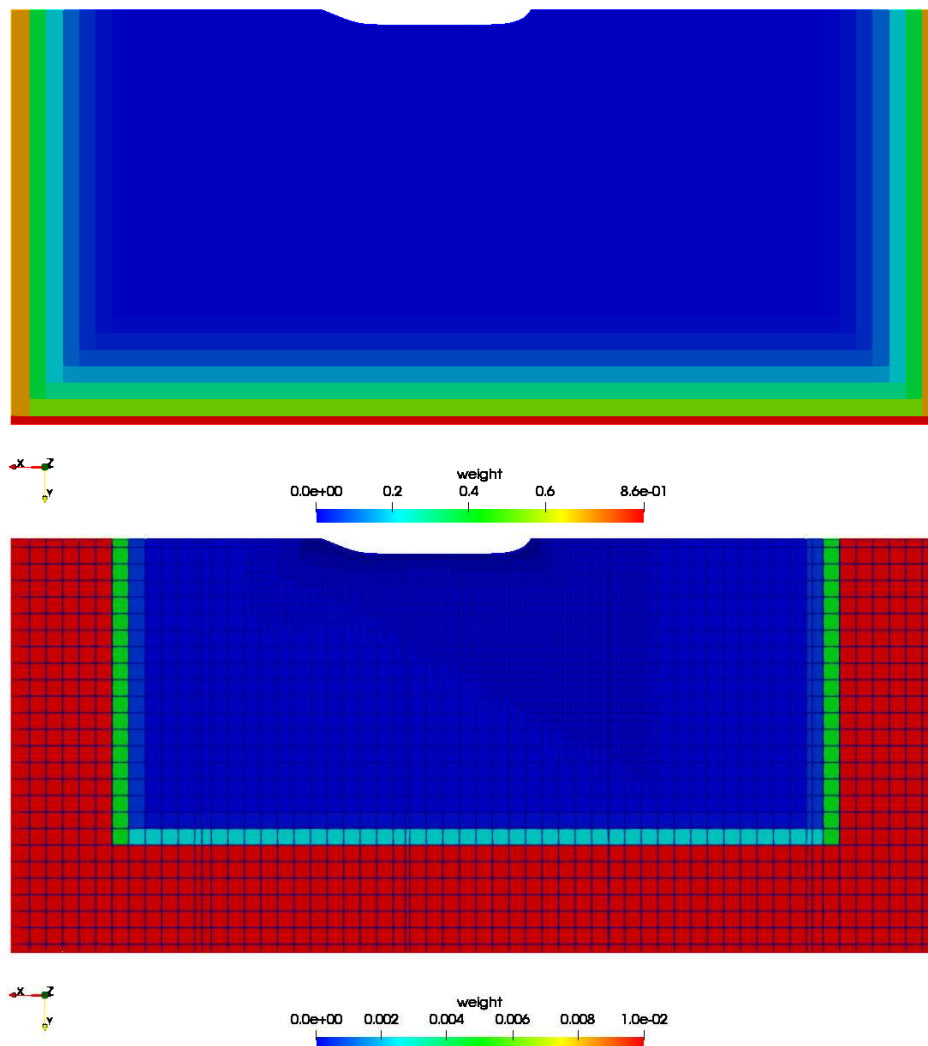


Figure 2.8: Relaxation zone blending function w . Note change in scale.

While w varies between 0 and 1, even a small value such as $w = 0.01$ indicates strong

influence on the CFD solution. For this reason, the domain size and length of far-field relaxation zones should be such that $w < 0.01$ at a sufficient distance from the hull, Fig. 2.8.

3 Types of Simulation

While **NavPak** implements substantial capabilities, from basic simulation types to complex free sailing and manoeuvring in regular and irregular sea states, the GUI concentrates on the set of well-defined and well-controlled simulation classes. More complex setup can be easily achieved with combinations of these, but typically requires more experienced users.

The case templates cover most common simulation type naval hydrodynamics:

- **Steady Resistance**,
- **Sea-keeping**, and
- **Self-Propulsion**,

The default configurations and tutorials are based on the Yipeng benchmark cases. Meshes are provided for reference, in standard configuration. Consequently, the physical parameters can be used without modification when reproducing the benchmark case.

3.1 Modelling Approach: Ship, Fluid, Far-Field, Propeller

In all simulation cases, some aspects of the setup remain the same. This is captured by individual panels of the **NavPak** interface.

The **Ship Panel** contains properties of the hull: LPP, hull mass, moment of inertia, centre of gravity *etc.* Note that this needs to be adequately adjusted if half-hull is modelled with a symmetry plane. The ship panel also allows specification of constraints (motion is blocked) and restraints (motion is damped) in 3 translation and 3 rotation motions.

Under the orientation conditions shown above, meaning of translations and rotations is given in Table 1 as follows.

Motion Type	x-axis	y-axis	z-axis
Translation	Surge	Sway	Heave
Rotation	Roll	Pitch	Yaw

Table 1: Definition of ship hull motions in reference frame.

The **Fluid Panel** contains density and viscosity of the water and air.

The **Far-Field Panel** contains the properties of the far field, be it for calm seas or incident waves. Parameters include ship speed, turbulence properties of the incoming flow and various wave-related parameters such as wave type, direction, height (peak-to-trough) and wave period; air and water current (assuming stationary ship hull in simulation reference frame) and similar.

The **Propeller Panel** contains the properties of the propeller, such as its location, hub and propeller diameter, rotation rate, left- or right-handedness, half patch and $K - t$ and $K - Q$ curves.

Combination of the above allows the user to quickly and intuitively set up all simulation types.

3.2 Steady Resistance Simulations

Steady resistance simulations refer to standard model tests for a ship hull form under towed conditions in calm seas as zero degrees heading. They can be performed in symmetry plane conditions.

Simulation settings allow two modes:

1. Fixed hull run, where the hull is not allowed to move;
2. Free sinkage and trim, where the hull reaches equilibrium in sinkage (buoyancy matches ship weight and pitching moment equals zero).

In the first case, the mesh remains fixed, while in the second it deforms using the motion mask strategy.

Parameters for steady resistance simulations include ship properties and fluid properties. Ship speed is defined in the far field properties.

Simulation results are expressed in terms of ship hull forces (pressure and viscous), hull sinkage and trim. While ship resistance simulations are typically performed in model scale, full scale ship resistance data may carry useful information but it is not directly comparable with sea trials results.

In steady resistance simulations, release of hull motion is automatically delayed to avoid start-up-related oscillations. Further, sinkage and trim damping is applied by default: this stabilises natural (transient) 6-DOF solid body oscillations and improves the convergence of sinkage and trim to steady-state.

An example result of the steady resistance simulation is shown in Fig. 3.1 for the Yipeng hull tutorial.

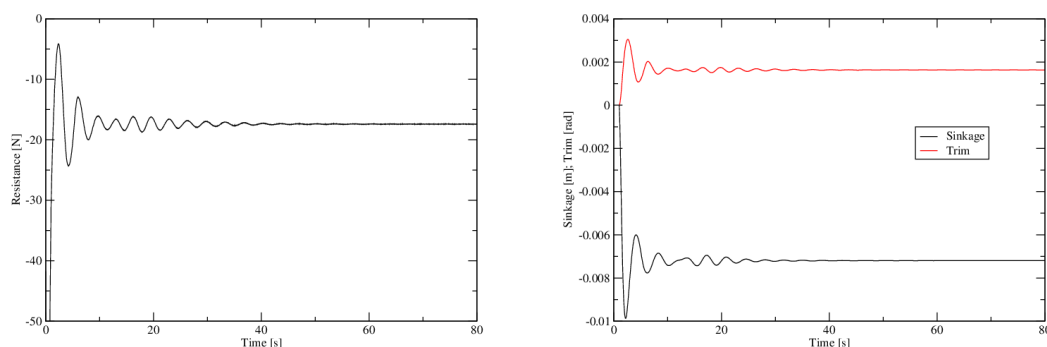


Figure 3.1: Steady resistance simulation: convergence of force and motions.

3.3 Sea-Keeping Simulations

Sea-keeping setup builds on the steady resistance simulations: a hull is towed at constant speed, typically in regular head waves. Here, far field properties involve wave type, height, period and direction. Flow current is added in water and air, depending on type of potential flow wave theory imposed in far field.

Result of sea-keeping simulations is considered in terms of force and motion transfer functions. Unlike steady-resistance, where steady-state is sought, this is a truly transient simulation, looking to capture the force and motion response. In regular waves, a quasi-periodic solution is achieved.

An example result of the sea-keeping simulation is shown in Fig. 3.2 for the Yipeng hull tutorial.

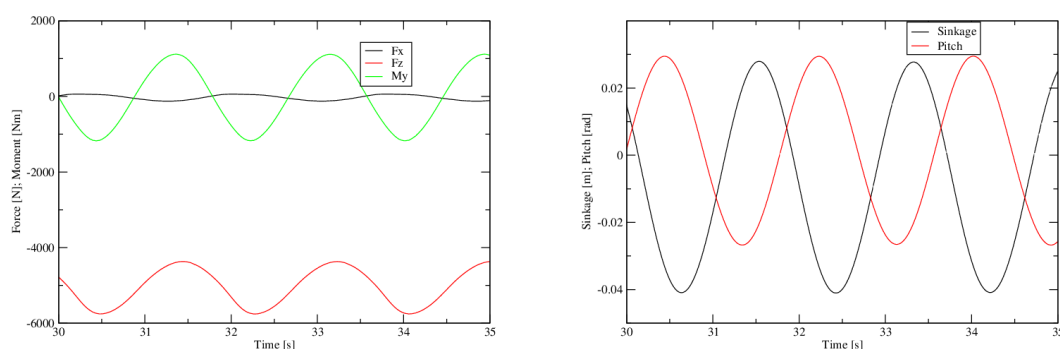


Figure 3.2: Sea-keeping simulation: convergence of force and motions.

3.4 Self-Propulsion Simulations

The major deficiency of the first two types of simulations presented above is that the hull is in towed conditions at prescribed speed and heading. In short, they correspond to towing tank experiments. For full-scale sea trials, such simulations are inadequate, as they do not account for hull-propeller-rudder interaction and the fact that the propeller provides a given thrust to the hull, while the speed is a derived quantity.

Starting from steady resistance, self-propulsion simulations add the action of the propulsor and seek to achieve balance between hull resistance and propeller thrust. Self-propulsion is directly comparable to sea trials measurements, either seeking to establish shaft power at a given speed or ship speed at a given propeller rpm.

Additional data is handled by the Propeller Panel; in terms of further complications, propeller controllers are now needed. They operate in two modes: either keeping propeller speed constant; or adjusting propeller rpm to achieve desired ship speed.

An example result for a self-propulsion simulation is shown in Fig. 3.3, with force balance converging to zero. Note that the propeller is activated with delay. The point where force balance converges to zero is termed the “free-sailing point”.

3.5 More Complex Cases

With the tools described above, it is possible to achieve further simulation types. A direct extension is self-propulsion in waves; an alternative is a free sailing simulation, where a hull with constant propeller rpm advances in waves and the result consists of time-varying resistance and ship speed in waves.

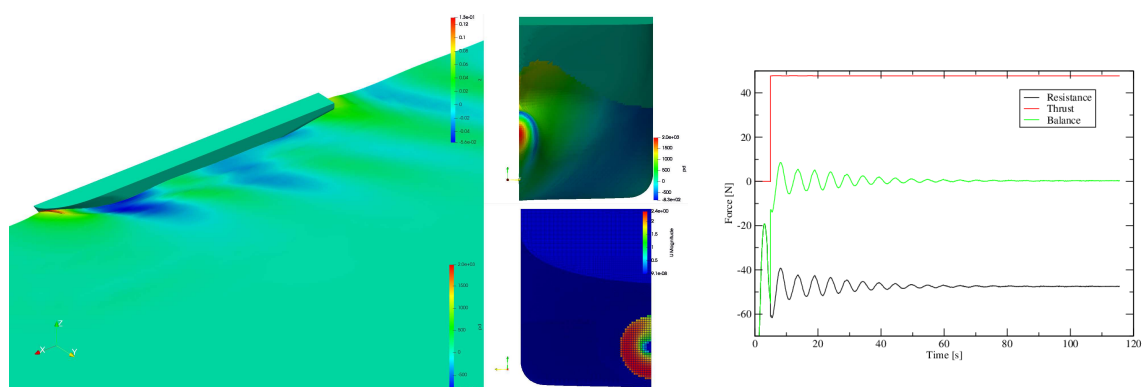


Figure 3.3: Self-propulsion simulation.

3.6 Manoeuvring Cases

NavPak is equipped with Overset Mesh capabilities, for simulations involving moving rudders and course changes. For further details, please contact Wikki.

4 Installation and Running

NavPak is provided as a self-contained installation in a single file. For purposes of demonstration, a set of ready-to-run meshes is also provided.

4.1 Preparing the Simulation Environment

This manual describes the complete workflow for setting up and running a hydrodynamic simulations using the **NavPak** GUI application. The application is distributed together with all the required libraries and example meshes.

Before starting the GUI, extract the two archive files supplied with the application:

- `meshes.zip`
- `navpak.zip`

The extracted `navpak.zip` directory contains the **NavPak** GUI executable and its associated distribution files. The extracted `meshes.zip` directory contains the meshes used to create the benchmark simulation cases that will be demonstrated in the manual.

4.2 Creating a Case Directory

Create a new directory that will serve as the working directory for the simulation. The directory name is arbitrary. For example: **my_case/** The simulation must be initialised and run from this working directory. Two meshes are provided with the application. The mesh located at **meshes/propeller_constant** contains an actuator-disk representation of the propeller and is intended for **self-propulsion** simulations. The mesh located at **meshes/hull_constant** does not contain a propeller and is intended for **seakeeping** and **steady-resistance** simulations. Copy the desired mesh into the newly created case directory. The mesh directory must be named **constant**.

Under normal operating conditions **NavPak** does not rely on files on disk, beyond the initial reading of the mesh. All solver controls are present and active in the GUI environment and communicated directly to the solver.

5 Starting the NavPak GUI

The GUI is started from the command line. Make sure that the current working directory is the case directory created in the previous step.

NavPak simulations always use parallel execution. From the number of processes selected by the user, 1 process (1 core) is used to drive the GUI and others execute the CFD work. To start the execution, use the following command:

```
mpirun -np N ../navpak.dist/navpak.bin .
```

where N is the number of processors that should be used for the simulation. For example, to run the application using four processors:

```
mpirun -np 4 ../navpak.dist/navpak.bin .
```

The path to `navpak.bin` must point to the executable contained in the extracted `navpak.zip` directory. Adjust the relative path if the directory is located elsewhere. The final period (.) in the command is important. It specifies the current directory as the working directory for the GUI and tells the application which simulation case to use. The GUI may take some time to start, particularly on machines with limited computational resources. This delay is normal.

Under normal operation, **NavPak** will automatically perform all parallel processing operations as needed. Parallel decomposition, data reconstruction, management of objects and other auxiliary operations are done without user intervention.

6 Configuring the Simulation

Once the GUI has started, the simulation is configured and run using the **Menu**, **Meshing**, **Simulation**, and **Results** tabs. The general workflow is:

1. Select the simulation type and configure the physical parameters;
2. Convert/read/create and load the mesh;
3. Configure the flow boundary conditions on named mesh patches;
4. Initialise the solver;
5. Configure the simulation time and number of iterations;
6. Run the simulation;
7. Monitor the simulation and inspect the results.

6.1 Configuring Physical Properties

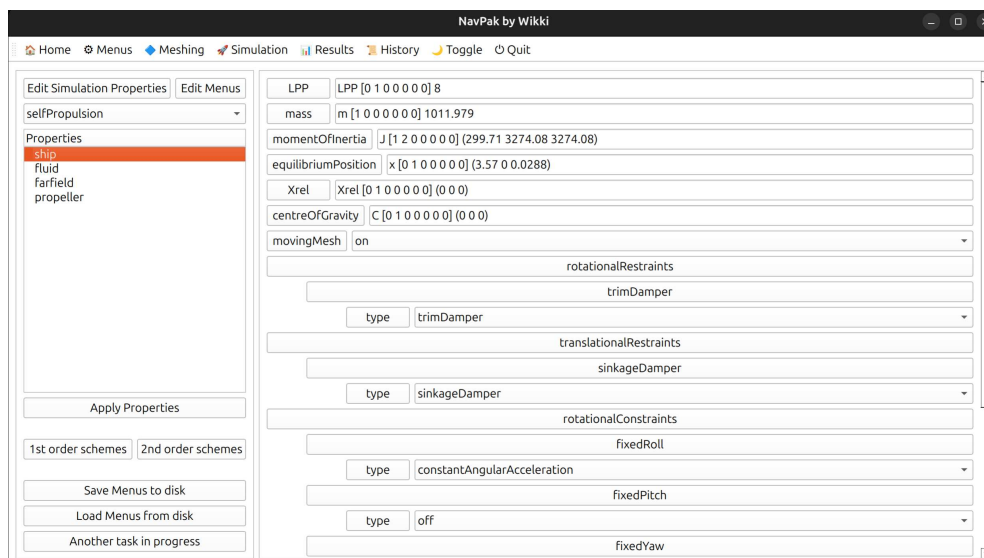


Figure 6.1: Menu tab.

Open the **Menu** tab and select the appropriate simulation type, or load simulation properties from a previous simulation (via a **pkl** file).

The **Menu** tab provides access to the main physical and environmental parameters of the simulation. These parameters are organised into the following categories:

- ship
- fluid
- farfield

- propeller

The default configuration corresponds to the Yipeng benchmark case. If the objective is to reproduce the benchmark case, the default values can be left unchanged when running a tutorial. For practical cases, the user will adjust ship hull parameters, fluid properties or operating conditions as needed by the simulation case.

If any property is modified, click the **Apply properties** button before continuing.

Changing these menus and applying properties will automatically update complex settings needed for the CFD simulation. This process is abstracted to the user to allow quick and accurate setup of the cases.

6.2 Mesh Functionality

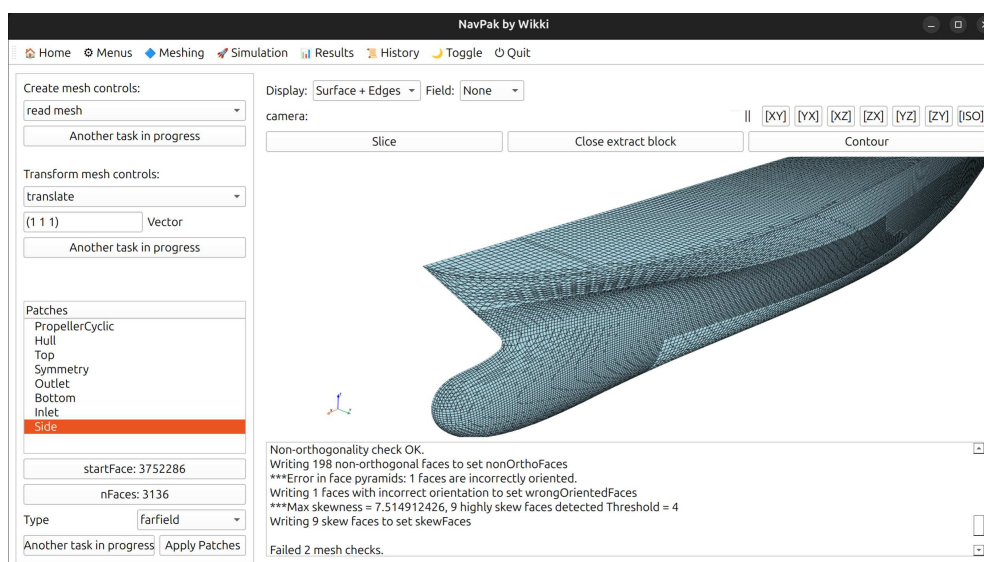


Figure 6.2: Meshing tab.

After configuring the simulation type and physical properties, open the **Meshing** tab. For a demonstration case select **read mesh** and click **Create mesh**. The GUI will read and process the mesh. Depending on the size of the mesh and the available computational resources, this operation may take some time. Wait until the mesh has been completely read before proceeding.

Once the mesh has been successfully read, click **Load mesh**. Loading the mesh into the GUI may take a few seconds. After the mesh has been loaded, the available mesh patches are displayed in the **Meshing** tab.

Each patch must be assigned the appropriate boundary type. Once all patches have been configured, click **Apply Patches**. This automatically defines the boundary conditions for the simulation. Velocities and turbulence properties needed for the definition of boundary conditions come from the previously defined Menus.

Types of boundary conditions are given in Table 2: they are translated into boundary conditions for individual fields as needed by the solver.

Patch	Boundary Type
Hull	movingWall
Top	atmosphere
Symmetry	symmetryPlane
Outlet	outlet
Bottom	seabed
Inlet	inlet
Side	farfield
Propeller	propeller

Table 2: Boundary condition specification.

7 Running the Solver

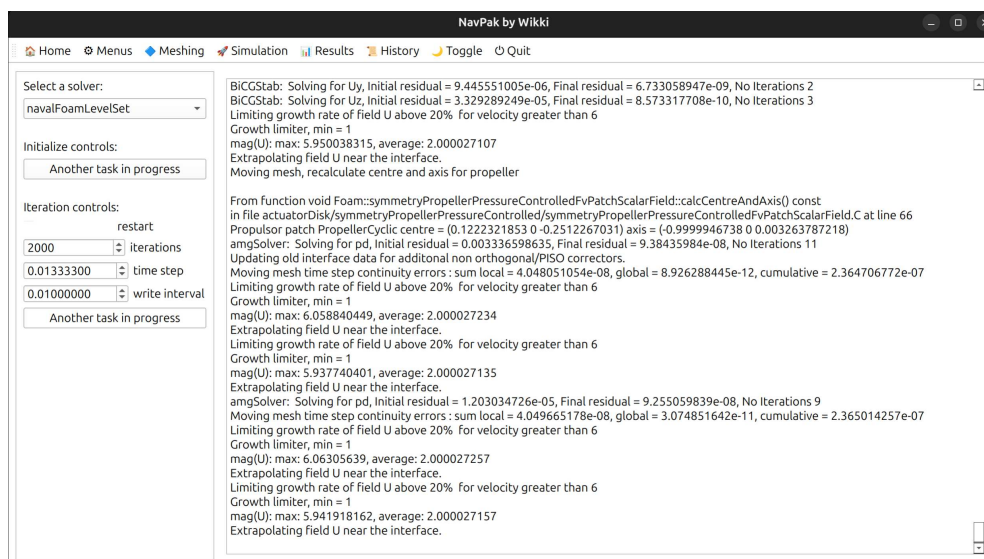


Figure 7.1: Simulation tab.

7.1 Field Initialisation

At the start of the simulation, it is necessary to initialise the fields. This primarily involves the free surface shape, based on the far field properties menu settings. Initialising the solver will create a free surface conforming to the initial guess, either with calm water for steady resistance or with prescribed far-field waves.

To initialise the solver before the solution, open the **Simulation** tab and Select: **navalFoamLevelSet**.

After selecting the solver, click **initialise solver**. The GUI will prepare the simulation using the mesh, physical properties, and apply individual patch-and-field boundary conditions configured in the previous steps.

7.2 Simulation Controls

The simulation time step is determined automatically from the relevant case parameters. These include the ship's length between perpendiculars (LPP), the wave period, and other simulation parameters. For a standard benchmark simulation, the automatically calculated time step can normally be used. If required, the time step can be specified manually through the corresponding simulation parameter in the **Simulation** panel. Specify the number of iterations that the solver should perform. In a transient simulation, an iteration corresponds to a simulation time step. For an initial test, it is recommended to use a small number of iterations.

A short test run is useful for verifying that the mesh, boundary conditions, solver configuration, and physical parameters have been set correctly before starting a longer simulation. Once the desired number of iterations has been specified, click **RUN** to start the simulation. This tab will present the simulation logs.

8 Results and Simulation Monitoring

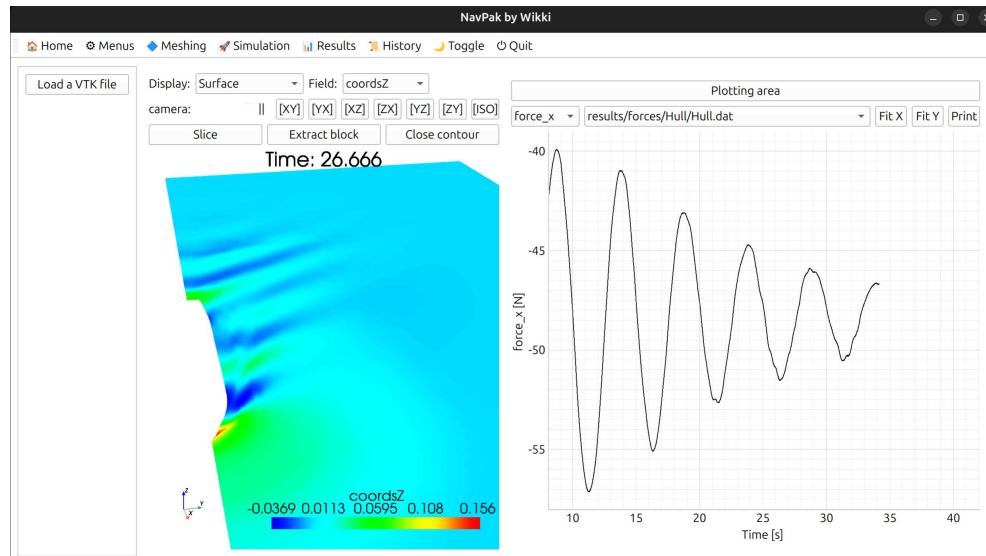


Figure 8.1: Results tab.

During the simulation, the **Simulation** and **Results** tabs can be used to monitor the progress of the calculation.

The **Results** tab provides plots of important simulation quantities, including: **forces** and **residuals**. These plots provide an overview of the numerical progress of the simulation. Results are updated as simulation time steps are completed.

8.1 Accessing Simulation Results

After the simulation time step has been completed, the solver generates a corresponding results directory.

To open and load the results in the VTK viewer within **NavPak**:

1. Open the **Results** tab.
2. Click **load a VTK file**.
3. Navigate to the results directory corresponding to the desired simulation time.
4. Select the file: **results/your_simulation_time/results.vtk**

8.2 VTK Post-Processing

The VTK viewer provides several tools for inspecting the computed solution. The available tools can be used to:

- visualise result fields;
- extract individual patches;

- create cut slices through the computational domain;
- display contours of solution fields;
- inspect the free-surface shape.

The free surface can be identified using the `levelSet` field. The resulting `levelSet = 0.0` iso-contour represents the free surface. To visualise the wave elevation, select `coordsZ` as the colouring field. The free-surface contour will then be coloured according to the `coordsZ` value. This provides a visual representation of the wave elevation over the free surface. This visualisation is useful for examining the shape of the generated waves and the interaction between the free surface and the ship.

To create a free-surface contour:

1. Select the field `levelSet`;
2. Click the **contour** button;
3. Create a contour using the value `0.0`;
4. Colour it with `coordsZ`.

If this post-processing functionality is not sufficient, VTK files can be directly loaded into external post-processing tools, such as ParaView.

9 Advanced User Interface Functionality

Under normal conditions, the user is expected to operate exclusively under the GUI interface, manipulating basic simulation settings and relying on default numerics. In this section, further GUI functionality for parametric studies, check-pointing and process automation is available.

9.1 Logs, Scripting and Parametric Studies

During the run, the case directory is populated with raw simulation results data. The directories contain log files, results in terms of force, motion and residual traces, **json**-formatted settings files, and field data for the solver. They can be inspected manually and modified for further use.

9.2 Saving and Exchanging Simulation Cases

When a simulation case needs to be moved between computers, this can be achieved by transferring the computational mesh and the complete simulation settings file in **pkl** format. The **Menus** panel offers options to save menus to disk (**pkl** format) and load them in the same format without further interaction. This can be used to recover the state of the GUI in case an operation has been interrupted.

9.3 Command-Line Dictionary Export

If it is necessary to compare the performance of **NavPak** against its command-line equivalent, the **Menus** panel offers to write case dictionaries in the Naval Hydro Pack dictionary format. This is to be used for debugging purposes only.

9.4 GUI Interface for Advanced Users

In rare cases where it is necessary to inspect individual discretisation, solver or dynamic mesh settings, users have access to the detailed definitions by clicking on **Edit Menus**. This is recommended for the expert users only.

10 In Case of Problems

In case of problems with GUI or simulation, inadequate results or longer-than-expected run-time, please contact Wikki for support.