



---

## RhinoCFD Tutorial

---

Free Surface - Height Of Liquid



RhinoCFD Official Document Produced by CHAM  
October 13, 2021

## Introduction

This tutorial focuses on free surface models applied to boat hulls. It is designed to get you up to speed on how to apply the Height Of Liquid (HOL) free surface model correctly and provide a glimpse in outputting forces and the more-advanced topic of 'InForm'. The .3dm file 'Hull Geometry - DTMB 5415' can be downloaded from [RhinoCFD Tutorials](#) for use in this case. Users can also use their own geometry for this tutorial.

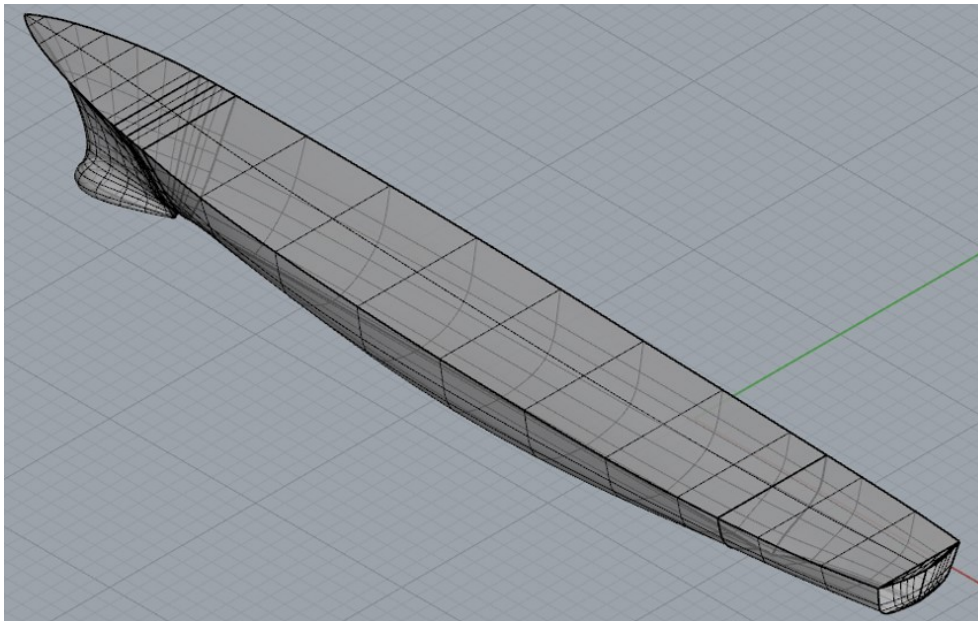


Figure 1: Boat Geometry

To complete this tutorial in RhinoCFD Lite the geometry will need to be simpler to account for the reduction in cells, see the Appendix for more information.

Free surface models are required when there is an interaction between two or more distinctly different fluids, separated by sharply defined interfaces. For this reason it is required for analysing the joint water and air flow past boat hulls. This tutorial focuses on applying the HOL method which should be used when the interface is not convoluted and needs no dispersion device. It is simple and effective at analysing simpler flows, but is limited to simple steady state analysis.

# CFD Analysis

## Main Menu

First, import the hull geometry into Rhino and then create a domain around the boat hull, by clicking on the first button on the toolbar. Select the desired working directory.



Figure 2: RhinoCFD tool bar

Resize your domain using the gumball, leaving enough space for the flow behind the boat to develop. This should be approximately half the boat length to the front and one and half times the boat lengths to the rear. The height of the domain should be increased to around four times the boats height. As a reference the domain is approximately  $180 \times 36 \times 38$  m.

We will now set up the Height Of Liquid model. Enter the main menu and click on models, then change free surface models from off to Height Of Liquid.

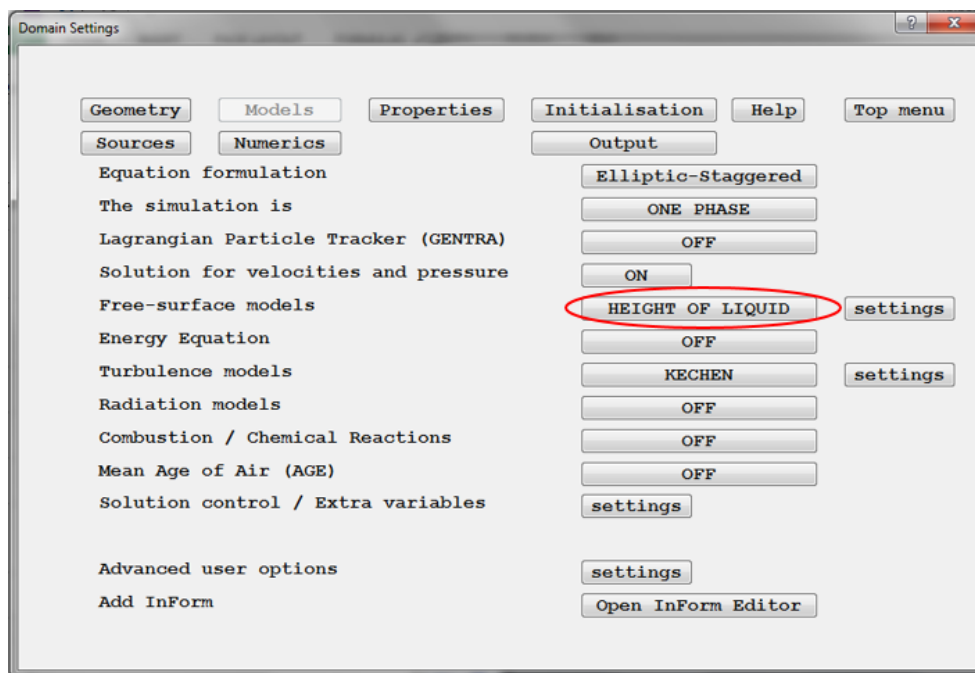


Figure 3: Models settings

Click on Settings to open up a new menu, change 'height direction' to +Z, 'upper cut off

for interface sharpening' to 1 and 'update frequency' to 15.

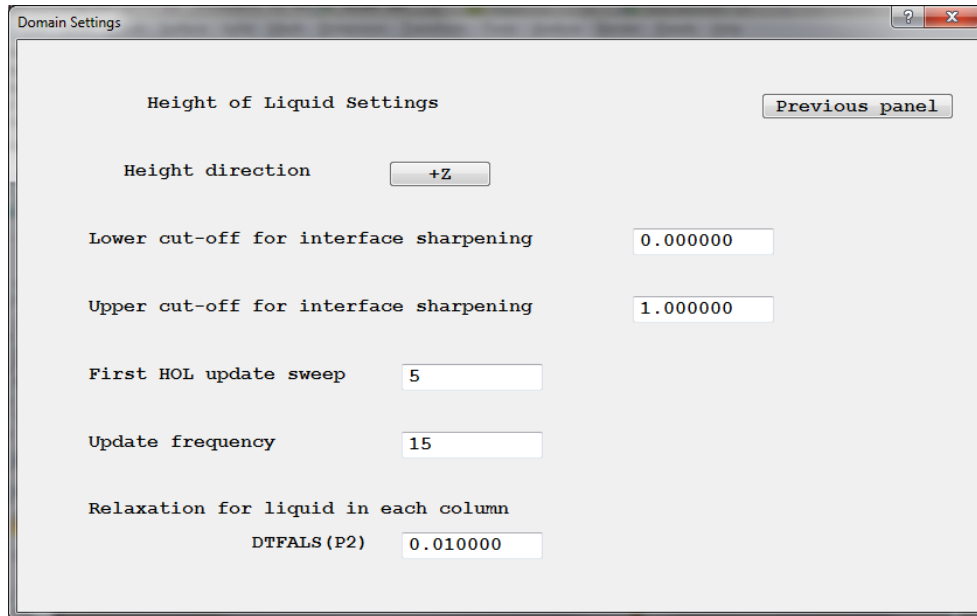


Figure 4: HOL settings

Next enter properties. You should see that there is now the option to select a light and heavy fluid. Air and water have been selected automatically, and are fine for this simulation. Change the setting of 'Domain is initially full of' to heavy.

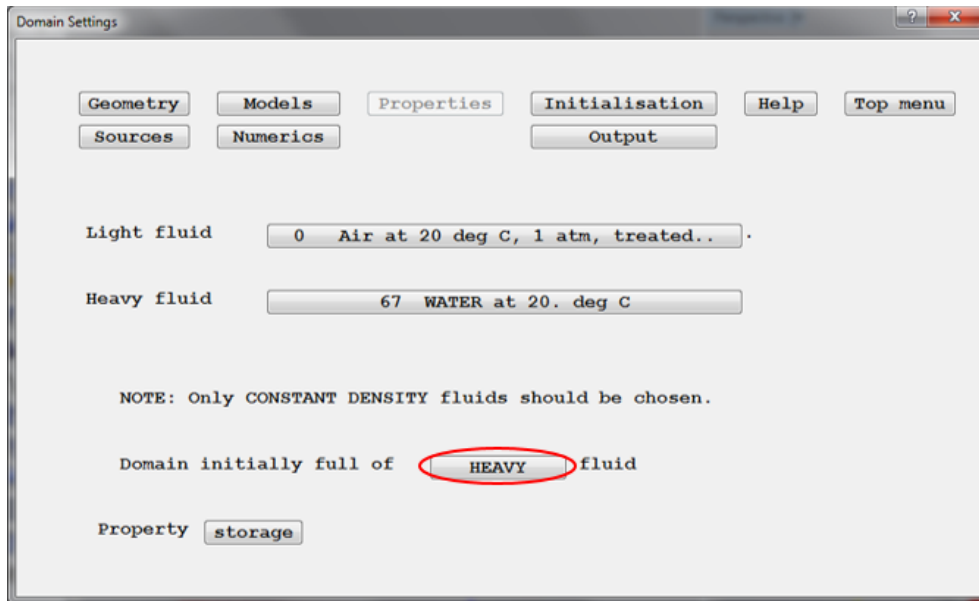


Figure 5: Properties settings

Next enter the sources tab and turn ‘gravitational forces’ on, and ensure the ‘buoyancy model’ is set to constant. The ‘gravitational acceleration’ should be set to -9.81 in the Z axis.

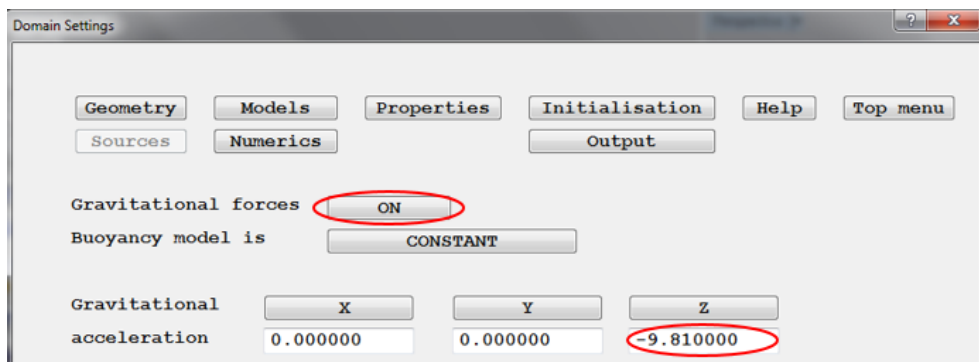


Figure 6: Sources settings

In the Numerics tab set the number of iterations to 3000, then enter relaxation control and change Automatic convergence control from on to off. As a guide line, change the value below U1, V1 and W1 to 0.01. More complex geometry (finer meshes) will require lower values and more iterations. See the RhinoCFD basics [Convergence Tutorial](#) for further information.

## Output of Extra Variables

RhinoCFD can also calculate the forces applied to any object in the domain. To activate this option go to Output on the main menu and select 'output of forces and moments on blockage objects' and set it to ON. Clicking on settings leads to the following panel, where you can include options such as the calculation of Friction forces (part of the drag component) and drag coefficients.

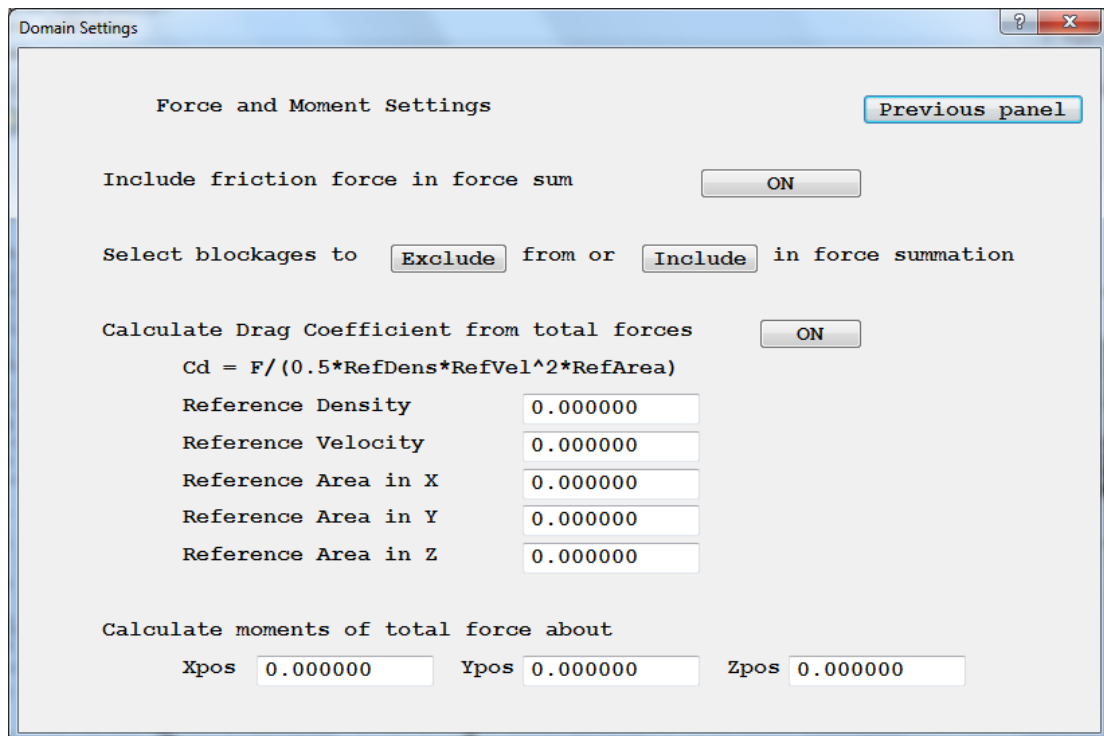


Figure 7: Output Forces Menu

Switch 'Include friction force in force sum' to on. Friction forces are an important component in the drag calculations when viscous fluids are involved. Use the exclude button to ensure that only the hull is included in the force calculations. If your boat is made up of multiple objects then all should be selected in the include menu.

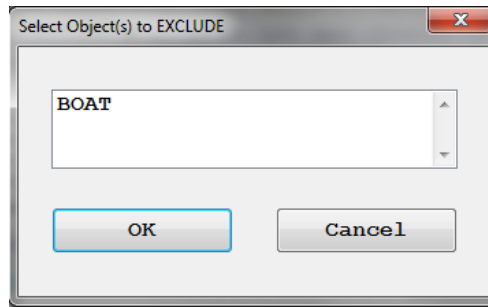


Figure 8: Exclude Dialog

Switch on Drag Coefficients and enter the following data for the tutorial geometry:

Reference	Value
Density	998.0
Velocity	10.0
Area in X	50.5
Area in Y	321.5
Area in Z	430.0

Table 1: Input values for Drag calculation

The density of interest is of course water, the velocity is the same as set at the inlets and the areas in X, Y and Z are the projected areas of the hull in each plane. To calculate the projected area in each plane, either:

- Simply draw a rectangle in each plane, use the area command in Rhino and then approximate how much of the rectangle area to deduct; or
- draw a closed curve in each plane around the hull, create a surface and use the area command to find a more exact value

It is of course recommended that the second option is used as, the closer the value to the realistic reference area, the more accurate the drag calculation.

The moments use the coordinate system of the domain i.e. Xmin, Ymin, Zmin is (0,0,0), ensure the coordinates entered for this moment calculation are with respect to this origin.

Under 'Derived Variables' on the Output page we can also store values such as shear stress and skin friction as well as individual frictional forces in each direction. These variables can be plotted in the domain or on the surface of the hull when viewing the results.

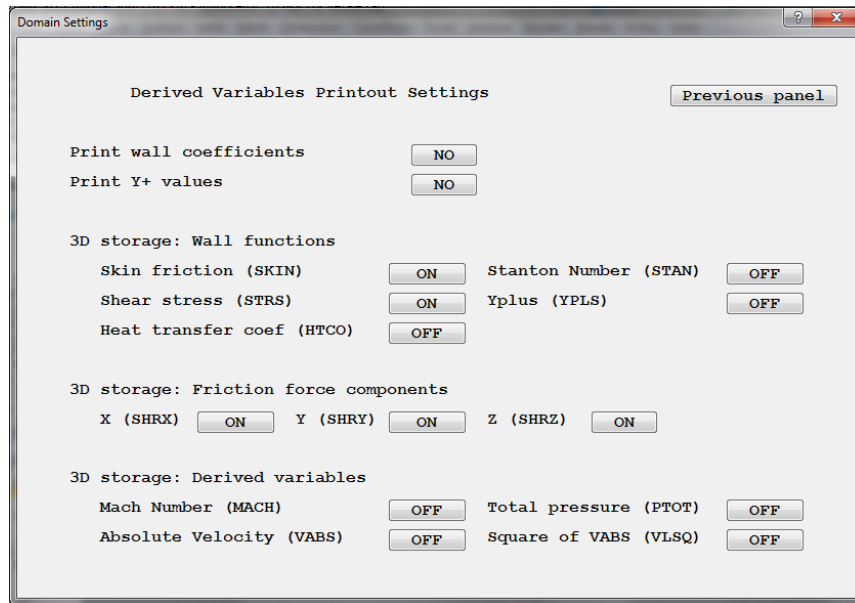


Figure 9: Derived variables Menu

### Locating the Probe

Finally, locate the probe using the RhinoCFD toolbar by left clicking on 'show probe', then ensure it is positioned behind and below the boat.

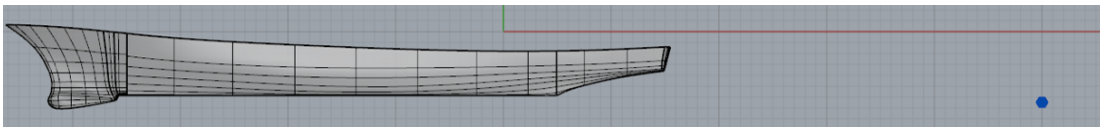


Figure 10: Probe location

### Creating Fluid Boundaries

The next step is to create the boundaries between the two fluids. Place a cube at the top of the domain in order to represent the air. Ensure that this object extends to the boundary of the domain and down to the waterline. The cube should be around a quarter to a third of the size of the domain. Place the boat so that it is between both regions.



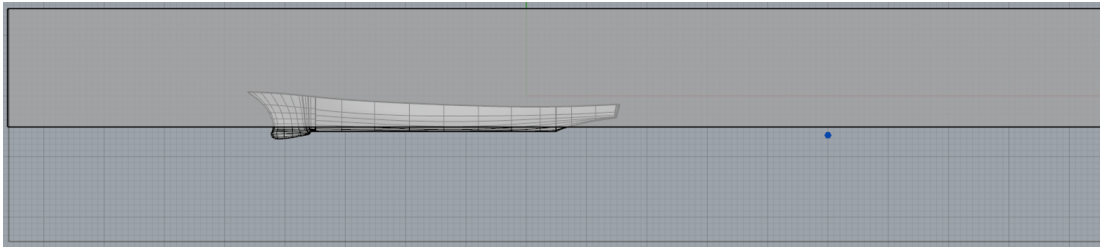


Figure 11: Fluid boundaries around boat

Edit the new cube and change the name to air then enter the attributes menu. Click on types and change this to 'light fluid', click ok and exit the menu.

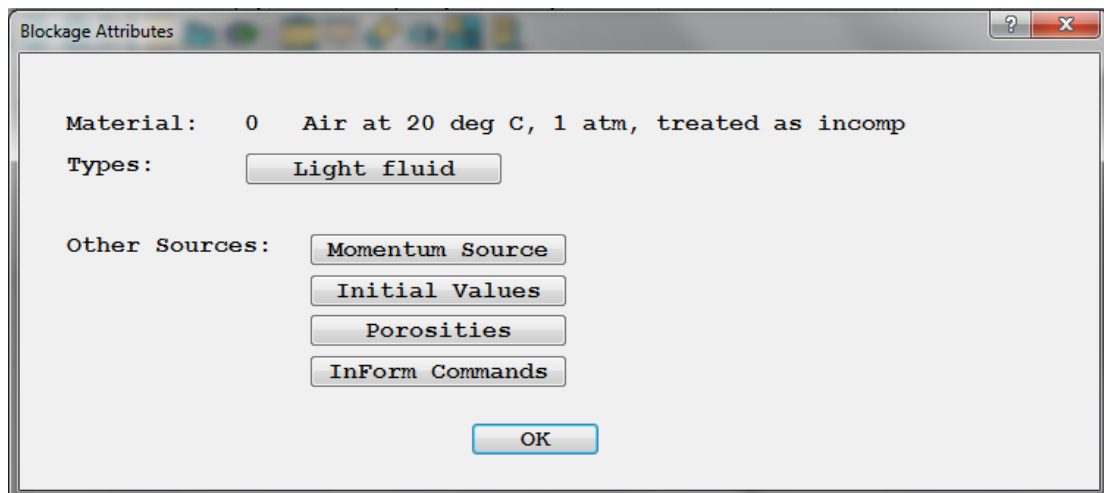


Figure 12: Blockage attributes

## Boundary Conditions

We now need to apply inlets and an outlet to represent the correct fluid in each domain.

### Inlets

Start by creating a cube from the top of the domain to the waterline, with a small thickness in the X direction. The cube can then be moved using the Gumball in just X axis so that it lies half in and half out the domain. Edit the object and change the name to `In_Air`, then change the type to inlet and select object lies in the x plane. Click on attributes to ensure the inlet density is 'light' and change the velocity to 10m/s. Create a second cube in the water region below `In_Air`, edit the cube, change the name to `In_Water` and

type to inlet. Edit the attributes and change inlet density to heavy and x velocity to 10m/s.

## Outlet

Right click on the second toolbar icon to bring up the domain faces menu and make xmax open = yes, then click ok until you exit the menu. It is essential that the name of the outlet is changed, as will become apparent in the next section. Edit the outlet and change the name from DOM\_XMAX\_0 to Exit and ensure that the external fluid is set to 'heavy' in the attributes dialog.

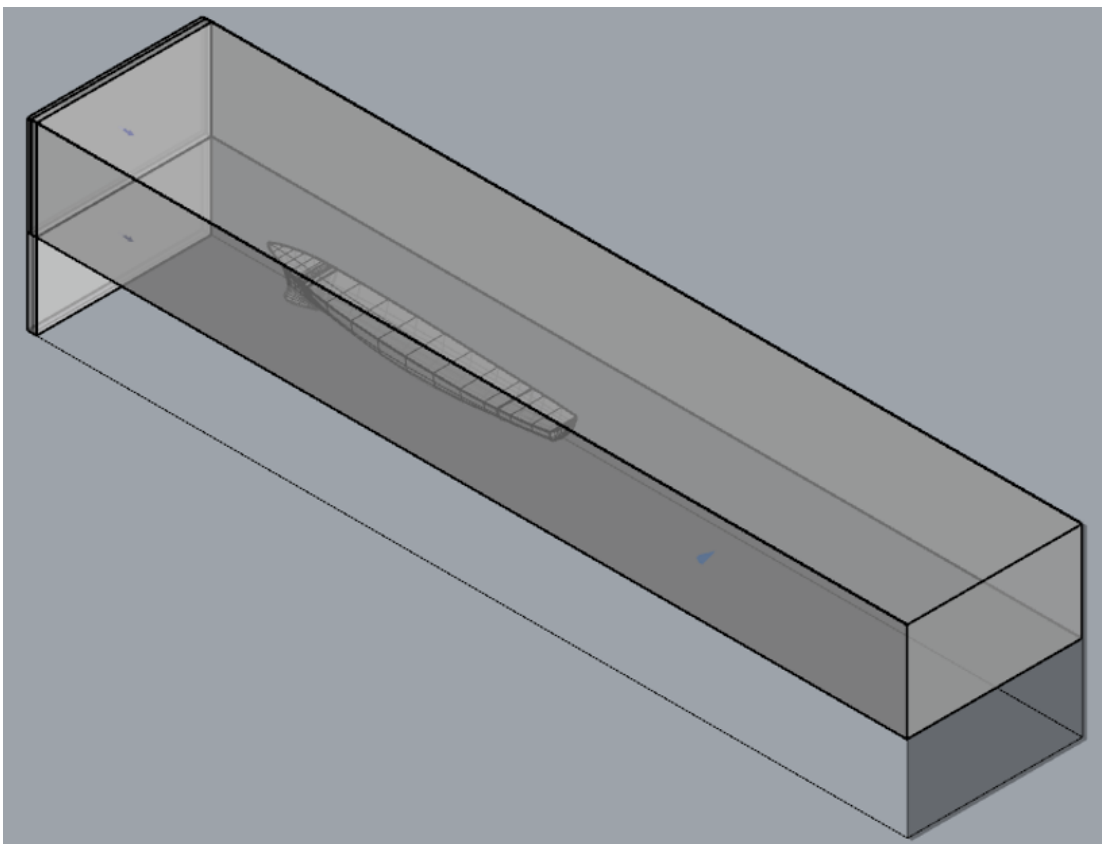


Figure 13: Fluid boundaries

## Object Order

To ensure RhinoCFD correctly models the overlapping boat and air blockages, we need to change the order in which the objects are called. Right click on third icon on the toolbar to bring up the object management menu. Select the boat object, right click above the

reference number and choose ‘page down’ or ‘bottom’ to move it firmly below the air object.

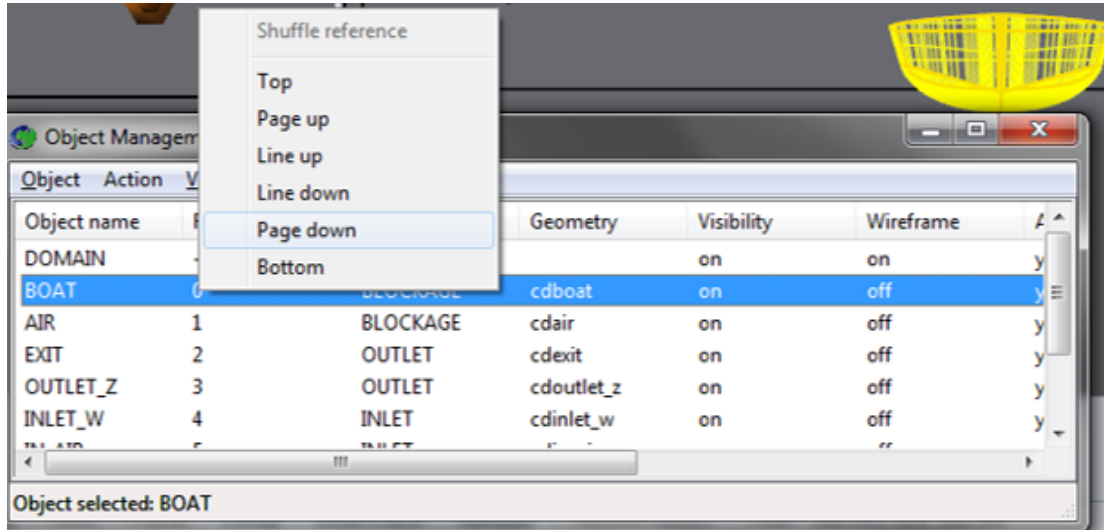


Figure 14: Object order

## Meshing

It is important to refine the mesh around the hull geometry so that the forces and flow attributes are correctly simulated. The boat used for this tutorial is approximately 60m, and has sharp edges so will require a reasonably fine mesh and so we will aim for 0.5m per cell. To do this enter the geometry menu by left clicking on the 4th tool bar icon. ensure that X is set to manual and enter the x-direction menu by clicking on ‘X-direction’. Edit region 2 and type in 120 to get a cell size of approximately 0.5m cells. We then need to match region 1 and 3 so that the first or last cells in opposing regions are not 2 times greater or smaller. Apply a geometric power of -1.2 in region 1 with approximately 15 cells, and then a geometric progression of 1.2 and 20 cells in region 3. Depending on the size of the domain or the boat geometry used, these values may need to be iterated until a reasonable mesh is found. A similar process can be completed for the Y and Z axis. The following table below gives the values used for grid, which can be used as guide lines for other sizes of domains and geometry.

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.

After applying these settings the grid should look as displayed in the following figures:

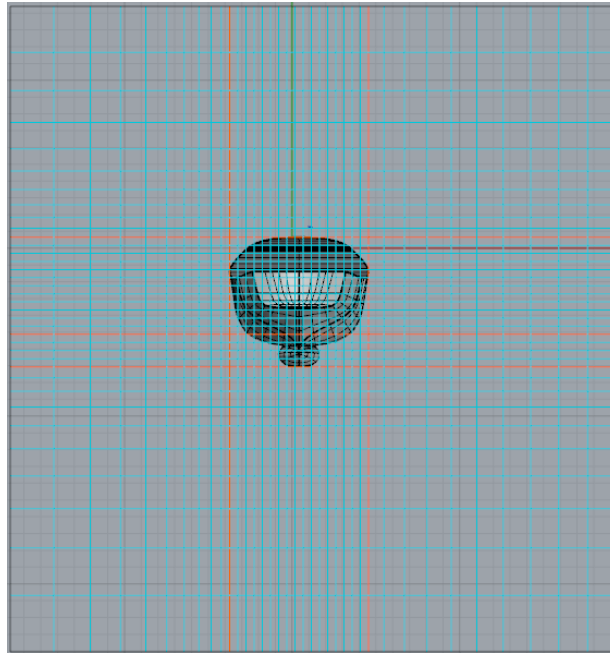
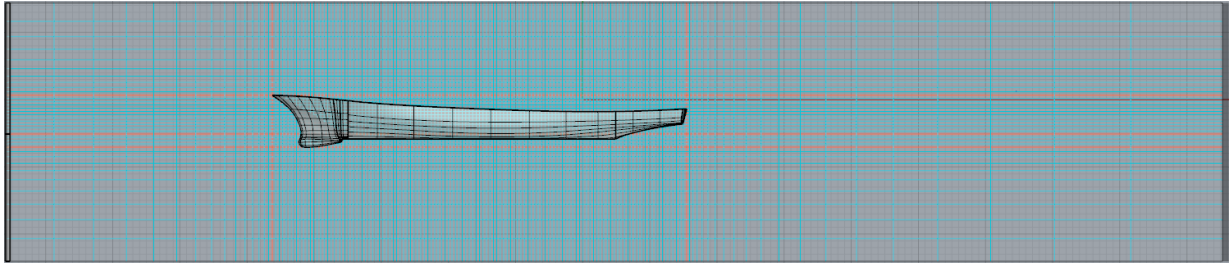


Figure 15: Grid

## Inform

We are now going to enter InForm using the main menu. Inform commands are part of the CHAM developed CFD input language, which enables the user to specify the initial values and sources of variables as well as store our own at specific locations.

Open the main menu and click on models and 'Open InForm Editor'. The window in Figure 16 should appear.

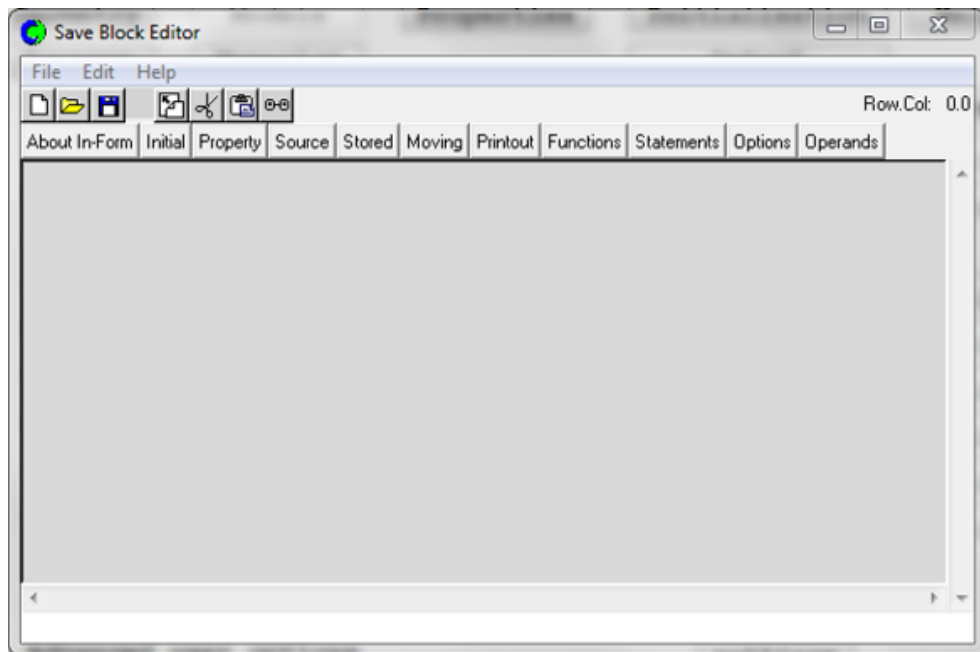


Figure 16: Inform editor

Click on file, create new save block and then double click on save11. This produces two lines in the InForm editor: save11begin and save11end, note that they are indented by two lines. Use the cursor to click at the end of the first line and push return and enter the following:

```

!Set variable values
Real(g, rhoa, rhow, waterh)
g= 9.81 !value of gravity
rhoa = 1.189 !density of air
rhow = 9.9822998E02 !density of water
waterh = 2.663116 !height of water from bottom of domain

!Set initial values
(Initial of p1 is  $g \cdot \text{rhoa} \cdot (\text{zwlast} - \text{zg})$  with if(zg.gt.waterh))
(Initial of p1 is  $g \cdot (\text{rhow} \cdot (\text{waterh} - \text{zg}) + \text{rhoa} \cdot (\text{zwlast} - \text{waterh}))$  with if(zg.le.waterh))
(Initial of den1 is rhoa with if(zg.gt.waterh))
(Initial of den1 is rhow with if(zg.le.waterh))

!Store variables
(Stored of pval is  $\text{den1} \cdot g \cdot (\text{zwlast} - \text{ZG})$  with if(zg.gt.waterh))
(Stored of pval is  $g \cdot (\text{den1} \cdot (\text{waterh} - \text{ZG}) + \text{rhoa} \cdot (\text{zwlast} - \text{waterh}))$  with if(zg.le.waterh))
(Stored of pdif is pval - p1)

!Set source terms
(Source OF R1 at exit is coval(DEN1,pval) with AREA)
(Source of vfol at exit is coval(onlyms,1/den1))

```

Click the save button or file, save current block.

Looking at the commands, you will see 'waterh' has been given the value of 2.663116. This will be different for each simulation, depending on where the boat and two domains have been placed. To find this value, close the InForm editor and left click on the icon in figure 17.

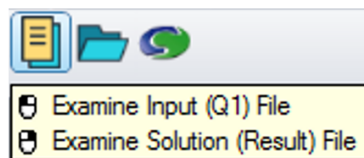


Figure 17: Q1 tool bar button

Scroll down until you see group 24 and the object named Air. Copy the Z position value.

```

> OBJ,      NAME,      Air
> OBJ,      POSITION,   -4.476458E-07, 7.195445E-08, 2.663116E+00
> OBJ,      SIZE,      2.542800E+01, 9.186000E+00, 1.532884E+00
> OBJ,      DOMCLIP,   NO
> OBJ,      GEOMETRY,  cdb2
> OBJ,      GRID,      N,N,N
> OBJ,      TYPE,      USER_DEFINED

```

Figure 18: Z position of object 'air for InForm

Close the Q1 and open the Inform editor once more. We need to view the variables we entered previously. Click on file, open existing save block, double click on save11 and all the information entered earlier should appear. Change the value of waterh to the value just copied from the Q1. Click file, save current save block and then exit the menu. Before running the solver check the settings in the main menu for any errors. Click run earth to start the simulation.

## Results

Click on load results and click ok on the dialog that appears. Choose 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.

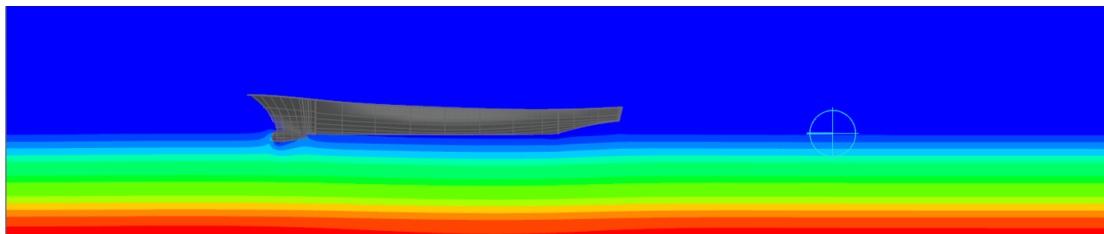


Figure 19: Pressure contour at Y plane

Finally 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 same drop down as previous. Change the variable to VFOL and click in the IsoSurface box, which should say '@probe' to activate the results. The IsoSurface should locate to the exact point between the water and air region. Inspecting the IsoSurface you should see the wave effects forming around the hull. To get a clearer view of the water surface it is useful to use rendering tools.

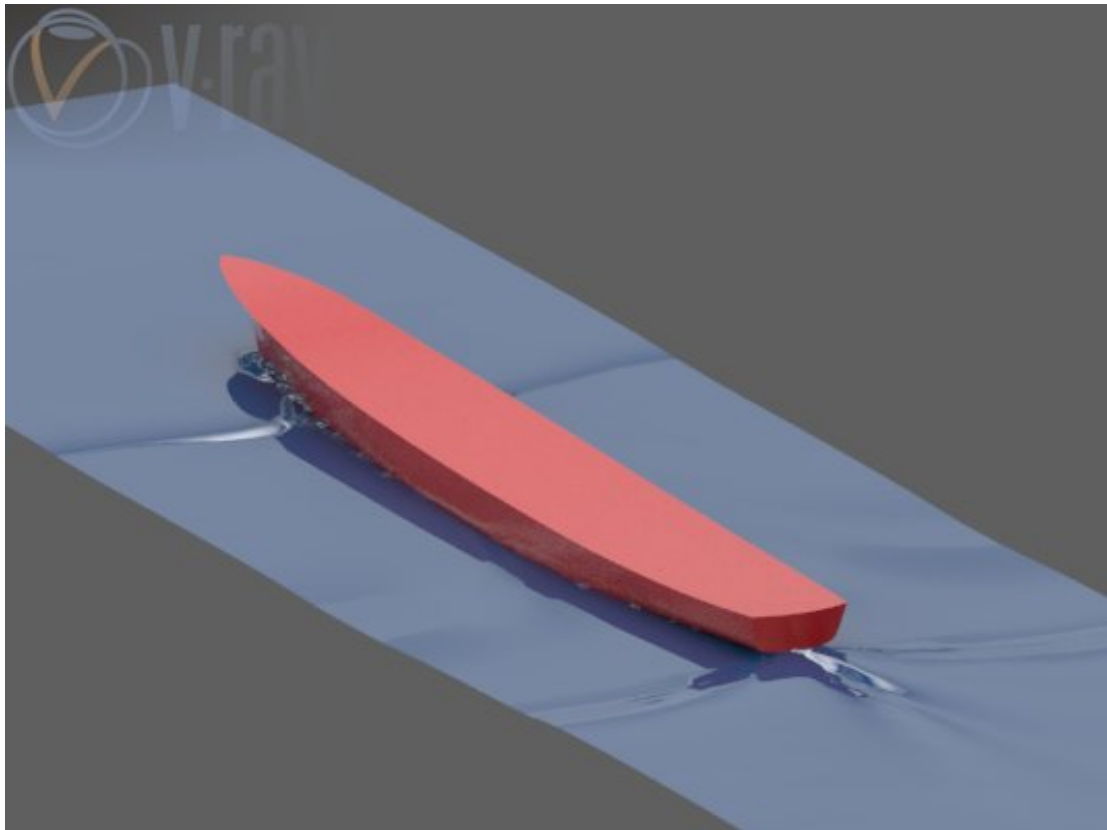


Figure 20: Rendered VFOL Iso Plane Isometric View

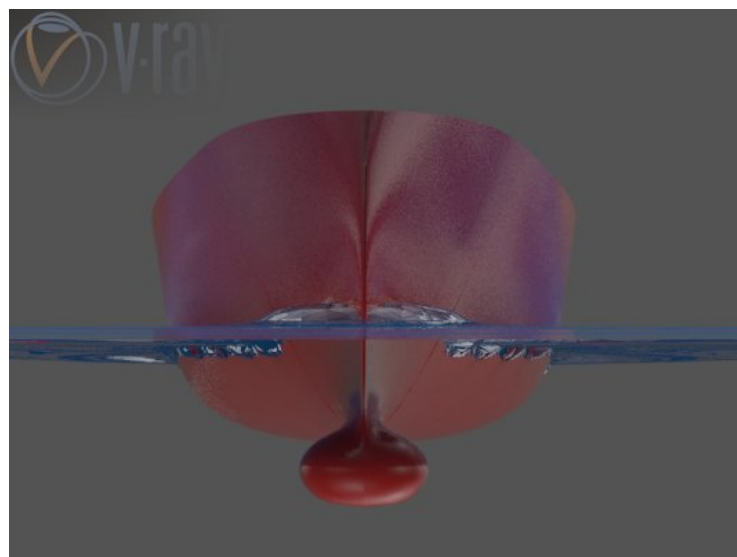



Figure 21: Rendered VFOL Front Iso Plane Front View



To view the results produced from the 'output' menu specified earlier, 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 figures.

#### Integrated force on object: BOAT

```

-----
Fx = 1.083474E+05 (Pressure= 1.083474E+05, Friction= 0.000000E+00)
Fy = -7.926476E+03 (Pressure= -7.926476E+03, Friction= 0.000000E+00)
Fz = 1.845181E+06 (Pressure= 1.845181E+06, Friction= 0.000000E+00)
Pressure force on West side: 1.910970E+05
Pressure force on East side: -8.274967E+04
Pressure force on South side: 4.864483E+05
Pressure force on North side: -4.943748E+05
Pressure force on Low side: 2.070209E+06
Pressure force on High side: -2.250277E+05
Force unit vector: 5.861759E-02 -4.288345E-03 9.982712E-01

Total moment about X axis = -3.192158E+07
Total moment about Y axis = 1.243164E+08
Total moment about Z axis = 2.774966E+06

Moment of Fx about Y axis = -2.448835E+06 at distance Z = 2.260170E+01
Moment of Fy about X axis = -1.407579E+05 at distance Z = 1.775794E+01
Moment of Fz about X axis = -3.178091E+07 at distance Y = 1.722374E+01
Moment of Fx about Z axis = 2.364435E+06 at distance Y = 2.182273E+01
Moment of Fy about Z axis = 4.105201E+05 at distance X = 5.179100E+01
Moment of Fz about Y axis = 1.267649E+08 at distance X = 6.870052E+01
Total exposed vertical area = 4.052099E+02
Sum vert. area with p1 > 10.00 = 4.052099E+02
Ratio area p1 > 10.00/Totalarea =100.00 %

```

Figure 22: Forces on Boat

**Integrated forces for all included objects**


---

```

Fx = 1.083474E+05
Fy = -7.926476E+03
Fz = 1.845181E+06
Ftot= 1.848376E+06
    
```

Force unit vector: 5.861759E-02 -4.288345E-03 9.982712E-01

The total force acts at (centre of pressure):

```

X = 6.870052E+01
Y = 1.722374E+01
Z = 2.260170E+01
    
```

Moments about origin:

```

Total moment about X axis = -3.192158E+07
Total moment about Y axis = 1.243164E+08
Total moment about Z axis = 2.774966E+06
    
```

```

Moment of Fx about Y axis = -2.448835E+06 at distance Z = 2.260170E+01
Moment of Fy about X axis = -1.407579E+05 at distance Z = 2.260170E+01
Moment of Fz about X axis = -3.178091E+07 at distance Y = 1.722374E+01
Moment of Fx about Z axis = 2.364435E+06 at distance Y = 1.722374E+01
Moment of Fy about Z axis = 4.105201E+05 at distance X = 6.870052E+01
Moment of Fz about Y axis = 1.267649E+08 at distance X = 6.870052E+01
    
```

Normalisation areas:

```

AREAx = 5.050000E+01
AREAy = 3.215074E+02
AREAz = 4.300000E+02
    
```

Reference density = 9.980000E+02, Reference velocity = 1.000000E+01  
 Free-stream dynamic head = 4.990000E+04

Drag coefficients based on total forces:

```

(Cd = Force / (Dynamic head * Normalisation area)
Cdx = 4.299584E-02
Cdy = 4.940702E-04
Cdz = 8.599437E-02
    
```

```

Total exposed vertical area = 4.052099E+02
Sum vert. area with P1 > 10.00 = 4.052099E+02
Ratio area P1 > 10.00/Totalarea = 100.00 %
    
```

Figure 23: Forces on all Objects

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	CDx	CDy	CDz
250	1.83E+06	1.13E+05	-3.52E+04	1.83E+06	-3.22E+07	1.26E+08	4.42E+06	7.02E+01	1.72E+01	1.77E+01	4.48E-02	-2.19E-03	8.52E-02
500	2.43E+06	1.40E+05	-1.36E+04	2.43E+06	-4.21E+07	1.66E+08	3.35E+06	6.93E+01	1.72E+01	1.79E+01	5.54E-02	-8.47E-04	1.13E-01
750	2.63E+06	1.36E+05	-2.66E+03	2.62E+06	-4.52E+07	1.81E+08	2.32E+06	7.00E+01	1.72E+01	1.79E+01	5.38E-02	-1.66E-04	1.22E-01
1000	2.70E+06	1.33E+05	-7.70E+03	2.69E+06	-4.66E+07	1.87E+08	2.70E+06	7.02E+01	1.72E+01	1.79E+01	5.29E-02	-4.80E-04	1.26E-01
1250	2.64E+06	1.40E+05	-9.74E+03	2.64E+06	-4.56E+07	1.81E+08	2.97E+06	6.98E+01	1.72E+01	1.79E+01	5.55E-02	-6.07E-04	1.23E-01
1500	2.55E+06	1.45E+05	-9.76E+03	2.55E+06	-4.41E+07	1.74E+08	3.05E+06	6.94E+01	1.72E+01	1.79E+01	5.76E-02	-6.09E-04	1.19E-01
1750	2.48E+06	1.48E+05	-9.15E+03	2.48E+06	-4.29E+07	1.69E+08	3.06E+06	6.91E+01	1.72E+01	1.79E+01	5.89E-02	-5.70E-04	1.15E-01
2000	2.42E+06	1.50E+05	-8.68E+03	2.41E+06	-4.17E+07	1.64E+08	3.04E+06	6.89E+01	1.72E+01	1.79E+01	5.95E-02	-5.41E-04	1.12E-01
2250	2.35E+06	1.51E+05	-8.33E+03	2.35E+06	-4.06E+07	1.59E+08	3.03E+06	6.88E+01	1.72E+01	1.79E+01	5.99E-02	-5.19E-04	1.09E-01
2500	2.29E+06	1.51E+05	-8.21E+03	2.29E+06	-3.96E+07	1.54E+08	3.03E+06	6.87E+01	1.72E+01	1.79E+01	6.00E-02	-5.12E-04	1.07E-01
2750	2.23E+06	1.51E+05	-8.17E+03	2.23E+06	-3.85E+07	1.50E+08	3.02E+06	6.86E+01	1.72E+01	1.79E+01	5.98E-02	-5.09E-04	1.04E-01
3000	2.18E+06	1.50E+05	-8.23E+03	2.17E+06	-3.75E+07	1.46E+08	3.01E+06	6.85E+01	1.72E+01	1.79E+01	5.95E-02	-5.13E-04	1.01E-01
3250	2.12E+06	1.49E+05	-8.35E+03	2.12E+06	-3.66E+07	1.42E+08	3.00E+06	6.85E+01	1.72E+01	1.79E+01	5.91E-02	-5.20E-04	9.86E-02
3500	2.07E+06	1.48E+05	-8.34E+03	2.06E+06	-3.57E+07	1.39E+08	2.98E+06	6.85E+01	1.72E+01	1.79E+01	5.87E-02	-5.20E-04	9.62E-02
3750	2.02E+06	1.46E+05	-8.37E+03	2.02E+06	-3.49E+07	1.36E+08	2.95E+06	6.85E+01	1.72E+01	1.79E+01	5.80E-02	-5.22E-04	9.41E-02
4000	1.98E+06	1.44E+05	-8.38E+03	1.98E+06	-3.42E+07	1.33E+08	2.92E+06	6.85E+01	1.72E+01	1.79E+01	5.73E-02	-5.22E-04	9.21E-02
4250	1.94E+06	1.42E+05	-8.22E+03	1.94E+06	-3.35E+07	1.30E+08	2.88E+06	6.86E+01	1.72E+01	1.79E+01	5.65E-02	-5.12E-04	9.03E-02
4500	1.91E+06	1.41E+05	-7.75E+03	1.90E+06	-3.29E+07	1.28E+08	2.82E+06	6.86E+01	1.72E+01	1.79E+01	5.58E-02	-4.83E-04	8.87E-02
4750	1.88E+06	1.39E+05	-7.63E+03	1.87E+06	-3.24E+07	1.26E+08	2.78E+06	6.87E+01	1.72E+01	1.78E+01	5.51E-02	-4.76E-04	8.72E-02
5000	1.85E+06	1.08E+05	-7.93E+03	1.85E+06	-3.19E+07	1.24E+08	2.77E+06	6.87E+01	2.18E+01	2.26E+01	4.30E-02	-4.94E-04	8.60E-02

Figure 24: Forces on all Objects

# Appendices

## RhinoCFD Lite Guidelines

### Geometry

RhinoCFD Lite is limited to  $40 \times 40 \times 40$  cells and 1000 time sweeps, so the geometry used must be significantly simpler to allow the simulation to completed within these limits. The limit on cells means that sharp edges are not advisable, and simple shape similar to the one shown in the following figure should be used.

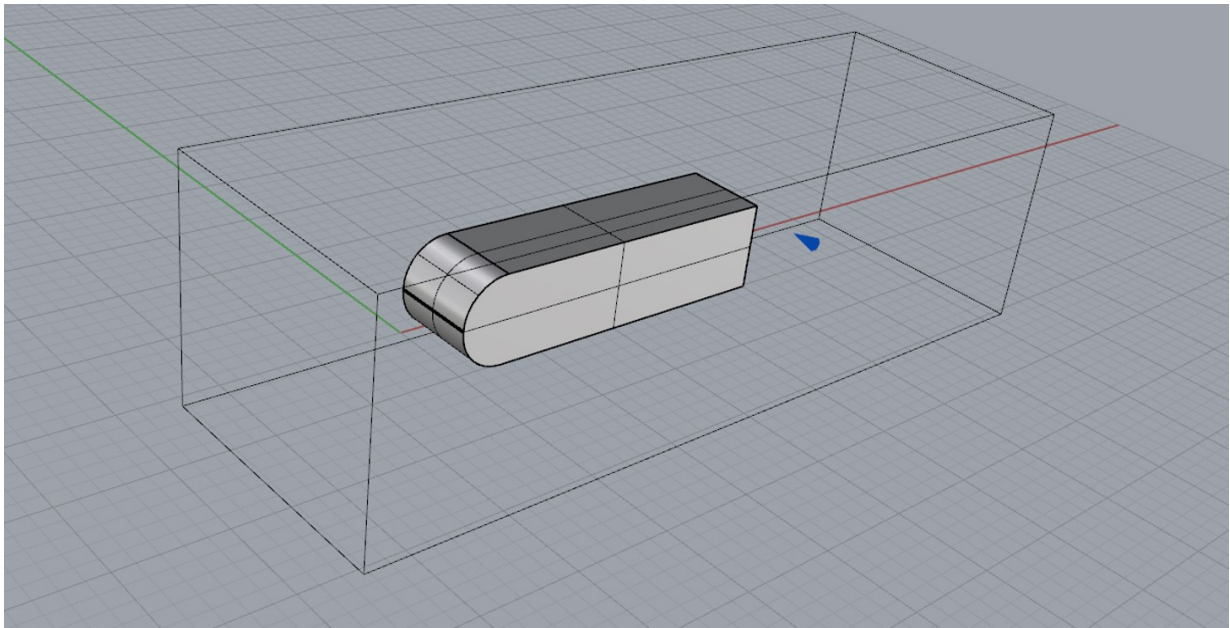


Figure 25: Example Geometry for using HOL in RhinoCFD Lite

The above figure shows a  $20 \times 5 \times 5$  block, which has been curved at the front end using the chamfer command in Rhino.

### Meshing

Make use of all the cells available in RhinoCFD lite, by using the  $40 \times 40 \times 40$  limit. The process for generating the mesh should be the same as explained previously. The following table shows the recommended distribution of cells for the geometry used.

Axis	Region			
	1	2	3	4
X	6	24	10	-
Y	10	20	10	-
Z	9	7	15	9

Again all first and last regions have a geometric power law of -1.2 and 1.2 respectively. The following figures show the grid around the object:

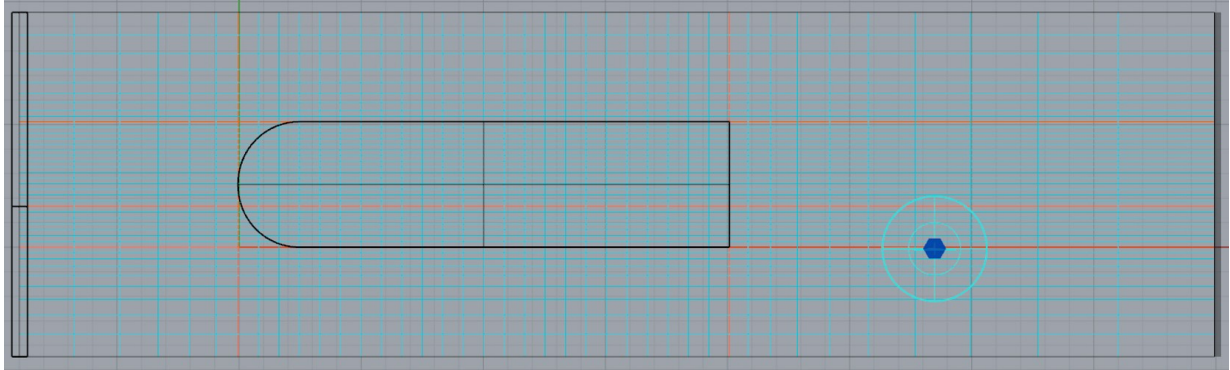


Figure 26: Example Grid x,z view

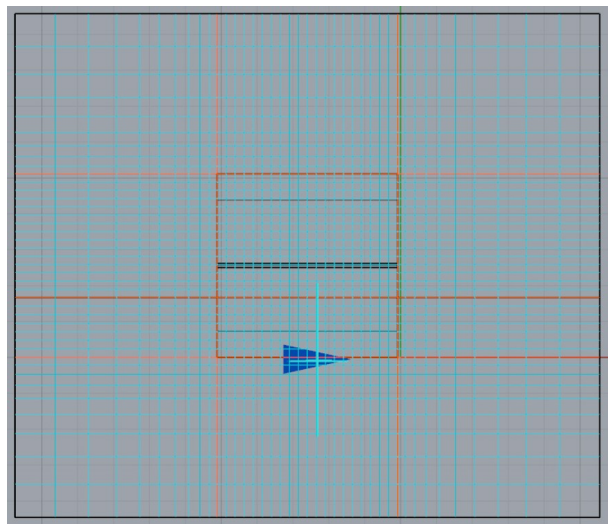


Figure 27: Example Grid y,z view

## Results

Using the same method as for RhinoCFD the results can be viewed:

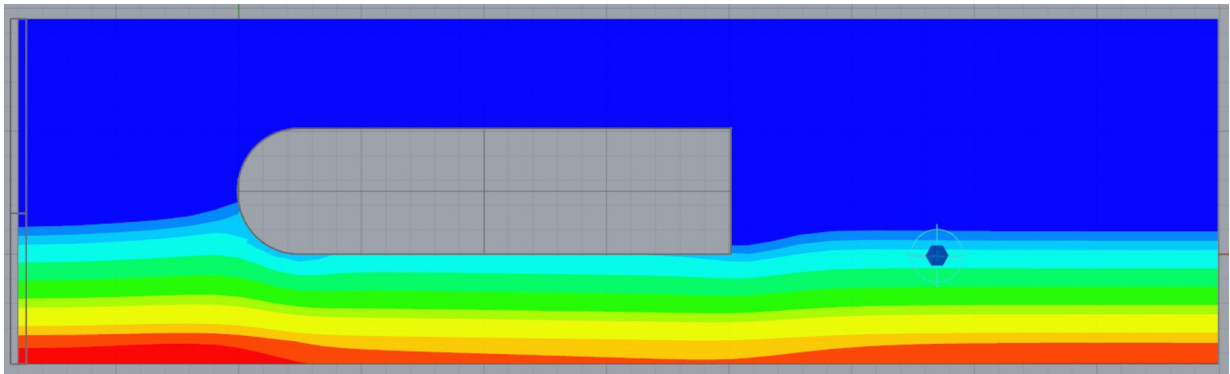


Figure 28: Pressure

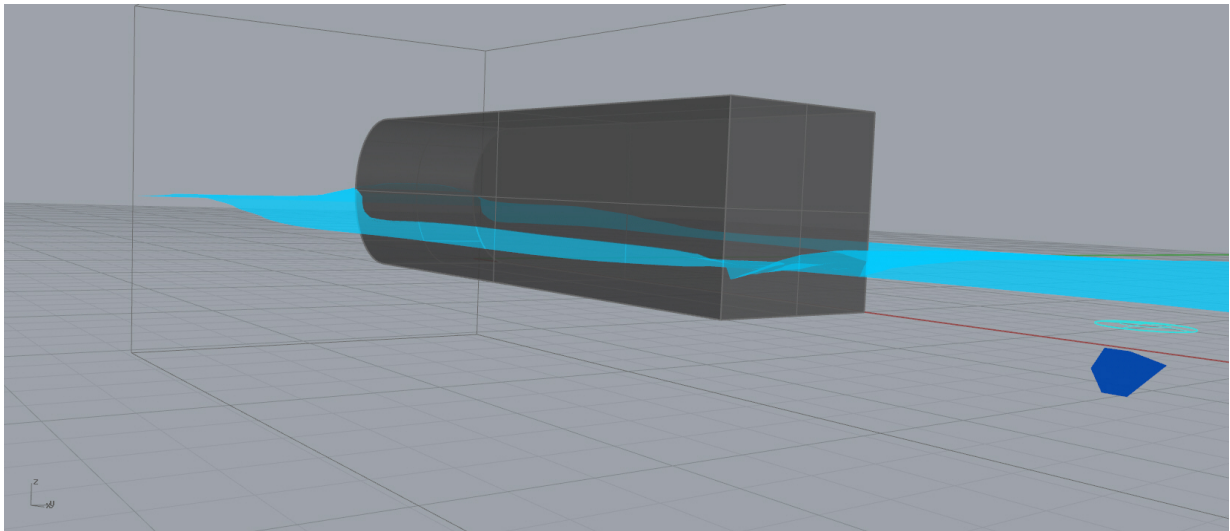


Figure 29: Iso Surface Volume Fraction



---

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