RhinoCFD Tutorial

Free Surface - Scalar Equation Model

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 Scalar Equation free surface model (SEM) correctly and provide a glimpse in outputting forces and the more-advanced topic of 'InForm'. This tutorial assumes that you already understand how to set up a transient simulation. See the RhinoCFD basics Transient videos for more information.

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, but care must be taken.

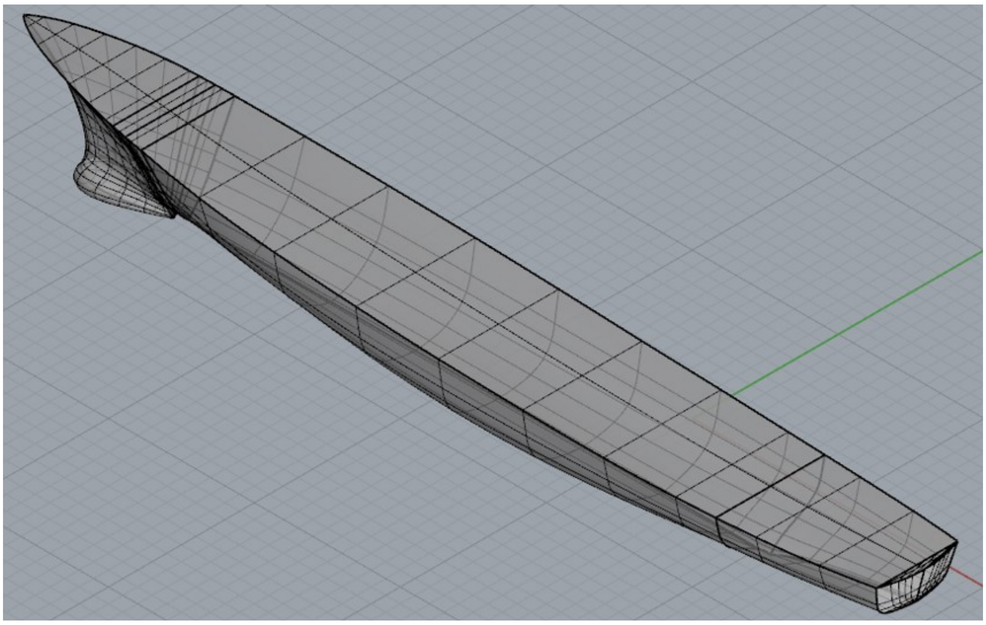

Figure 1: Boat gemoetry

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 SEM which is a more precise implimentation of the free surface model than HOL, and can be used on more complex geometry and results can be viewed for a range of time steps, rather than for just one steady state solution. However this method is much more computational expensive and simulations take much longer to run.

CFD Analysis

Main Menu

First, create a domain around the boat hull, by clicking on the first button on the toolbar. Select the desired working directory.

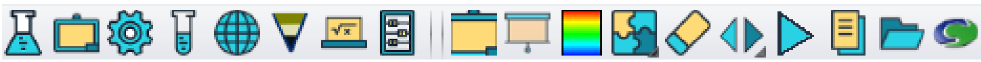

Figure 2: RhinoCFD tool bar

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.

Next set up a transient case that will run for 30 seconds and 3000 steps. Each step should be around 30 iterations and should dump results every 5 steps. The initial velocity should be 10m/s.

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

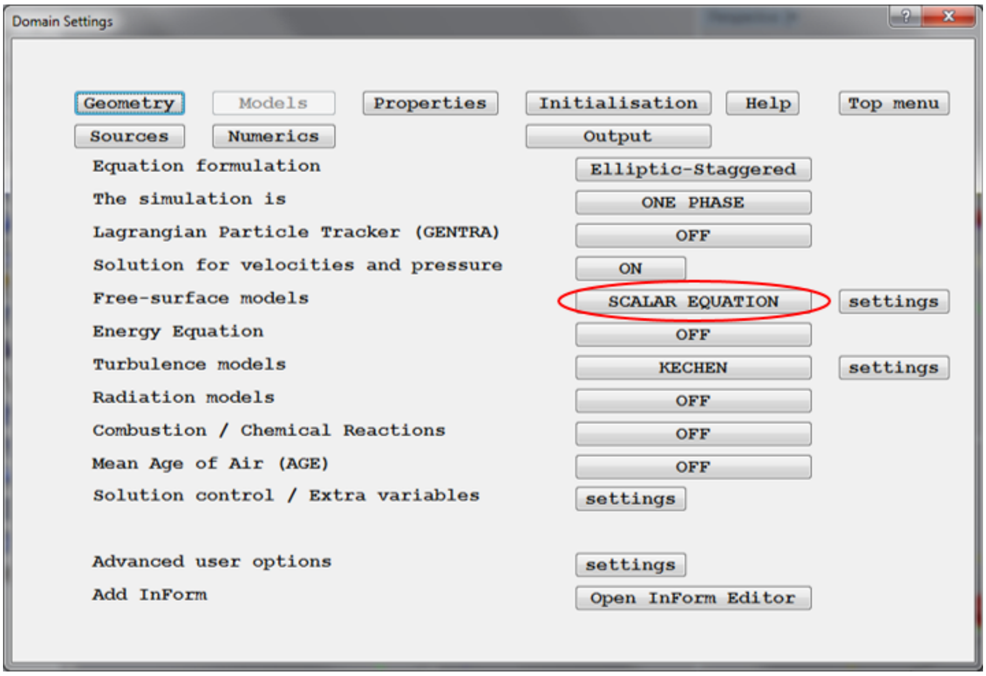

Figure 3: Models settings

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.

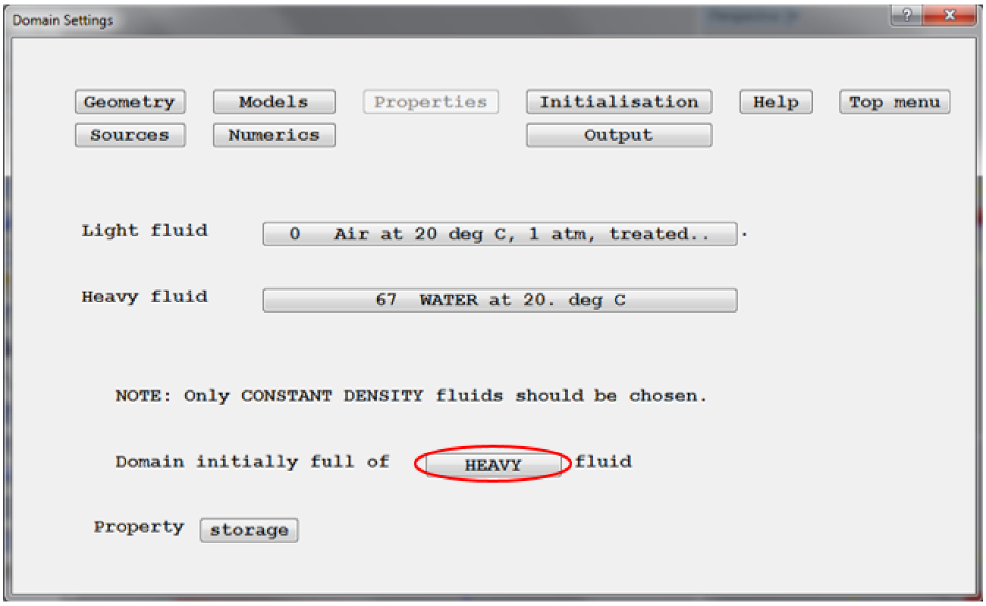

Figure 4: Properties settings

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.

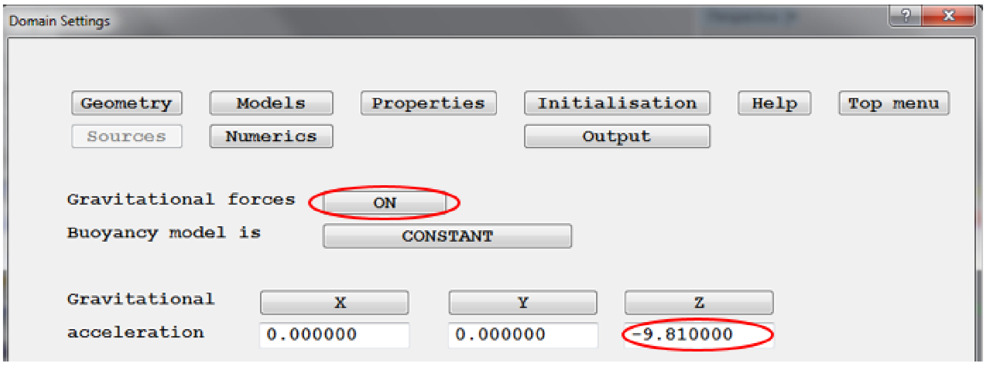

Figure 5: Sources settings

In the Numerics tab set 'Total number of iterations' to 30, 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 video for further information.

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

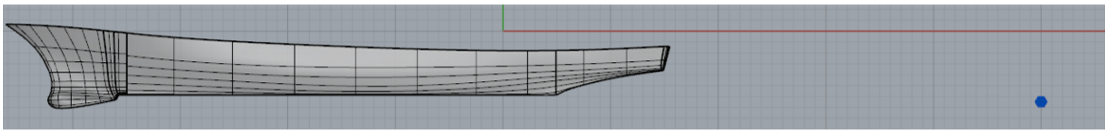

Figure 6: Probe location

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.

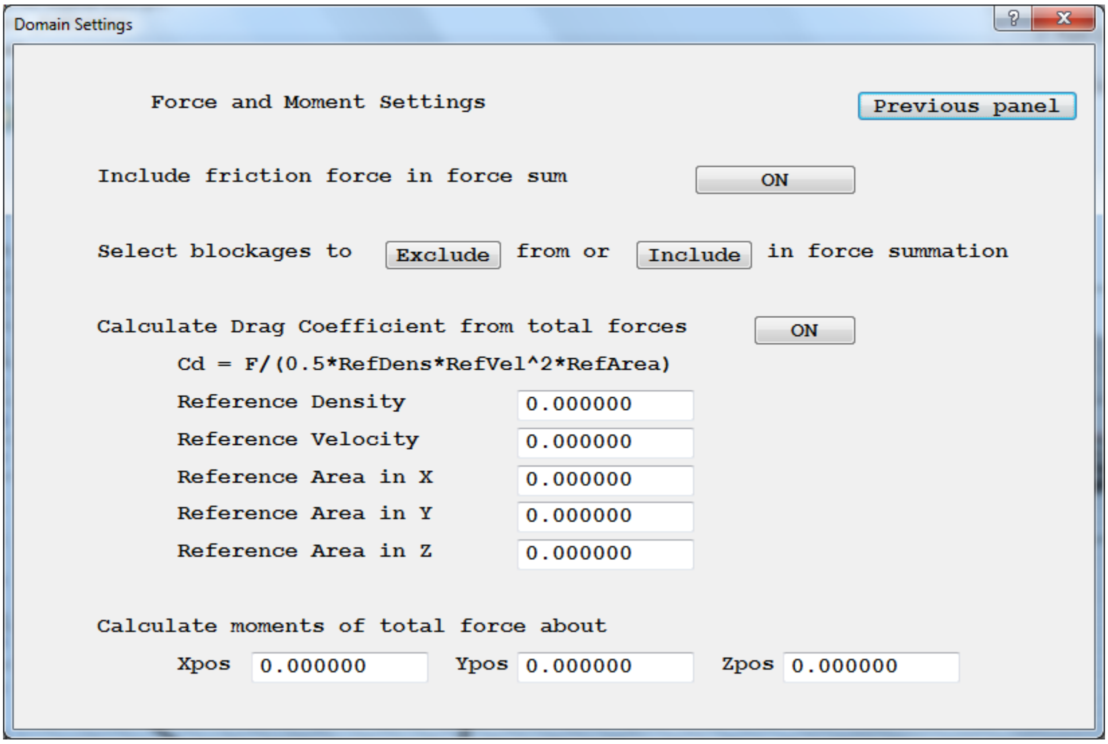

Figure 7: Output Forces Menu

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.

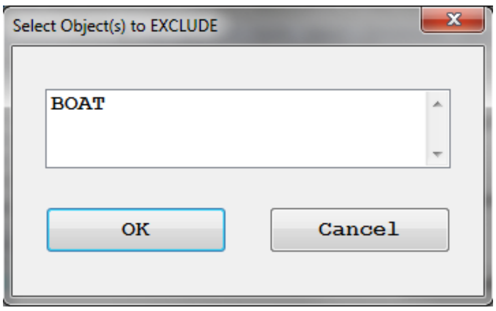

Figure 8: Exclude Dialog

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

Table 1: Input values for Drag calculation

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

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.

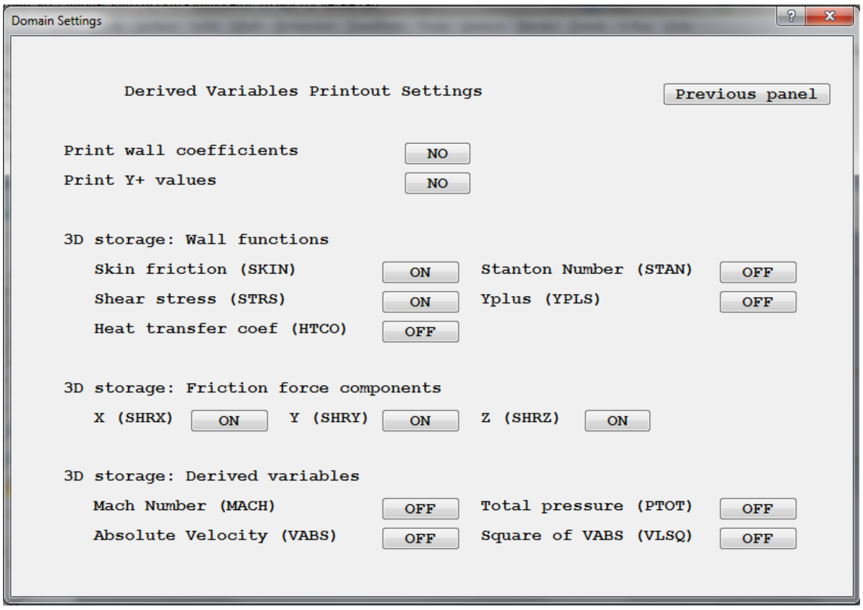

Figure 9: Derived variables Menu

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.

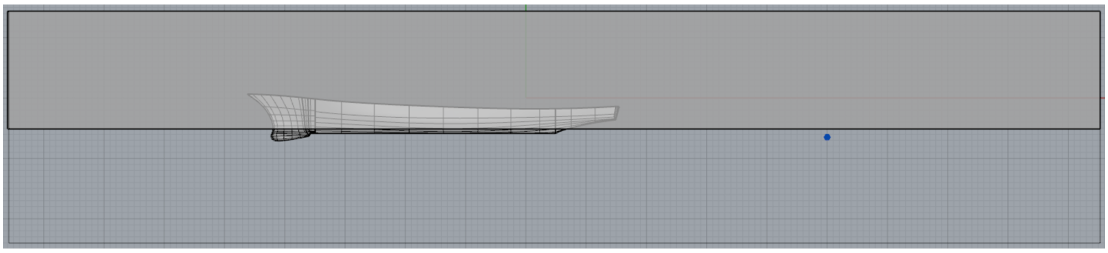

Figure 10: Fluid boundaries around boat

Edit the new cube and change the name to ‘air’ then set the type as ‘user defined’. Click on attributes and create a new patch name by the typing in ‘air’ into the new box. Click out of the box and then on to the blank ‘patch number’ box, which will bring up the newly created patch. Change the type by typing in ’INIVAL’ and then change the coefficient and value of PRPS, VFOL and SURN to 0.0. Click OK to exit attributes, and then click apply and then OK.

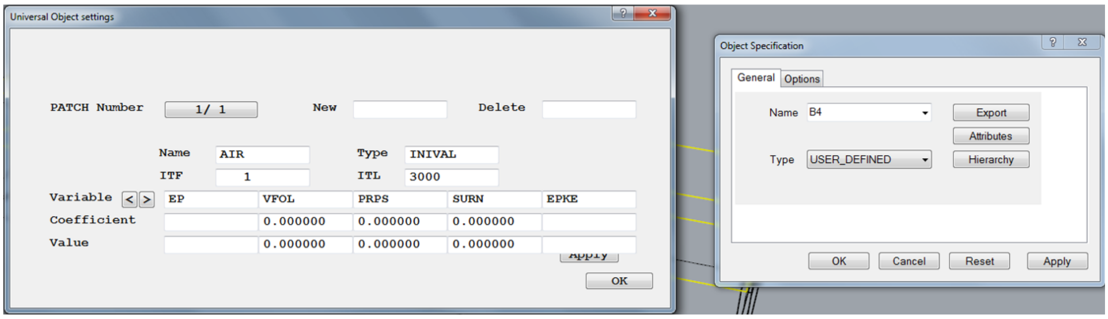

Figure 11: User defined object settings

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 12 should appear.

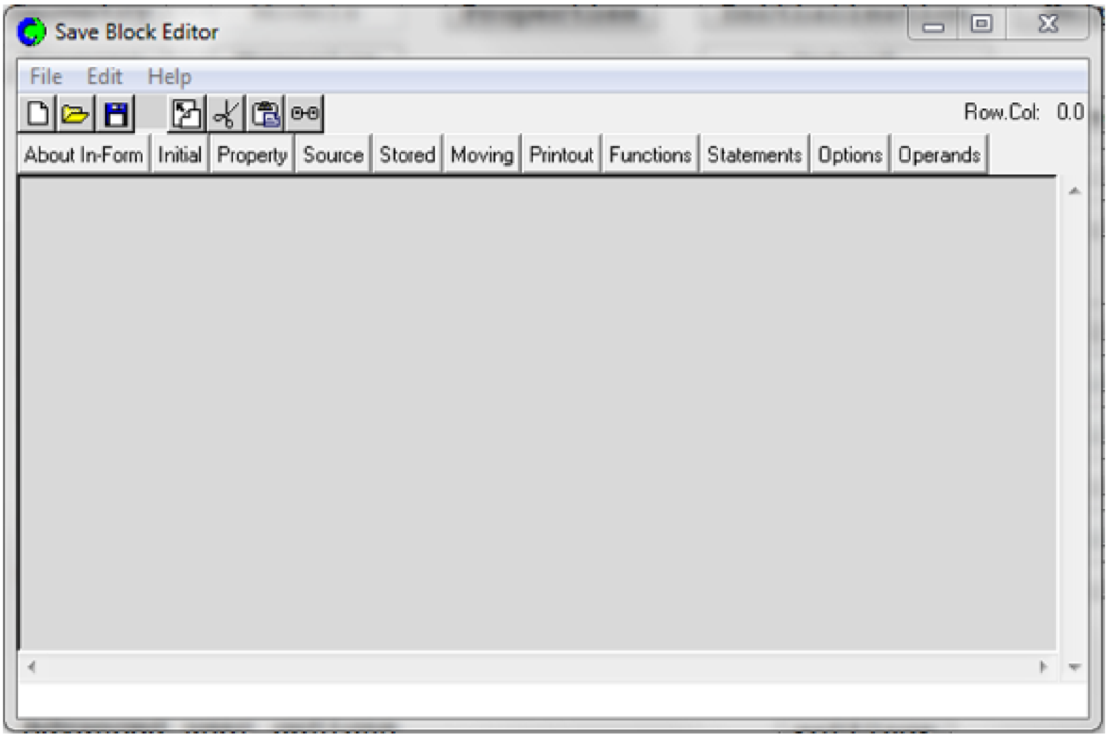

Figure 12: Inform editor

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*rhoa*(zwlast - zg) with if(zg.gt.waterh))
(Initial of p1 is g*(rhow*(waterh - zg) + rhoa*(zwlast - 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 den1*g*(zwlast - ZG) with if(zg.gt.waterh))
(Stored of pval is g*(den1*(waterh - ZG) + rhoa*(zwlast - 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 highlighted in figure 13.

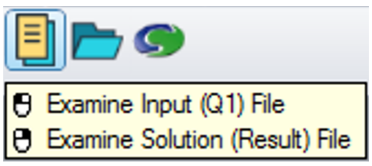

Figure 13: Q1 tool bar button

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

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.

```
> 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 14: Z position of object air for InForm

Before running the solver refine your mesh and check the settings in the menu for any errors.

Boundary Conditions

Finally, we need to create two inlets and an outlet. Create a cube normal to the x axis in the lower region. Edit the attributes, enter an inlet velocity of 10m/s and select 'heavy' as the density. Do the same for the top region and select 'light' as the density.Right click on the second icon on the toolbar to enter the 'domain faces menu and make make xmax open = yes, then exit the menu. It is now essential you change the name of the outlet to what is specified in the InForm equations. Select the outlet and left click on the third button in the toolbar and change the name from `DOM_XMAX_O` to exit enter the attributes and ensure that external fluid is set to 'Heavy' and click ok and exit the menu.

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 toolbar 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 the grid, which can be used as guide lines for different sizes of domain and geometry.

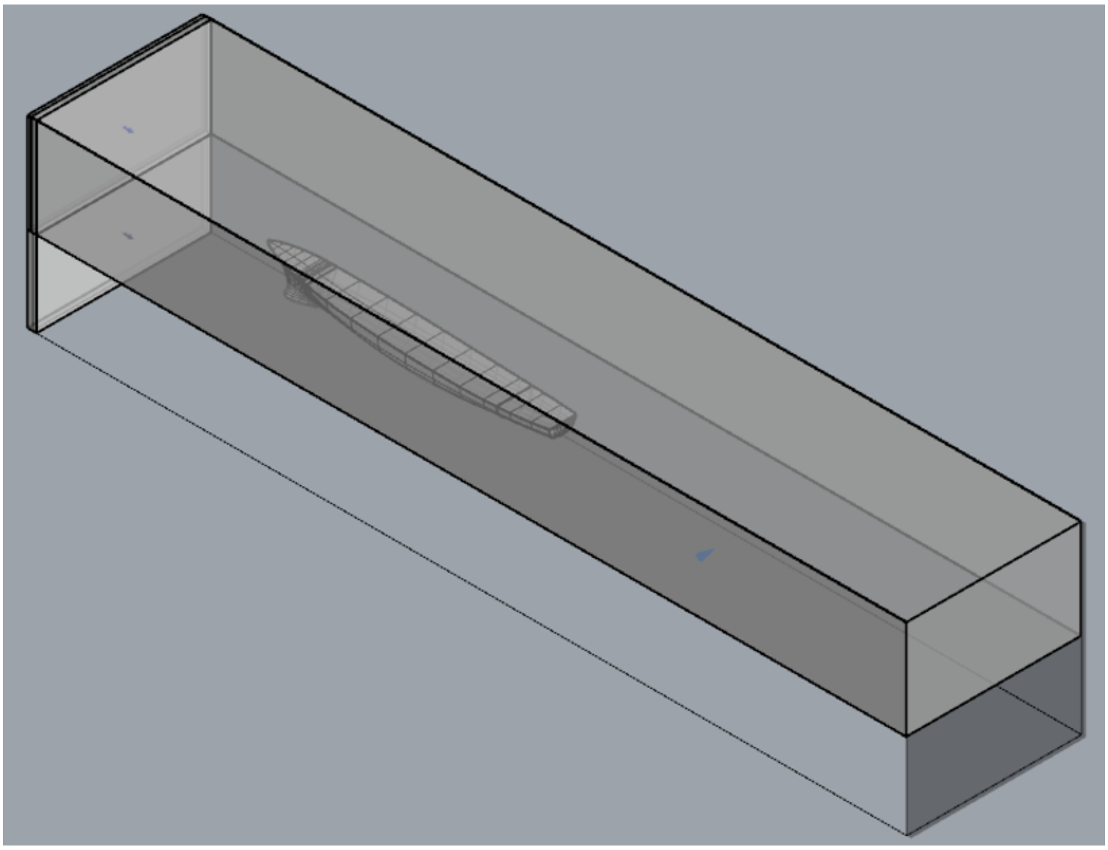

Figure 15: Fluid boundaries

Axis	**Region**				**Average cell**
	1	**2**	**3**	**4**	**size (m)**
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 setting the grid should look like this:

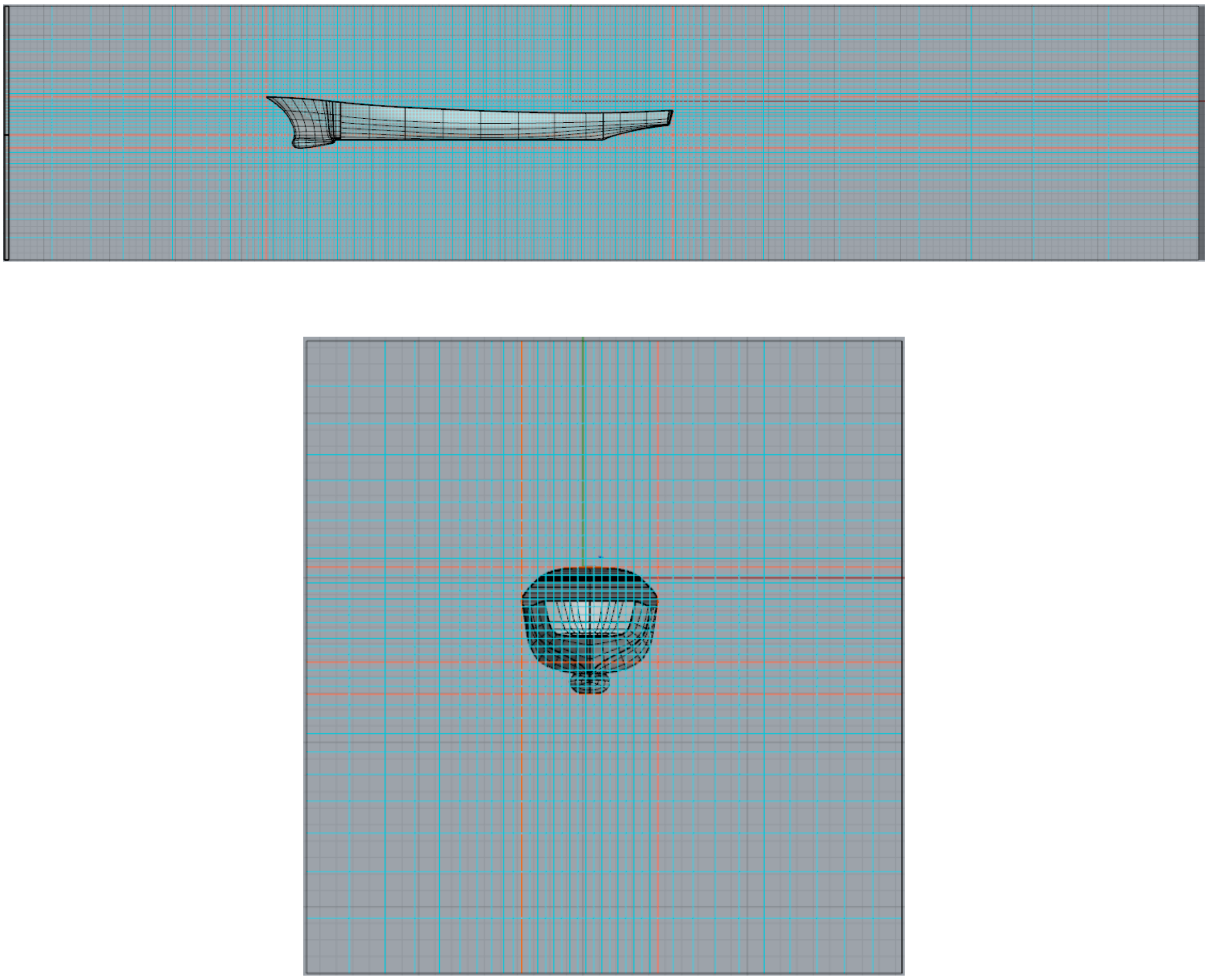

Figure 16: Grid

Results

On the tool bar click 'Load Results' and click OK. Rotate the cutplane 90 degrees so that it is in the Y plane and select P1 (Pressure). You should see a distribution in pressure in the water region.

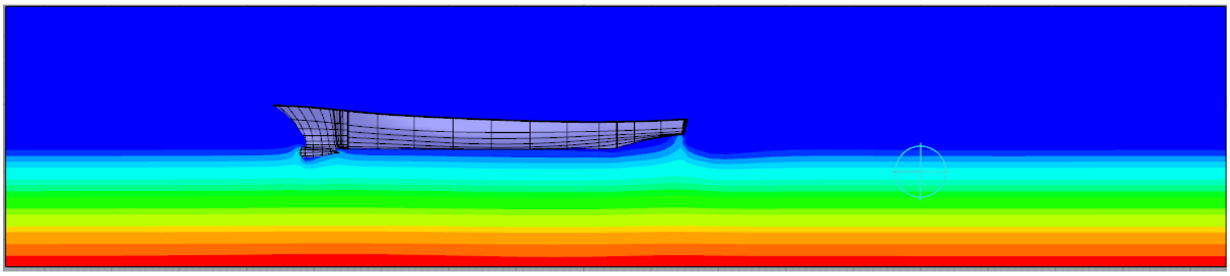

Figure 17: Pressure contour in Y plane

To view the effect the hull has on the water surface, hide the cutplane and create an IsoSurface. Select VFOL as the variable and deselect the tick box checked nexted to '@probe' and type in 0.5 to initialise the results. You will initially see the resulting flow field at the final step. Click on 'play time series' drop down under the time step button and the results for all time steps will be played.

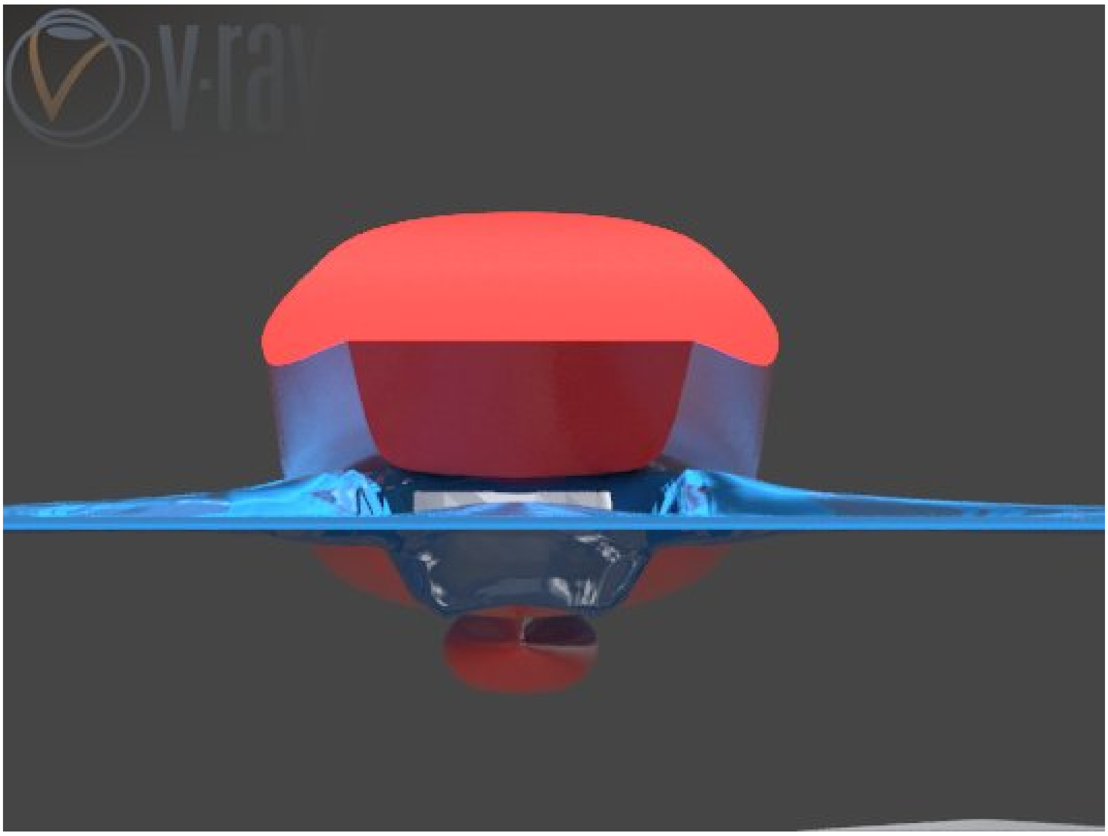

Figure 18: Redered VFOL Iso plane

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 reach text which will structured as in the following figures:

```
Integrated force on object: BOAT
------------------------------------
 Fx  =  8.926287E+04 (Pressure=  8.926287E+04, Friction=  0.000000E+00)
 Fy  = -6.167780E+03 (Pressure= -6.167780E+03, Friction=  0.000000E+00)
 Fz  =  2.737188E+06 (Pressure=  2.737188E+06, Friction=  0.000000E+00)
Pressure force on West side:  2.485233E+05
Pressure force on East side: -1.592605E+05
Pressure force on South side: 8.708944E+05
Pressure force on North side:-8.770620E+05
Pressure force on Low side:   3.283046E+06
Pressure force on High side: -5.458592E+05
Force unit vector:   3.259375E-02 -2.252125E-03  9.994662E-01

 Total moment about X axis = -4.726012E+07
 Total moment about Y axis =  1.940640E+08
 Total moment about Z axis =  2.398844E+06

 Moment of Fx about Y axis = -2.164704E+06 at distance Z =  2.425089E+01
 Moment of Fy about X axis = -1.092972E+05 at distance Z =  1.772066E+01
 Moment of Fz about X axis = -4.715099E+07 at distance Y =  1.722607E+01
 Moment of Fx about Z axis =  2.100576E+06 at distance Y =  2.353247E+01
 Moment of Fy about Z axis =  2.982476E+05 at distance X =  4.835574E+01
 Moment of Fz about Y axis =  1.962297E+08 at distance X =  7.169027E+01
 Total exposed vertical area     =  4.052092E+02
 Sum vert. area with P1 > 10.00 =  4.051857E+02
 Ratio area P1> 10.00/Totalarea = 99.99 %
```

Figure 19: Forces on Boat

```
Integrated forces for all included objects
------------------------------------------
 Fx   =  8.926287E+04
 Fy   = -6.167780E+03
 Fz   =  2.737188E+06
 Ftot=   2.738650E+06

Force unit vector:   3.259375E-02 -2.252125E-03  9.994662E-01

The total force acts at (centre of pressure):
 X    =  7.169027E+01
 Y    =  1.722607E+01
 Z    =  2.425089E+01

Moments about origin:
 Total moment about X axis = -4.726012E+07
 Total moment about Y axis =  1.940640E+08
 Total moment about Z axis =  2.398844E+06

 Moment of Fx about Y axis = -2.164704E+06 at distance Z =  2.425089E+01
 Moment of Fy about X axis = -1.092972E+05 at distance Z =  2.425089E+01
 Moment of Fz about X axis = -4.715099E+07 at distance Y =  1.722607E+01
 Moment of Fx about Z axis =  2.100576E+06 at distance Y =  1.722607E+01
 Moment of Fy about Z axis =  2.982476E+05 at distance X =  7.169027E+01
 Moment of Fz about Y axis =  1.962297E+08 at distance X =  7.169027E+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 =   3.542247E-02
 Cdy =   3.844478E-04
 Cdz =   1.275662E-01

 Total exposed vertical area    =  4.052092E+02
 Sum vert. area with P1 > 10.00 =  4.051857E+02
 Ratio area P1> 10.00/Totalarea = 99.99 %
```

Figure 20: 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 for each step and time.

TIME	ISTEP	FT	FX	FY	FZ	MOMX	MOMY	MOMZ	XBAR	YBAR	ZBAR	CDx	CDy	CDz
5.00E-02	5	2.65E+06	1.30E+05	-3.30E+03	2.65E+06	-4.57E+07	1.84E+08	2.40E+06	7.03E+01	1.72E+01	1.78E+01	5.15E-02	-2.06E-04	1.23E-01
1.00E-01	10	2.63E+06	1.32E+05	-5.75E+03	2.63E+06	-4.54E+07	1.82E+08	2.56E+06	7.03E+01	1.72E+01	1.78E+01	5.25E-02	-3.58E-04	1.23E-01
1.50E-01	15	2.62E+06	1.36E+05	-7.50E+03	2.62E+06	-4.52E+07	1.81E+08	2.71E+06	7.01E+01	1.72E+01	1.78E+01	5.40E-02	-4.67E-04	1.22E-01
2.00E-01	20	2.61E+06	1.39E+05	-8.91E+03	2.61E+06	-4.51E+07	1.80E+08	2.82E+06	7.01E+01	1.72E+01	1.78E+01	5.51E-02	-5.55E-04	1.22E-01
2.50E-01	25	2.61E+06	1.41E+05	-1.00E+04	2.60E+06	-4.50E+07	1.80E+08	2.91E+06	7.00E+01	1.72E+01	1.78E+01	5.59E-02	-6.23E-04	1.21E-01
3.00E-01	30	2.60E+06	1.42E+05	-1.08E+04	2.60E+06	-4.50E+07	1.79E+08	2.98E+06	7.00E+01	1.72E+01	1.78E+01	5.65E-02	-6.76E-04	1.21E-01
3.50E-01	35	2.60E+06	1.44E+05	-1.15E+04	2.60E+06	-4.49E+07	1.79E+08	3.03E+06	6.99E+01	1.72E+01	1.78E+01	5.70E-02	-7.17E-04	1.21E-01
4.00E-01	40	2.60E+06	1.45E+05	-1.20E+04	2.59E+06	-4.49E+07	1.78E+08	3.07E+06	6.98E+01	1.72E+01	1.79E+01	5.74E-02	-7.46E-04	1.21E-01
4.50E-01	45	2.59E+06	1.46E+05	-1.23E+04	2.59E+06	-4.48E+07	1.78E+08	3.10E+06	6.98E+01	1.72E+01	1.79E+01	5.78E-02	-7.66E-04	1.21E-01
5.00E-01	50	2.59E+06	1.46E+05	-1.24E+04	2.59E+06	-4.48E+07	1.78E+08	3.12E+06	6.97E+01	1.72E+01	1.79E+01	5.81E-02	-7.76E-04	1.21E-01
5.50E-01	55	2.59E+06	1.47E+05	-1.25E+04	2.59E+06	-4.48E+07	1.78E+08	3.13E+06	6.97E+01	1.72E+01	1.79E+01	5.83E-02	-7.78E-04	1.21E-01
6.00E-01	60	2.60E+06	1.47E+05	-1.24E+04	2.59E+06	-4.49E+07	1.78E+08	3.14E+06	6.97E+01	1.72E+01	1.79E+01	5.85E-02	-7.75E-04	1.21E-01
6.50E-01	65	2.60E+06	1.48E+05	-1.23E+04	2.59E+06	-4.49E+07	1.78E+08	3.14E+06	6.96E+01	1.72E+01	1.79E+01	5.86E-02	-7.68E-04	1.21E-01
7.00E-01	70	2.60E+06	1.48E+05	-1.22E+04	2.59E+06	-4.49E+07	1.78E+08	3.14E+06	6.96E+01	1.72E+01	1.79E+01	5.87E-02	-7.59E-04	1.21E-01
7.50E-01	75	2.60E+06	1.48E+05	-1.20E+04	2.59E+06	-4.49E+07	1.78E+08	3.13E+06	6.96E+01	1.72E+01	1.79E+01	5.87E-02	-7.47E-04	1.21E-01
8.00E-01	80	2.59E+06	1.48E+05	-1.18E+04	2.59E+06	-4.48E+07	1.78E+08	3.12E+06	6.96E+01	1.72E+01	1.79E+01	5.88E-02	-7.35E-04	1.21E-01
8.50E-01	85	2.59E+06	1.48E+05	-1.16E+04	2.59E+06	-4.47E+07	1.77E+08	3.12E+06	6.96E+01	1.72E+01	1.79E+01	5.89E-02	-7.24E-04	1.20E-01
9.00E-01	90	2.59E+06	1.49E+05	-1.15E+04	2.58E+06	-4.47E+07	1.77E+08	3.12E+06	6.96E+01	1.72E+01	1.79E+01	5.90E-02	-7.14E-04	1.20E-01
9.50E-01	95	2.58E+06	1.49E+05	-1.13E+04	2.58E+06	-4.46E+07	1.77E+08	3.12E+06	6.96E+01	1.72E+01	1.79E+01	5.92E-02	-7.03E-04	1.20E-01
1.00E+00	100	2.58E+06	1.50E+05	-1.11E+04	2.57E+06	-4.45E+07	1.76E+08	3.12E+06	6.96E+01	1.72E+01	1.79E+01	5.93E-02	-6.91E-04	1.20E-01
1.05E+00	105	2.58E+06	1.49E+05	-1.09E+04	2.57E+06	-4.45E+07	1.76E+08	3.10E+06	6.96E+01	1.72E+01	1.79E+01	5.91E-02	-6.79E-04	1.20E-01
1.10E+00	110	2.58E+06	1.46E+05	-1.07E+04	2.58E+06	-4.46E+07	1.78E+08	3.04E+06	6.98E+01	1.72E+01	1.79E+01	5.79E-02	-6.65E-04	1.20E-01
1.15E+00	115	2.59E+06	1.44E+05	-1.05E+04	2.59E+06	-4.48E+07	1.79E+08	2.98E+06	7.00E+01	1.72E+01	1.79E+01	5.70E-02	-6.52E-04	1.21E-01
1.20E+00	120	2.60E+06	1.41E+05	-1.03E+04	2.59E+06	-4.48E+07	1.79E+08	2.93E+06	7.01E+01	1.72E+01	1.79E+01	5.59E-02	-6.41E-04	1.21E-01
1.25E+00	125	2.58E+06	1.40E+05	-1.02E+04	2.58E+06	-4.46E+07	1.78E+08	2.92E+06	7.01E+01	1.72E+01	1.79E+01	5.57E-02	-6.33E-04	1.20E-01
1.30E+00	130	2.56E+06	1.42E+05	-9.95E+03	2.55E+06	-4.42E+07	1.76E+08	2.94E+06	6.99E+01	1.72E+01	1.79E+01	5.65E-02	-6.20E-04	1.19E-01
1.35E+00	135	2.55E+06	1.44E+05	-9.67E+03	2.54E+06	-4.40E+07	1.75E+08	2.96E+06	6.99E+01	1.72E+01	1.79E+01	5.72E-02	-6.03E-04	1.19E-01
1.40E+00	140	2.55E+06	1.44E+05	-9.36E+03	2.55E+06	-4.40E+07	1.76E+08	2.95E+06	7.00E+01	1.72E+01	1.79E+01	5.73E-02	-5.84E-04	1.19E-01
1.45E+00	145	2.57E+06	1.40E+05	-9.05E+03	2.57E+06	-4.44E+07	1.78E+08	2.86E+06	7.03E+01	1.72E+01	1.79E+01	5.55E-02	-5.64E-04	1.20E-01

Figure 21: 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, a simple shape similar to the one shown in the following figure is recommended.

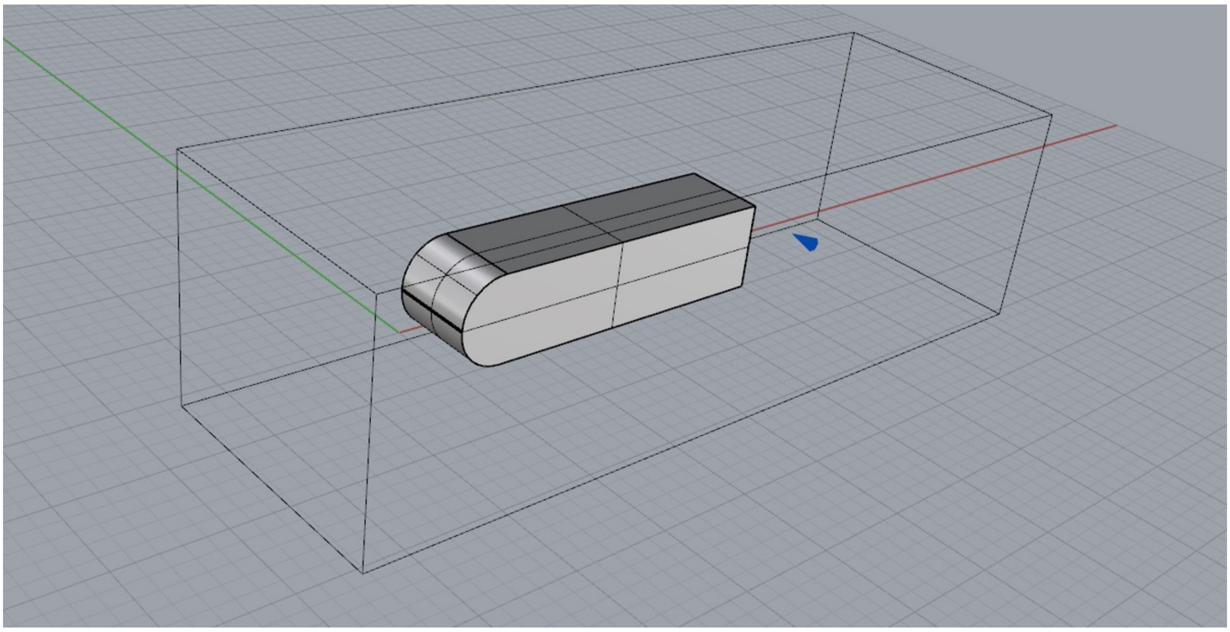

Figure 22: Example Geometry for using HOL in RhinoCFD Lite

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.

Setup

All steps are the same as presented in the main document, however settings will have to be changed to account for the restrictions mentioned previously.
RhinoCFD Lite allows for 1000 sweeps, this is calculated from the total number of iterations $\times$ time steps. You will need around 30 iterations leaving 33 time steps to be completed for 1 second. It is recommend that you dump the solutions every 1-2 time steps to receive more results.
Initialise the velocity to 5m/s and ensure that both inlets have a velocity of 5m/s.

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:

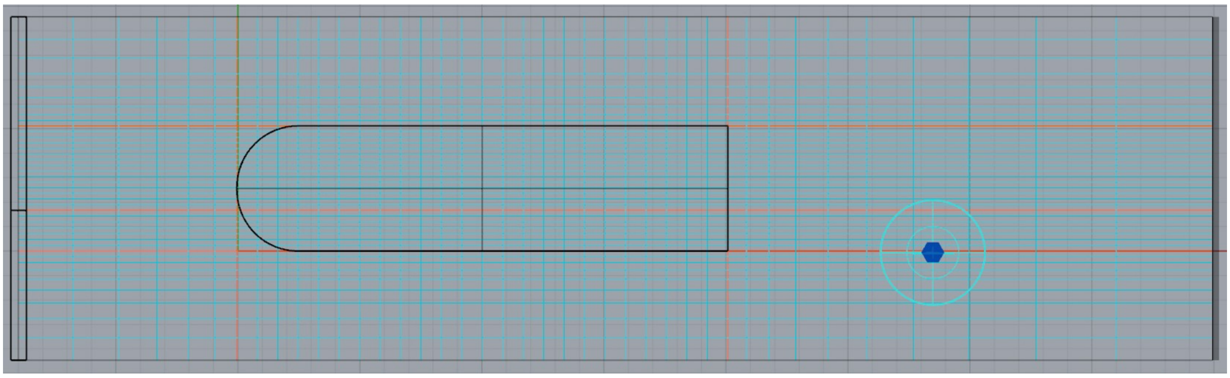

Figure 23: Example Grid x,z view

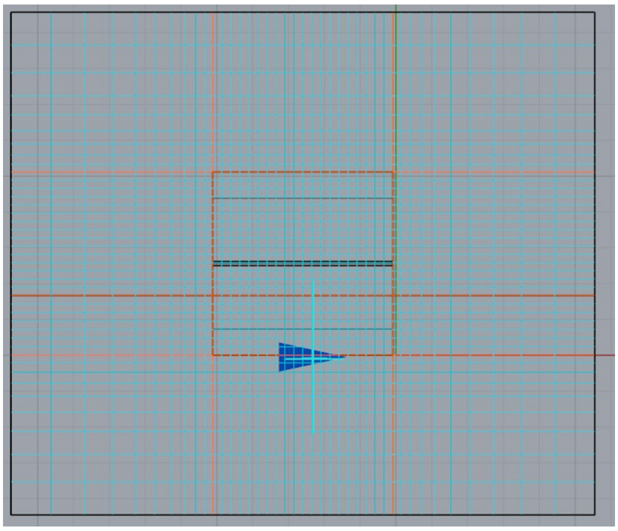

Figure 24: Example Grid y,z view

Results

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

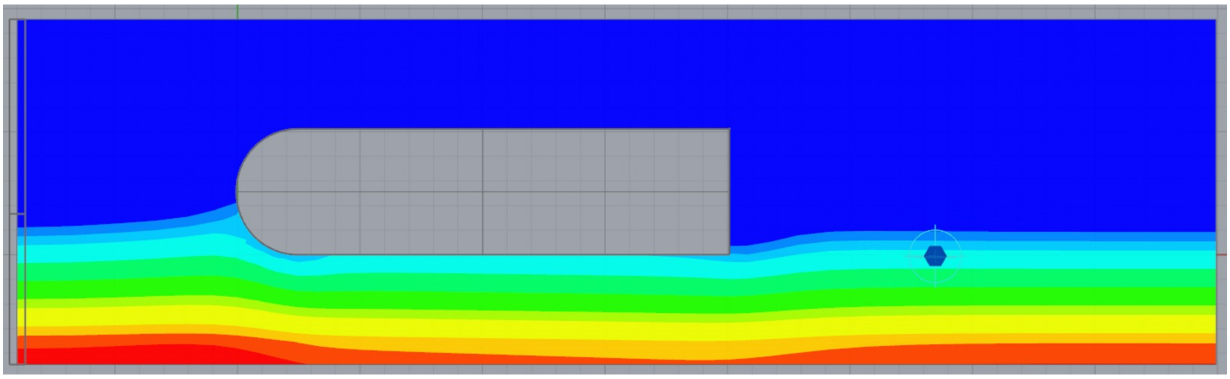

Figure 25: Pressure

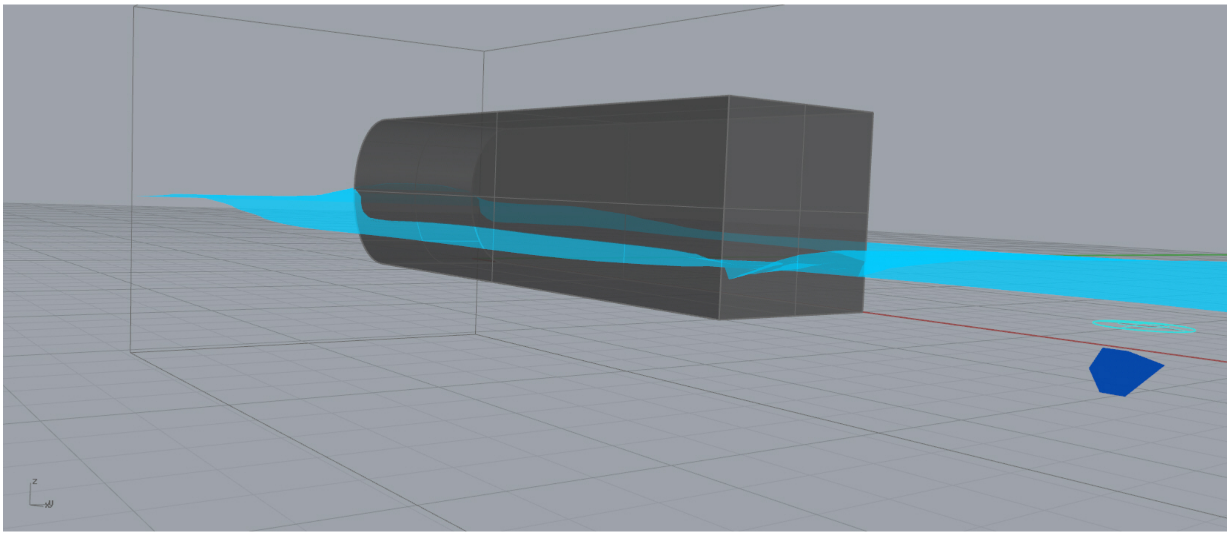

Figure 26: Iso Surface Volume Fraction

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