



RhinoCFD Quick Start Guide

RhinoCFD Marine

RhinoCFD Official Document Produced by CHAM
May 4, 2018

Introduction

This Quick Start Guide introduces RhinoCFD Marine as a package for the numerical analysis of ships hulls and general objects at a water-air interface. Marine simplifies the set-up of simulations with a less complex menu and automation of time consuming tasks. It also brings with it several new features such as improved interface modelling with Volume of Fluid (VOF) methods and wave simulation capabilities based on Stokes wave theory.

This guide focuses on free surface models applied to ships hulls. It will get the user up to speed with the main menu, steady (time independent) simulations using the Height Of Liquid (HOL) free surface model and transient (time dependent) simulations using a variant of the VOF method. The hull geometry used herein is the DTMB 5415, shown in Figure 1, which can be downloaded from [RhinoCFD Tutorials](#), but users can also use their own geometry.

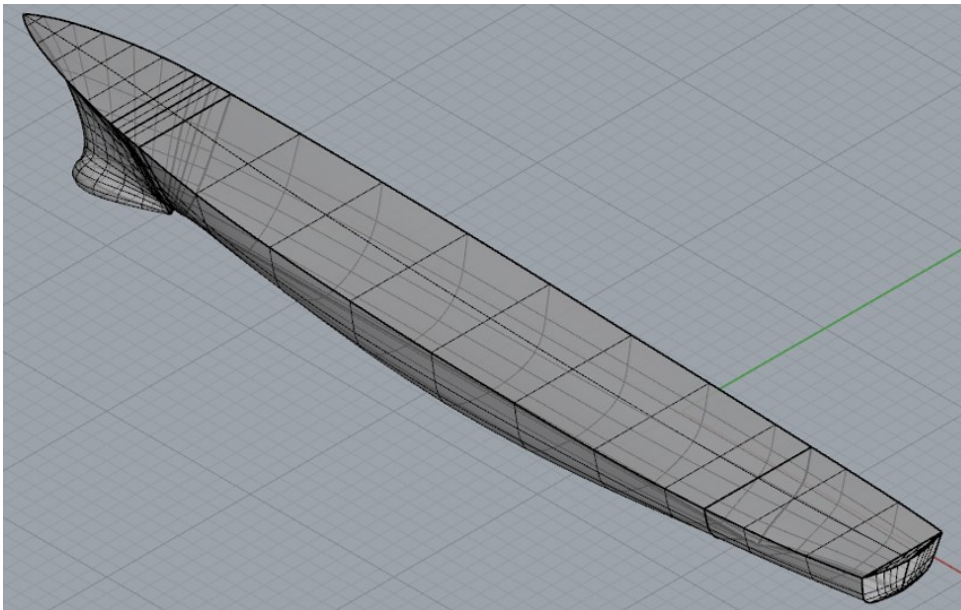


Figure 1: DTMB 5415 Hull Geometry

See the [RhinoCFD Basics Transient](#) videos for more information on running transient simulations.

CFD Analysis

Starting RhinoCFD Marine



Figure 2: RhinoCFD tool bar

To start, load the hull geometry into Rhino and ensure that the object is oriented such that the Z-axis is pointing upwards and the hull is pointing in the positive X-direction. Also ensure that Rhino is working in metres and the boat is approx 60m long.

Start RhinoCFD by creating a domain around the hull by selecting the first button (Erlenmeyer flask) of the RhinoCFD toolbar shown in Figure 2. Selecting the desired working directory (in which simulation data and auxiliary files will be stored) will prompt a second window, Figure 3. From the drop-down menu on the right select the ‘MARINE’ menu option instead of the default ‘Core’ option.

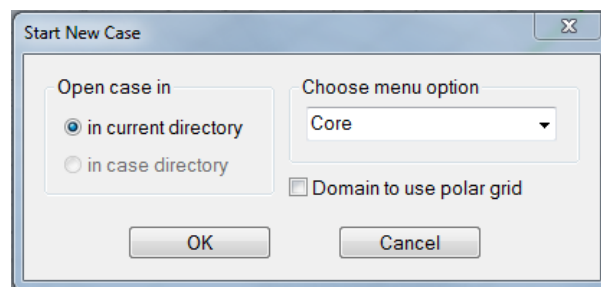


Figure 3: RhinoCFD Start New Case Prompt

You will first be prompted to select the hull. Using the mouse, left click on the dtmb geometry. Please note that for auto-meshing to work, you can only have one hull in the domain. A single hull geometry can be created from many objects by combining all hull geometry as a block.

RhinoCFD Marine Main Menu

Once the hull has been selected the RhinoCFD Marine Main Menu, Figure 4, will appear and in the background the domain, inlets and outlets as well as initialisation objects will be created automatically. Once OK is selected on the main menu, RhinoCFD Marine will then adjust the hull location, the domain size and water height to the default values set in the main menu. The main menu can then be opened again at any time by clicking on the second icon in the RhinoCFD toolbar.

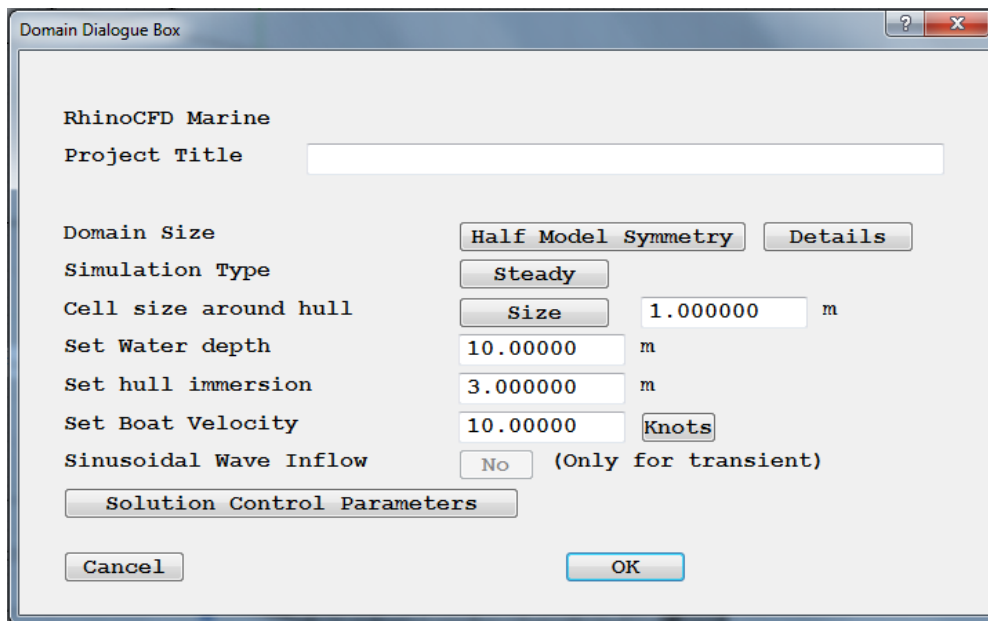


Figure 4: RhinoCFD Marine Main Menu

Domain Size

By default only half of the hull will be simulated, with a symmetry plane situated along the centre line of the hull. This can be changed by clicking the 'Half Model Symmetry' button, but for computational economy will be left on default for examples in this guide. Clicking on details will allow the size of the domain to be changed in each axis relative to the size of the hull. The expansion factor for the mesh can also be changed here. Leave the domain size as half model and all detailed settings as default for any examples in this guide.

Simulation Type

RhinoCFD Marine simulations can be either 'Steady' or 'Transient'. The Height Of Liquid method is used for steady cases, however its use should be limited to scenarios where the water-air-interface is not convoluted i.e. not overturning. It is a simple and effective method for analysing basic flows, but is limited to simple steady state analysis. Transient simulations use the Volume Of Fluid (VOF) model, and should be used to analyse scenarios where flow around the hull is not steady (time dependant). Transient simulations require more time to complete and a large amount of storage, so should only be used when necessary.

Cell Size around Hull

This is an easy way of setting the grid around the hull. The initial setting of 1m sets each cell to be as close to 1m as possible, decreasing this to 0.5 will double the number

of cells in each axis i.e. an 8 fold increase in cells. For more complex hull shapes, a more refined mesh will be needed, please see later section on the [grid setup](#).

Automatically Set Parameters

Air and water have been set automatically and the water level, and height of water relative to the hull are set using the 'Set Water Depth' and 'Set Hull Immersion' cells in the main menu. Additionally, 'gravitational forces' and 'buoyancy model' are set automatically with a gravitational acceleration of -9.81 in the Z-direction.

Boat Velocity

The next option sets the velocity of the air and water flowing past the hull. This is set in knots as default, but can be changed to m/s by clicking on the 'knots' button.

Solution Control Parameters

Additional settings can be found by selecting the 'Solution Control Parameters' button, which will open a new window shown in Figure 5.

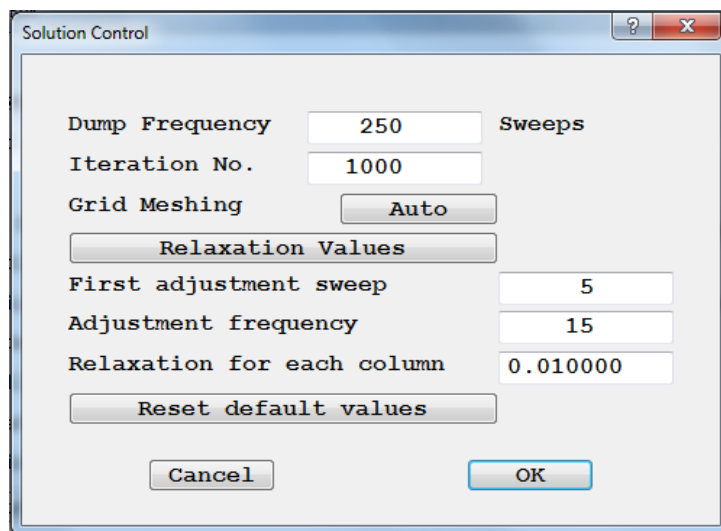


Figure 5: RhinoCFD Marine Main Menu

Here, the total number of iterations can be set as well as the frequency at which the output is produced. Fine grained control of the grid spacing can also be accessed from this menu.

Settings relating to the water modelling are found here, however, in general, the default settings should be able to produce good results.

For some flow problems the solution may be more difficult to attain than for others, and it then becomes necessary to change the solution procedures relaxation parameters. This can be done by selecting the 'Relaxation Values' button.

More complex geometries will generally require smaller cell sizes (finer grids) near the hull, take more iterations to converge and need stronger relaxation. More information on Relaxation and Convergence can be found in e.g. the [RhinoCFD Basics Convergence Tutorial](#).

Setting up a Steady Simulation

To run a steady simulation, ensure that Simulation type is set to Steady (this can be changed back from transient to steady by clicking on the 'Transient' option). Change the Cell size to 0.75 and the velocity to 30knots. You should see the following settings on the main menu dialog:

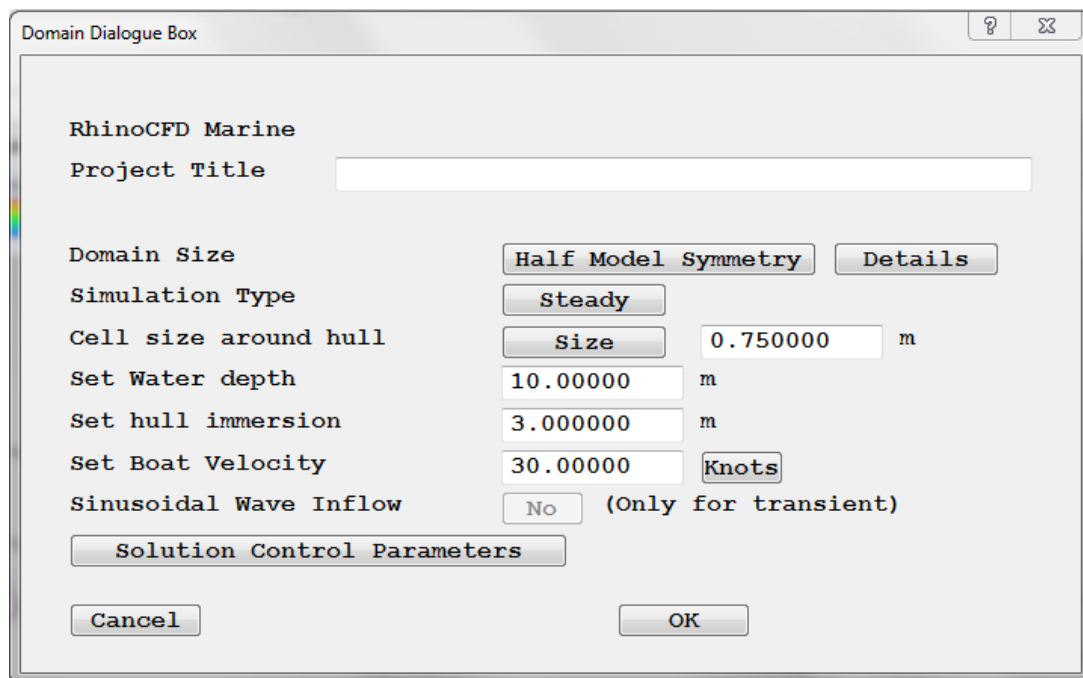


Figure 6: Steady simulation set up

Click OK and your simulation will be ready to run.

Setting up a Transient Simulation

Simulations without Waves

To activate a transient simulation, click on steady next to 'Simulation type' and a new option 'simulation time' will appear. Change the simulation time to 5 seconds and increase the velocity to 40 knots. This should cause a larger wave to appear at the bow, that a steady simulation may have trouble picking up.

Click ok and the simulation will be ready to run. This simulation will take around 20-30 minutes to complete.

Simulations with Waves

Once transient settings have been activated the option ‘Sinusoidal wave inflow’ becomes available in the main menu, click on No next to this to activate the 3 wave options. The first option governs the maximum height of the wave, the second governs the time it takes for a wave to travel from the waterline to its peak then to trough then to the water line again (i.e. the lower the period the more waves in the domain). It is important to note that there are no imposed limits on the period or wave height, so be sure use practical values so that the results are realistic. Typical wave heights range from approximately 0 → 2.5m and typical periods from approximately 2 → 30seconds. The final options allows the wave speed to be changed, the default setting for wave to travel at the speed of the boat. Clicking on ‘Same as boat’ will allow a user set value to be entered.

Enter a wave height of 0.5m and period of 5 seconds and reduce the velocity to 30 knots. Leave all other settings the same as in the previous transient simulation.

Grid Set-up

If desired, the domain and grid spacing growth can be altered by selecting the ‘Details’ button. The automatically generated grid can be inspected by selecting the globe icon (5th icon) of the RhinoCFD toolbar. The user has 3 options to adjust the grid spacing with increasing levels of control:

1. In the main menu by reducing the value in the cell ‘Cell Size Around Hull’
2. Double clicking on the grid in the active view windows and adjusting the slider for each direction
3. Manually setting spacings for each region under the ‘Solution Control Parameters’ button and subsequently switching the ‘Grid Meshing’ option from ‘Auto’ to ‘Manual’

For option 3 a typical manual grid, for the full dtmb hull, may follow that shown in the following table.

Axis	Region				Average cell size (m)
	1	2	3	4	
X	15	120	20	-	1.17
Y	10	17	10	-	0.97
Z	10	4	12	10	1.05

(The Average cell size has been included so that users can try to replicate this grid with differing geometry and cell sizes). Note that the first and last regions in X, Y and Z are given -1.2 and 1.2 geometric power laws respectively.

Locating the Probe

Locate the probe using the RhinoCFD toolbar by left clicking on 'show probe', then ensure it is positioned behind the boat and just below the water line.

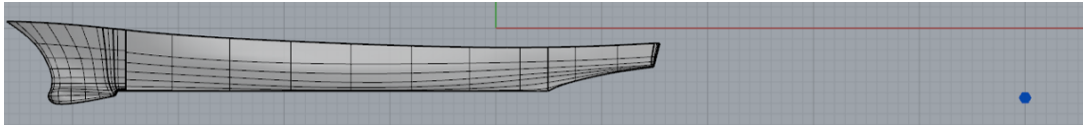


Figure 7: Probe location

Running the simulation

Once the simulation set up has been completed, click the 'run solver' toolbar icon and run on a single processor (multiple processor are not currently available for RhinoCFD Marine). For a steady simulation wait for the simulation to converge or complete the specified number of sweeps. For a transient simulation wait for all time steps to finish.

Results

This section will present the results of the steady and transient simulations set up in this guide, and provide some insight into investigating the results of your simulations.

Steady Simulation Results

Click on load results and type in a suitable name for the simulation, for example 'steady' and click save. This will save all files for this set up and can be loaded again at any time. Click cancel on the dialog asking if you would like to save the dumped files as this is only required for transient simulations. Finally click OK on the dialog that appears.

Choose pressure (p1) and rotate the cutplane 90 degrees to view the distribution of pressure y axis. There should be a distribution in pressure from the top to the bottom of the water region.

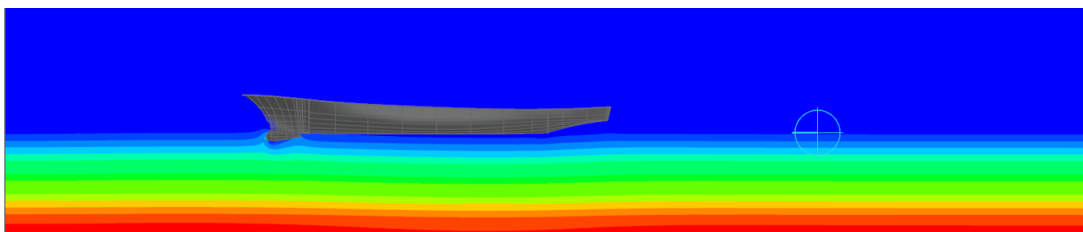


Figure 8: Pressure contour at Y plane

Rotate the probe so it lies along the width of the hull and move it so that is very close to the water line. Change the variable to volume fraction and you should see something similar to the following image.

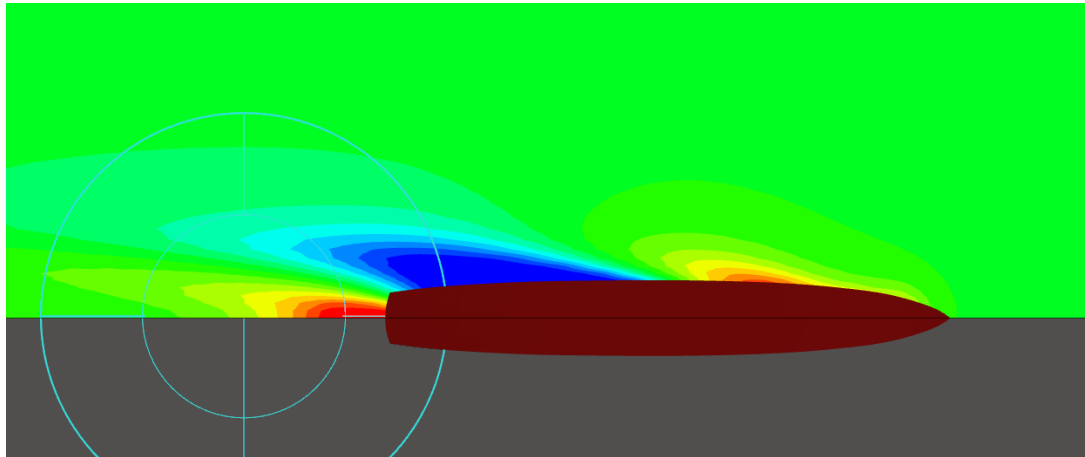
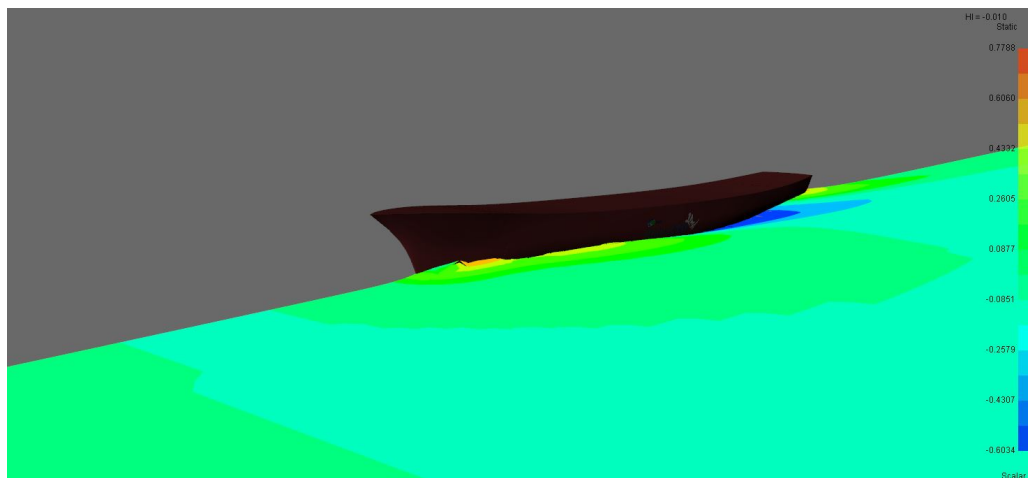


Figure 9: VFO Cut Plane Top View

This shows the wake regions around the hull. The water regions are identified by red colouring and the air by blue.

We can view the water surface showing the ripples produced by the boat. Hide all the other visualisations and create an IsoSurface probe from the probe drop down or from the results panel. Change the Scalar to HI and 'on' to volume fraction. Click on the Isosurf tab, then untick the box labelled 'use probe value' and type in 0.5 as the Isosurface value. The IsoSurface should locate to the exact point between the water and air region (you may need to move the Isosurface probe in the domain for the new settings to be activated). Inspecting the IsoSurface you should see the wave effects forming around the hull. These waves are now also coloured by the distance from the water line. Make sure to toggle the scale (third icon after the break) to see what height the colours represent.

(a) Front view



(b) Rear view

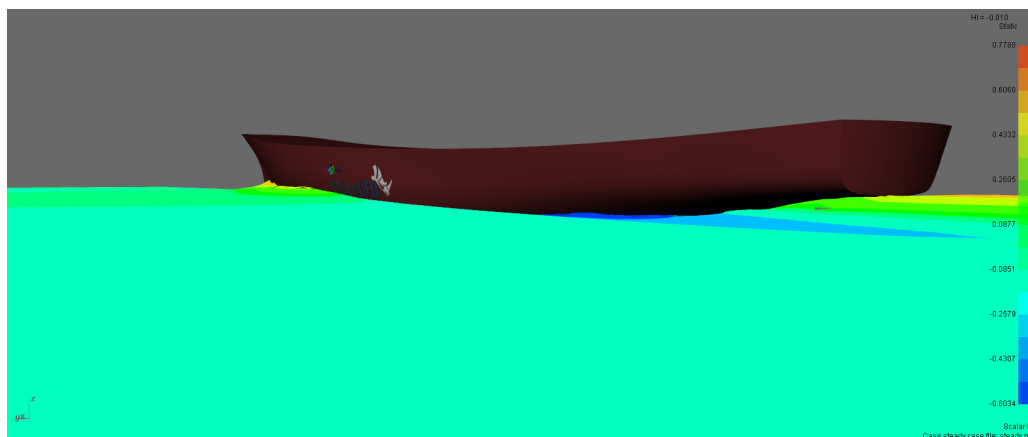



Figure 10: Isosurface Plane of Volume Fraction Coloured by HI,

To view the text results relating to your simulation, right click on the third to last toolbar icon  to bring up the 'result' file. As the name suggests, this file contains all the information resulting from the simulation, including convergence, object properties, settings and derived variables and forces. To view the forces scroll down to the very bottom until you see information structured as in the following figure.

Integrated force on object: HULL

```

-----
Fx = -1.432949E+05 (Pressure= -1.038750E+05, Friction= -3.941988E+04)
Fy = -1.222415E+06 (Pressure= -1.222724E+06, Friction= 3.093417E+02)
Fz = 1.346662E+06 (Pressure= 1.346828E+06, Friction= -1.661388E+02)
Pressure force on West side: 7.840651E+04
Pressure force on East side: -1.822815E+05
Pressure force on South side: 5.515106E+01
Pressure force on North side: -1.222780E+06
Pressure force on Low side: 1.705965E+06
Pressure force on High side: -3.591367E+05
Force unit vector: -7.854480E-02 -6.700473E-01 7.381513E-01

Total moment about X axis = -1.321460E+07
Total moment about Y axis = 2.125950E+08
Total moment about Z axis = 1.975873E+08

Moment of Fx about Y axis = 1.231471E+06 at distance Z = 8.593966E+00
Moment of Fy about X axis = -1.161758E+07 at distance Z = 9.503791E+00
Moment of Fz about X axis = -1.597125E+06 at distance Y = 1.185988E+00
Moment of Fx about Z axis = -1.445574E+05 at distance Y = 1.008811E+00
Moment of Fy about Z axis = 1.977335E+08 at distance X = 1.617564E+02
Moment of Fz about Y axis = 2.113632E+08 at distance X = 1.569534E+02
Total exposed vertical area = 2.025389E+02
Sum vert. area with P1 > 10.00 = 2.025389E+02
Ratio area P1> 10.00/Totalarea =100.00 %

```

Integrated forces for all included objects

```

-----
Fx = -1.432949E+05
Fy = -1.222415E+06
Fz = 1.346662E+06
Ftot= 1.824371E+06

```

Figure 11: Forces on Hull

The forces and drag information is also exported to a .csv file called 'Forces' and can be found in the working directory. This file provides information on all forces, moments and drag against the number of sweeps completed.


ISWEEP	FT	FX	FY	FZ	MOMX	MOMY	MOMZ	XBAR	YBAR	ZBAR
250	1.92E+06	-1.25E+05	-1.23E+06	1.47E+06	-1.35E+07	2.29E+08	1.97E+08	1.55E+02	1.02E+00	8.43E+00
500	1.86E+06	-1.35E+05	-1.22E+06	1.40E+06	-1.33E+07	2.20E+08	1.97E+08	1.56E+02	1.01E+00	8.52E+00
750	1.84E+06	-1.39E+05	-1.22E+06	1.37E+06	-1.32E+07	2.15E+08	1.97E+08	1.57E+02	1.01E+00	8.56E+00
1000	1.82E+06	-1.43E+05	-1.22E+06	1.35E+06	-1.32E+07	2.13E+08	1.98E+08	1.57E+02	1.01E+00	8.59E+00

Figure 12: Forces on all Objects

Transient Simulation Results

Without waves

Click on load results and type in a suitable name for the simulation, for example 'transient_no_waves' and click save. This will save all files for this set up and can be loaded again at any time. When prompted click OK to save the time sweep files. Finally click OK on the next dialog that appears.

Position the cutplane very close to the waterline. Change the current variable to volume fraction and Load the scalar key (using the third icon after the break) to show information on the time and colour scale of the variable. Then using the time step icon from the RhinoCFD toolbar:  left click until the scalar key shows roughly 'time=0.4'. You should see something like that shown in the top picture below. Now left click on the time step icon and notice the increase in wake around the boat.

(a) 0.4 Seconds



(b) 1.0 Seconds



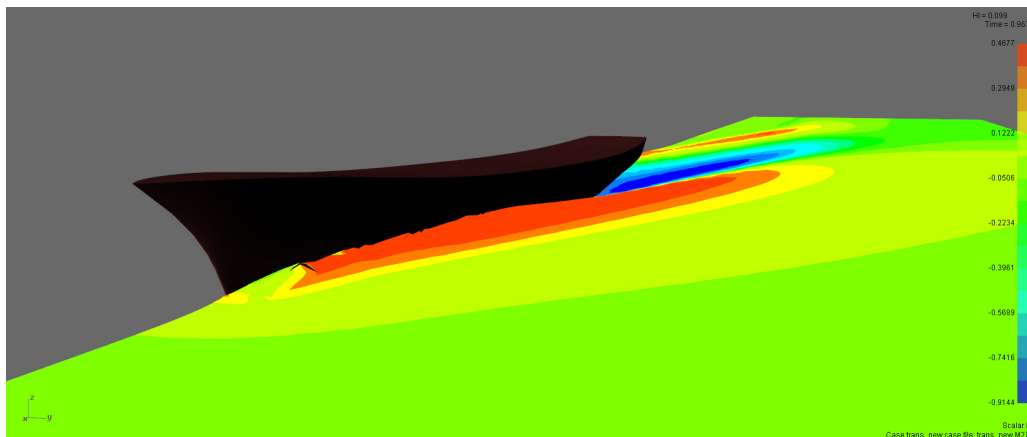
Figure 13: Cutplane Displaying VFOL at Varying Times

Try changing the variable to HI to see the height of the waves.

Hide the cutplane and use then select the isosurface tab in the results panel. Change the

scalar to HI and 'on' to VFOL. . Enter 0.5 into the 'Isosurface' input box and deselect the 'Use probe value' tick box next to it. Click 'Add probe' to generate the isosurface. You may need to move the Isosurface probe in the domain to activate the new settings. Inspecting the Isosurface you should see wave regions around the hull, and these should increase as the time is stepped forward. Use the 'play time series' icon to cycle through all time steps in a row. A front view of the hull will show an increased bow wave compared to the steady simulation.

(a) Front view



(b) Rear view

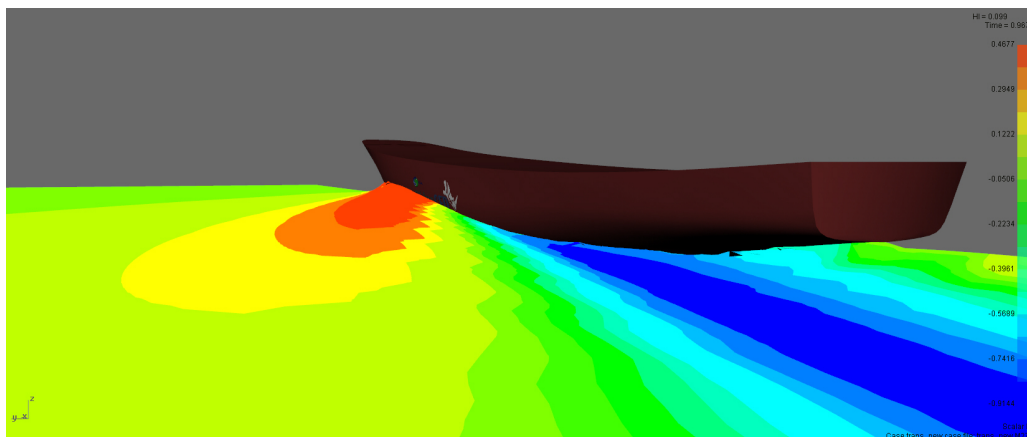


Figure 14: Isosurface Plane of Volume Fraction Coloured by HI, at the final time step

With Waves

Click on load results and type in a suitable name for the simulation, for example 'transient_with_waves' and click save. This will save all files for this set up and can be loaded again at any time. When prompted click OK to save the time sweep files. Finally click OK on the next dialog that appears.

Rotate the cut plane 90 degrees and move it so that it is just inside the domain and near the centre line of the hull. Change the current variable to volume fraction. You should

see waves along the length of the cutplane.

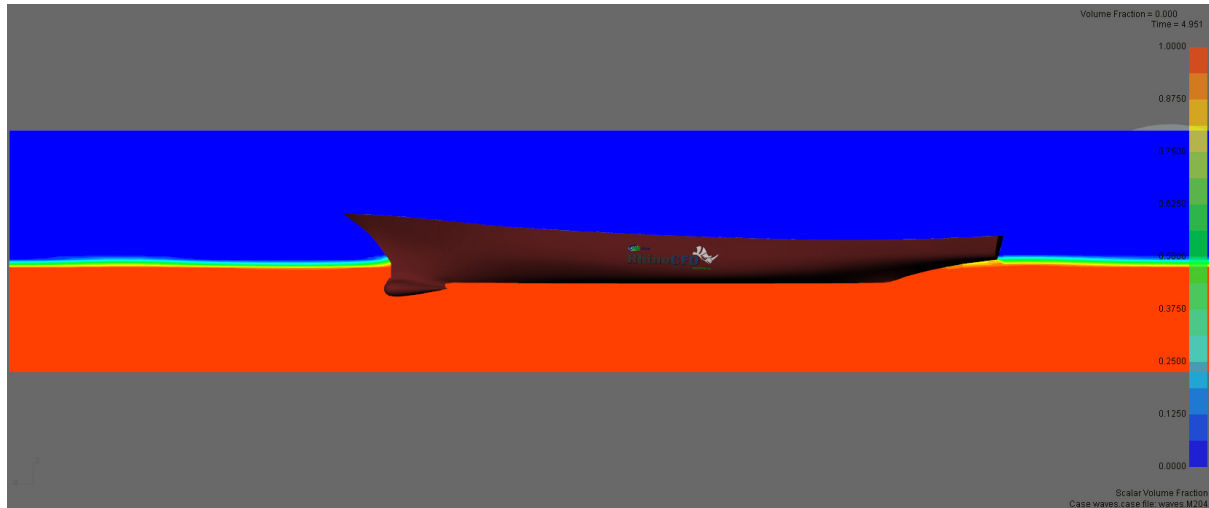
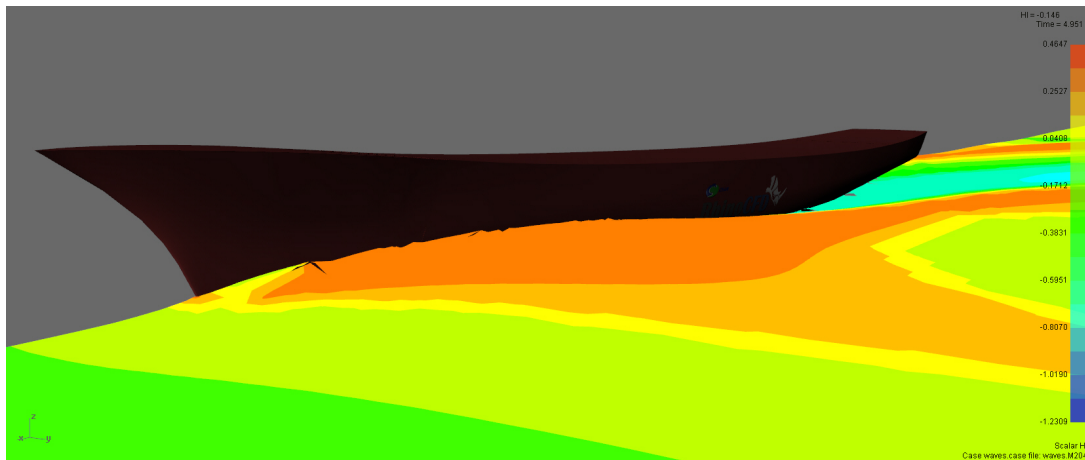


Figure 15: VFOL at Hull Centre-line at final timestep

Create an iso-surface as in the previous section and you will see waves throughout the domain. The wave length and height changes where it is interacting with the hull and will be higher at the bow.

(a) Front view



(b) Rear View

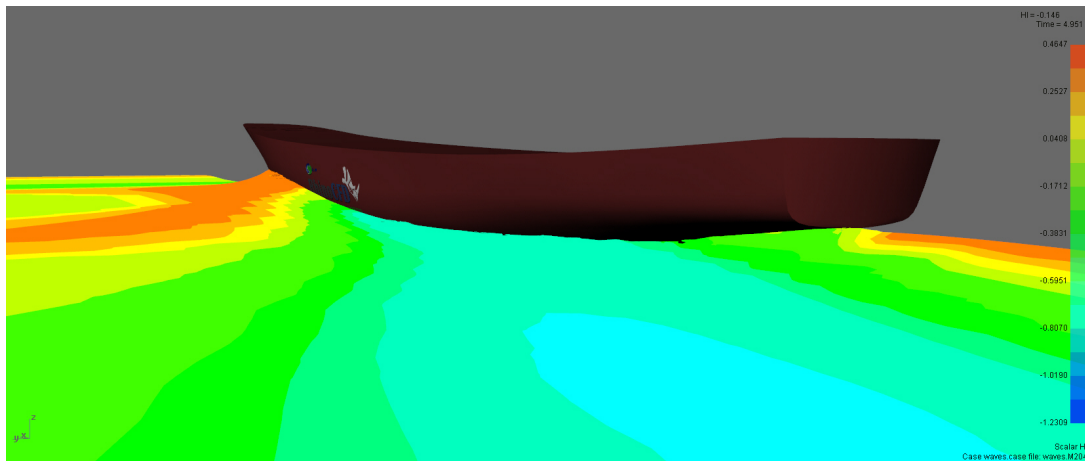


Figure 16: Isosurface showing VFOL coloured by Hi at the final time step

Step through the time series by clicking on 'play time series' to see the waves travel through the domain and interact with the hull.



RhinoCFD
CHAM ltd.
40 High Street, Wimbledon Village
SW19 5AU
London, UK