



---

# RhinoCFD Tutorial

---

Free Surface - Water Flow Past Obstacles

RhinoCFD Official document produced by CHAM  
October 13, 2021

## Introduction

This tutorial demonstrates how to set up a free surface model of flow in a dry channel with obstacles along the flow path. It is also suitable for simulations of e.g. riverbeds. It is designed to get you up to speed on how to apply the Volume Of Fluid (VOF) free surface model correctly. This tutorial assumes that you already understand how to set up a transient simulation. See the RhinoCFD basics [Transient](#) videos for more information.

The geometry required for this tutorial can be set up quickly in Rhino, but users can also use their own geometry or download it from [RhinoCFD Tutorials](#).

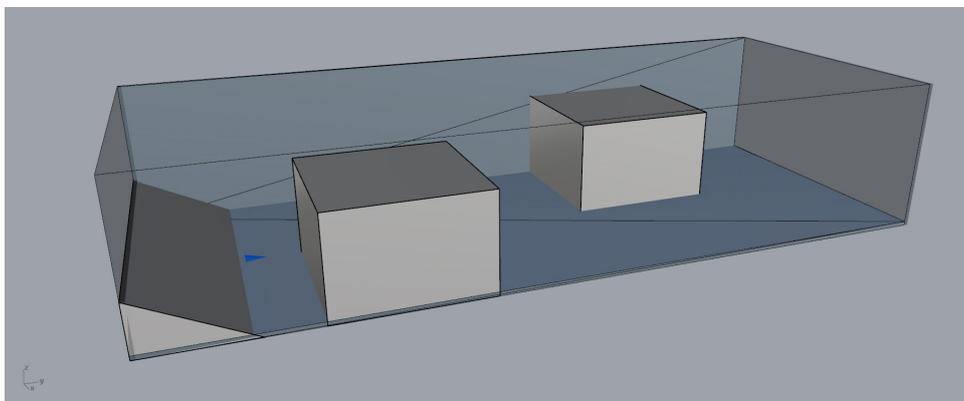


Figure 1: Simulation set up with two obstacles in the channel. The wedge is used only at the start of the simulation to initialise the inflow of water from left to right.

The resolution in this tutorial has been chosen to fit within the maximum cells per direction permitted in RhinoCFD Lite (i.e.  $40^3$ ). Other users may wish to increase the resolution to get more detailed results, in particular near the channel walls and obstacles, but should keep in mind that the simulations will take longer to run.

Free surface models are required when there is an interaction between two or more distinctly different fluids, separated by sharply defined interfaces. This tutorial focuses on applying VOF, one of the most modern free surface models available in CFD, to water flowing through air. It has a multitude of uses including flow past ships, slug flow (cavitation) in pipes, sloshing of liquids 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 also more computational expensive and transient simulations may take much longer to run.

## Geometry Set Up

### Box Obstacles

Create a box using the Box tool, dimensioning the box  $x = 150mm$ ,  $y = 150mm$  and  $z = 100mm$ . Place the box at  $x = 75mm$ ,  $y = -150mm$ ,  $z = 50mm$ . Make a copy of the box and place it at  $x = -75mm$ ,  $y = 150mm$ ,  $z = 50mm$ .

To resize and place the boxes the BoxEdit tool may be useful.

### Water Initialisation Wedge

Draw a right triangle with dimensions  $y = 150mm$ ,  $z = 50mm$ . Extrude the triangle along  $x = 300mm$ , to form a wedge. Place the wedge at  $x = 0mm$ ,  $y = -325mm$ ,  $z = 25mm$ .

The three objects should look like those shown in Figure 1. The domain, walls and in- and outlets also visible in Figure 1 will be created in the following section.

## CFD Analysis

### Domain and Edge Conditions

First, create a domain around the objects by clicking on the first button on the tool bar (Create Domain to fit objects). The domain defines the region in which fluid flow is possible. When prompted, select the desired working directory.



Figure 2: RhinoCFD tool bar

Resize the domain using the BoxEdit tool to  $x = 300mm$ ,  $y = 750mm$  and  $z = 150mm$ , then move the domain to the location  $x = 0mm$ ,  $y = 0mm$  and  $z = 75mm$ . This should align the wedge with the face of the domain that will be specified as an inlet and should align the outside of the boxes with the channel side walls.

To set up the boundary conditions of the simulation, right-click the second button on the tool bar (Edit Domain Edge Conditions). Specify the face next to the wedge (Ymin) to have a Flow condition and the opposite face (Ymax) to be Open. The side walls (Xmin & Xmax) as well as the floor (Zmin) should be set to Wall conditions and the top face (Zmax) should be set to Open or left unchanged. These menu settings are shown in Figure 3.

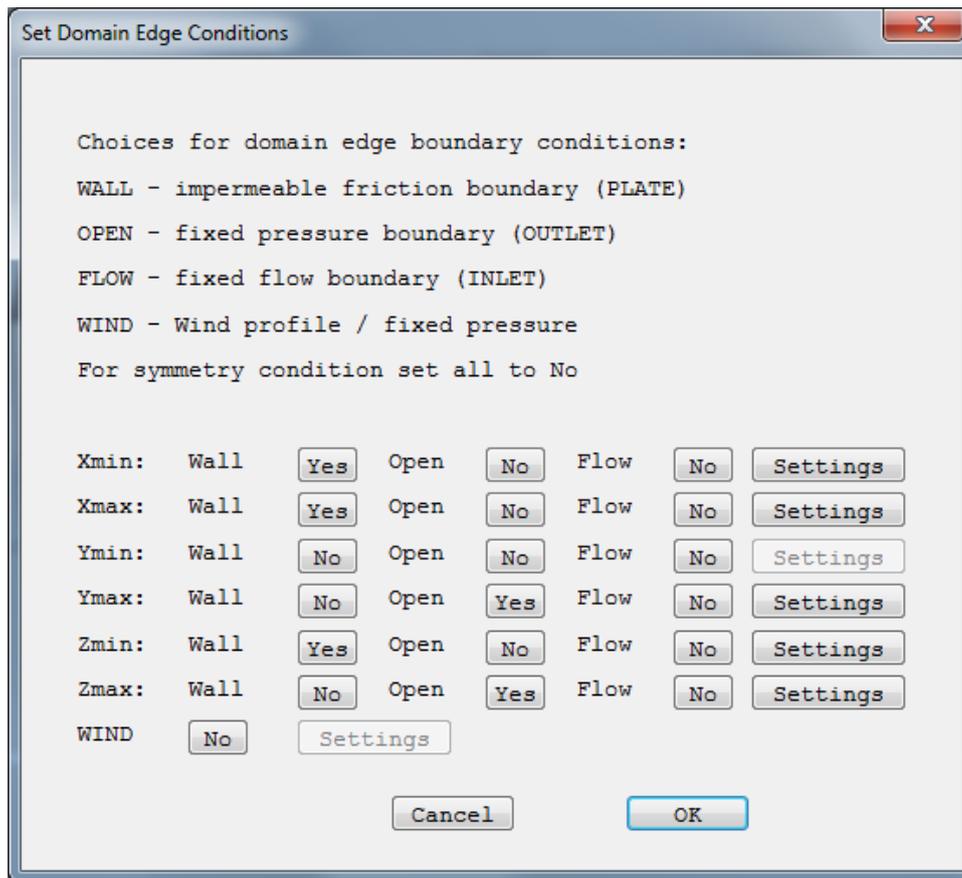


Figure 3: Domain Edge Conditions

In order to separately control the inflow of both water and the air above it at the inlet, re-dimension the new newly created geometry at the inlet and make a copy of it, to create one inlet for water and one inlet for air. Dimension the water inlet to  $z = 50mm$  and position it at  $z = 25mm$ , and dimension the copy for the air inlet to  $z = 100mm$  and position it at  $z = 100mm$ .

## Main Menu

### Geometry

In the main menu (third tool bar button, Edit Solution Parameters) under Geometry change the Time Dependence to Transient and set up the simulation to run for 3 seconds in 300 steps (Figure 4). A general guideline for VOF simulations is that information should only travel half a cell per time step (this is known as the Courant-Lawrence Friedrich (*CLF*) condition with  $CFL = 0.5$ ). A conservative estimate corresponds to dividing the minimum cell size by the maximum velocity and multiplying by the *CFL* number; in this tutorial we estimate the time step with  $0.5 \times 0.01m \div 0.5m/s = 0.01s$ .

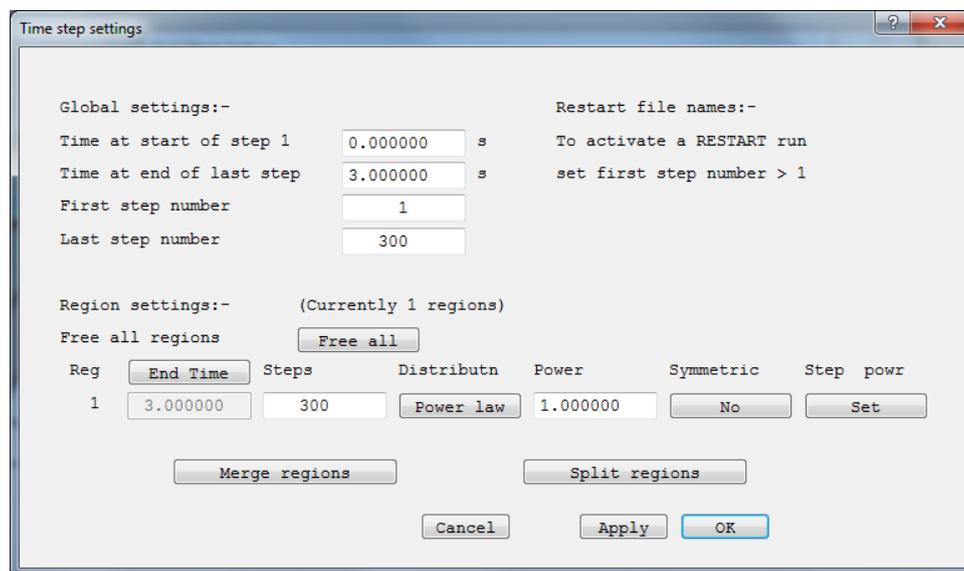


Figure 4: Time Step settings

Grid spacing will be set up in the following section after the main menu setting have been completed.

### Models

In the Models section enable the Free-surface model and select VOF - CICSAM as shown in Figure 5.

Of the available methods CICSAM is the best at preserving the interface between the fluids, when sufficiently small time steps are used. For large scale problems like ships, where the interface does not need to be as well preserved, or when large time steps need to be taken to reduce computational time, MHRIC or STACS should be considered respectively.

For more information about the available [VOF methods](#) please refer to this technical report.

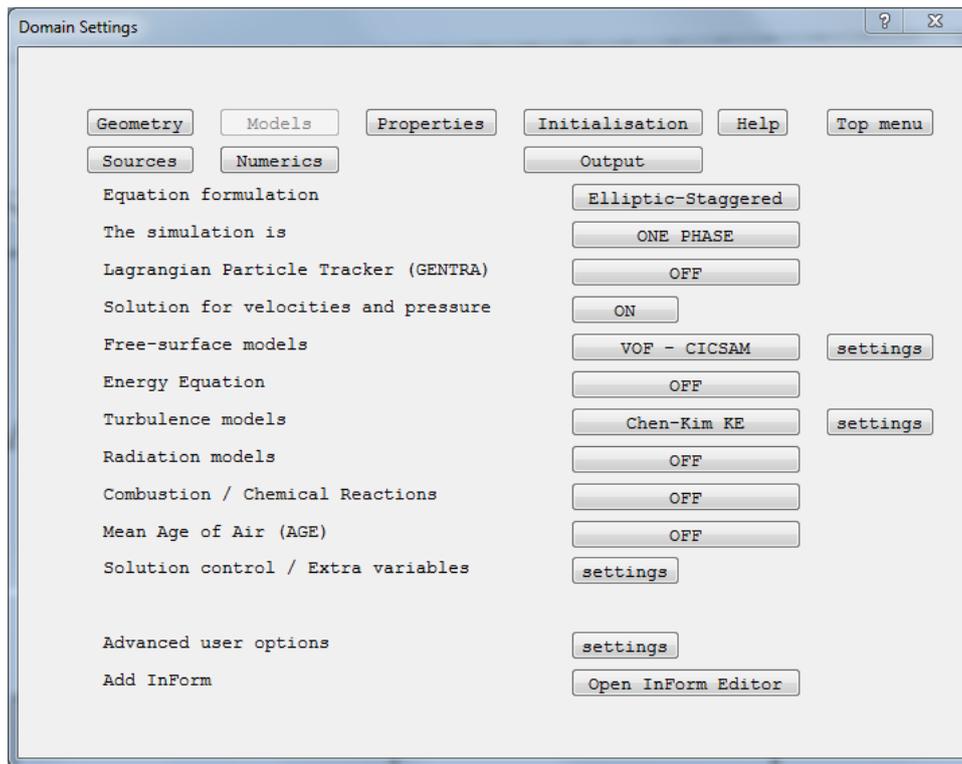


Figure 5: Model settings

## Properties

In the Properties section the settings have automatically been set to air for the light fluid, water for the heavy fluid and the domain to be initialised as the light fluid (Figure 6).

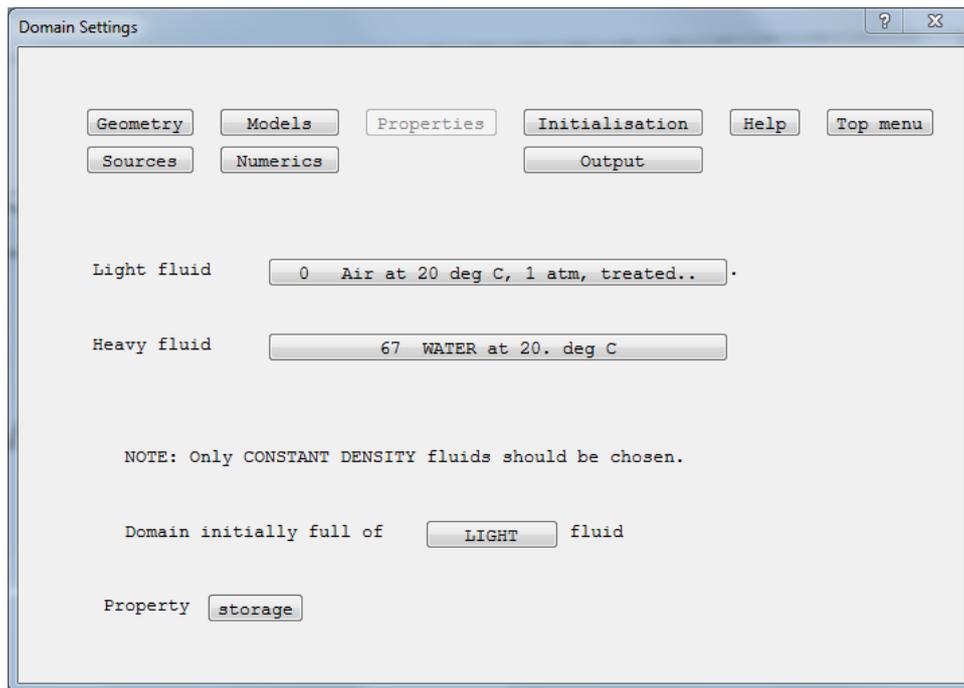


Figure 6: Properties settings

### Sources

In the Sources section enable gravity with a constant buoyancy model and gravity acting in  $gz = -9.81m/s^2$  (Figure 7).

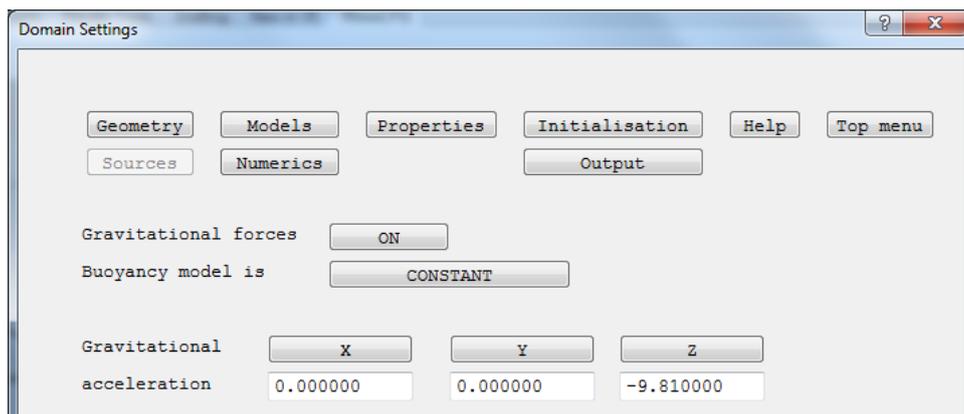


Figure 7: Sources settings (Additional options below Gravitational Acceleration will be left unchanged for this tutorial.)

## Numerics

In the Numerics section reduce the number of iteration for each step to 40 (If a simulation requires many more iterations per time step, it is recommended to reduce the time step size). Select the Relaxation control button and edit the values for reference velocity and reference length (Figure 8). These should be set to characteristic values of the flow, the velocity could be set to the inlet velocity and the reference length to e.g. the water depth (water inlet height). In some places in the domain the water will accelerate to velocities faster than the inlet, and near walls the characteristic length of the flow may be smaller than the water depth, therefore in this simulation set the reference velocity to  $v_{ref} = 1m/s$  and reference lengthscale to  $l_{ref} = 0.01m$ .

See the RhinoCFD basics [Convergence](#) video for further information.

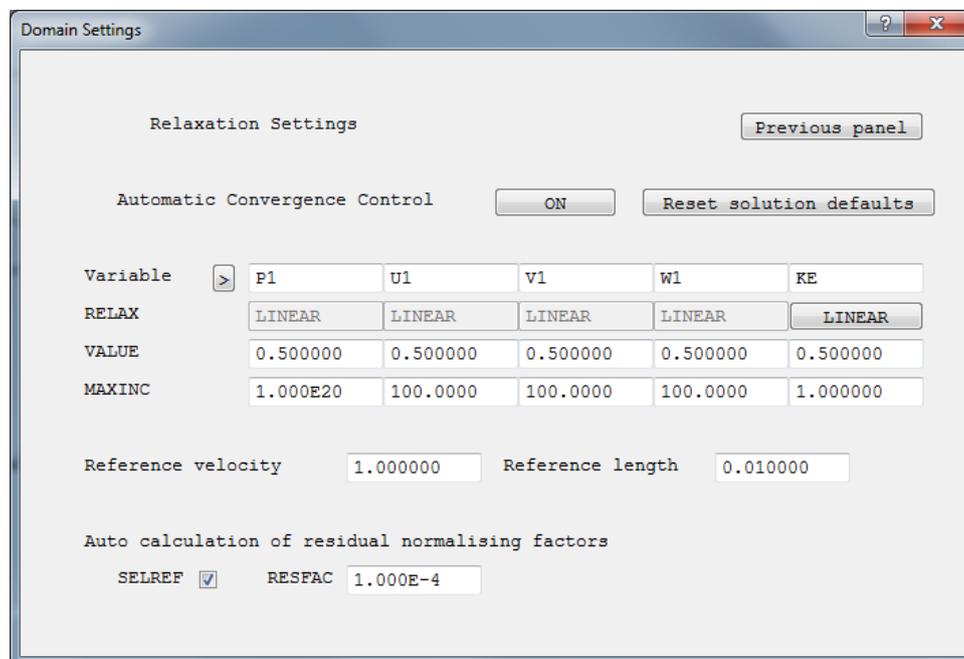


Figure 8: Relaxation setting using Automatic Convergence Control, given a Reference Velocity and Reference Length

## Output

In the Output section under Field dumping set results to be output every 5 steps. This should allow for the creation of a relatively smooth animation of the water flow in post-processing. To use less storage increase the number of steps between outputs, for an even smoother animation decrease the number of steps between outputs.

## Output of Extra Variables

Optionally, RhinoCFD can also calculate the forces applied to any object in the domain. To activate this option go to Output and select ‘output of forces and moments on blockage objects’ and set it to ON. Clicking on settings opens a panel where options such as the calculation of Friction forces (part of the drag component) and drag coefficients can be enabled.

Optionally, user may also wish to store additional useful quantities e.g. the height above the floor, which can be used to colour the water height. This can be done by selecting the seventh icon on the tool bar, which allows for the input of formulas via RhinoCFD’s In-Form feature. This is a very powerful feature, but for the purpose of this tutorial we will only use it to store an additional variable for post-processing.

Open saveblock19 and enter ‘(Stored of HGHT is ZG)’ to store the height.

## Object Properties

The next step is to adjust the inflow boundary conditions and initialise the water inflow. To edit the properties of the objects in the domain there are two options: 1) select the object in the Rhino screen so that it is highlighted and then click on the third tool bar button (Edit CFD Properties), or 2) open the object list by right-clicking the third button on the tool bar (Show table of Objects) and selecting the object from the list.

## Setting Boundary and Initial Conditions

Edit the Water Inlet properties to have the inlet density of the Heavy liquid (Water) and a velocity of  $v = 0.3m/s$ , as shown in Figure 9. For the air inlet, set the density to Light and the velocity to  $v = 0.03m/s$ .

Next, edit the properties of the wedge by opening the CFD Properties window and clicking on the Attributes button. As shown in Figure 10, change the Type to Heavy fluid and then select the Initial Values button to specify an initial velocity of  $0.3m/s$  in the Y-direction.

Finally, locate the probe using the RhinoCFD tool bar by left-clicking the sixth tool bar button Show Probe, then positioning it in front of the wedge close to the floor. This way the simulations monitor plots will show changes in quantities allowing to assess if water is inflowing correctly.

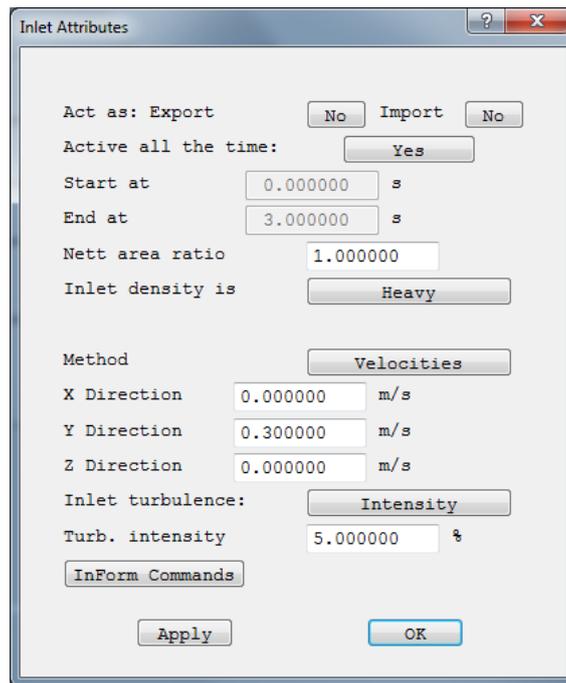


Figure 9: Water Inlet Properties

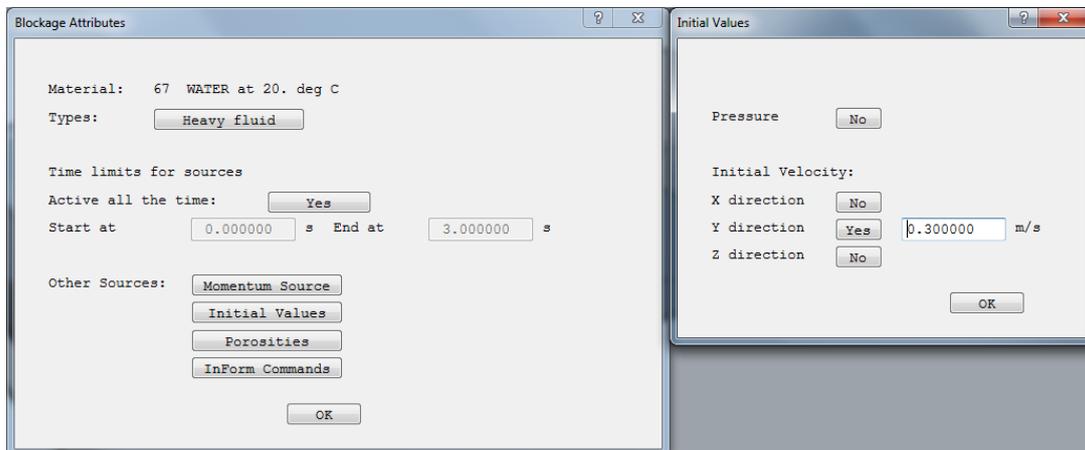


Figure 10: User defined object settings

## Meshing

It is important to refine the mesh near geometry so that their effect on the flow are correctly simulated. In this tutorial the geometry is rather simple and our goal is to stay with in the 40 cells per direction limit of RhinoCFD Lite. For this purpose we will manually edit the grid spacing in each of the three directions. To do this enter the geometry menu by left clicking on the 4th toolbar icon. Ensure that each direction is set to manual.

Enter the x-direction menu by clicking on 'X-direction' and edit regions 1 and 2 to each

have 10 cells with symmetrical Power Law distributions with a power of 1.3. This clusters the grid near the inside faces of the boxes and the walls.

In the y-direction set the first 4 regions to have 9 cells and the last region to have 4. For the first 4 set the distributions to be symmetrical Power Laws with powers of 1.2 and for the last region non-symmetrical with power 1.3, as shown in Figure 11. This will cluster the grid near the front and back faces of the boxes.

In the z-direction set all regions to non-symmetric. Set the first to Power Law with power 1.3 and the second and third to a Geometric Law with powers 1.05 and 1.10 respectively. This clusters the grid near the floor with increasing cell spacing away from the floor.

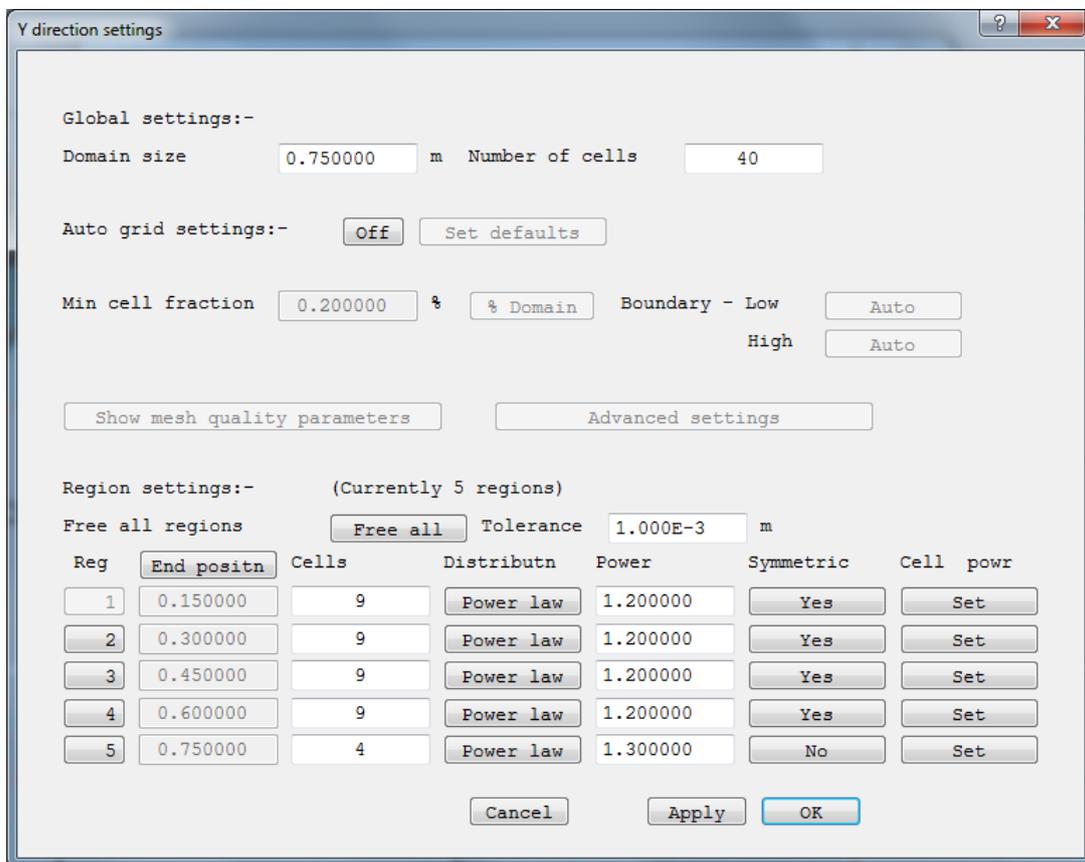


Figure 11: Grid Mesh Settings in the Y-Direction

Depending on the size of the domain or the geometry used, these afore mentioned values may need to be iterated until a reasonable mesh is found. Table 1 summarises the values used for the grid. Figures 12 and 13 show the resulting grid from two different directions.

Axis	Region				
	1	2	3	4	5
X	10	10	-	-	-
Y	9	9	9	9	4
Z	8	6	4	-	-

Table 1: Number of mesh cells in each region of each direction

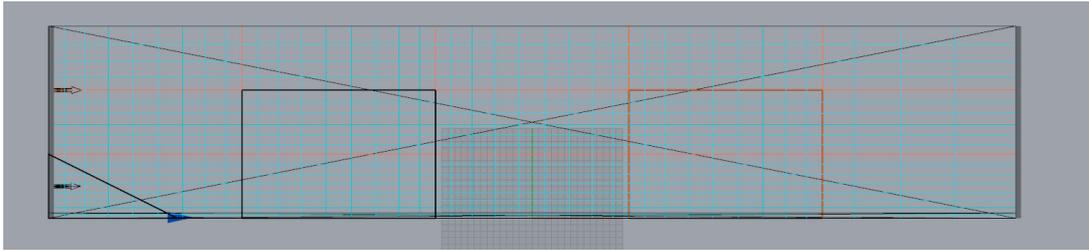


Figure 12: Grid in Y-Z-plane

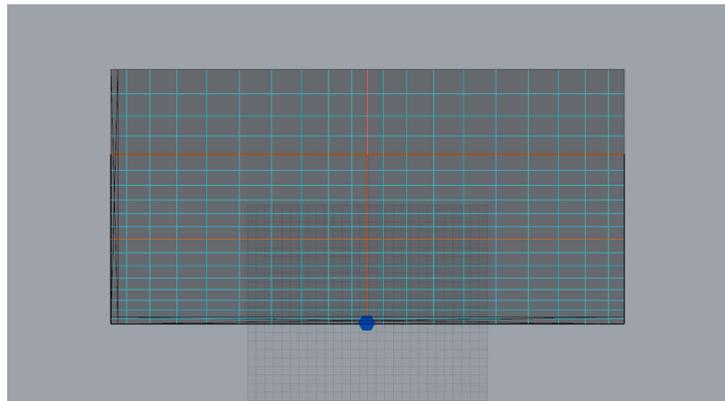


Figure 13: Grid in X-Z-plane

## Results

On the tool bar click 'Load Results' and click OK. Click on 'play time series' drop down under the time step button and the results for all time steps will be played. Examples of iso-surfaces of  $SURN = 0.5$ , which denotes the interface between water and air, coloured by flow variables are shown in Figures 14 and 15. Figure 16 shows example output to the Results File listing the forces acting on the blocks at the final time step.

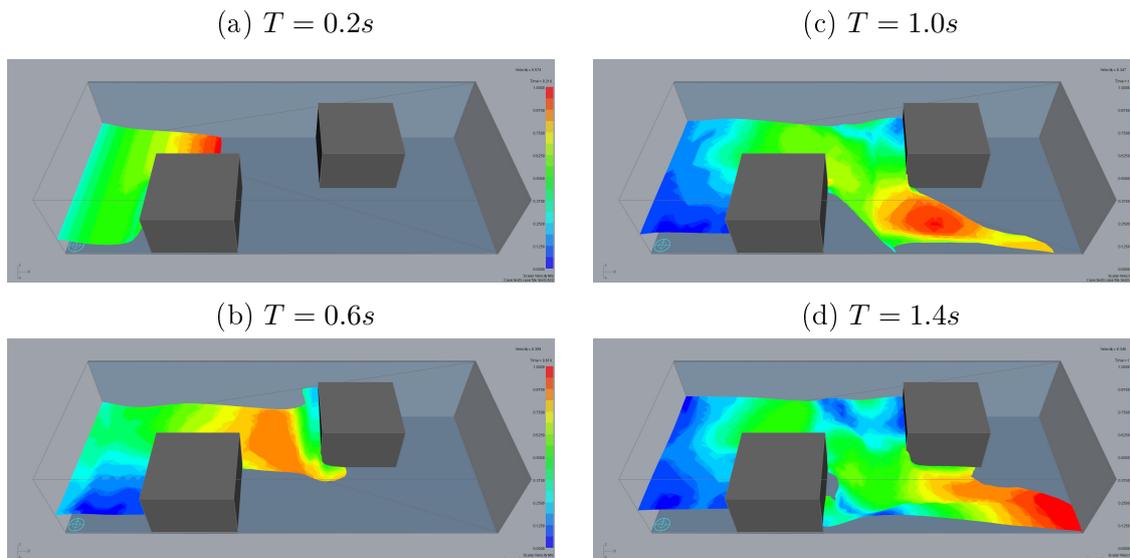


Figure 14: Iso-surface of  $SURN = 0.5$  coloured by velocity magnitude

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.

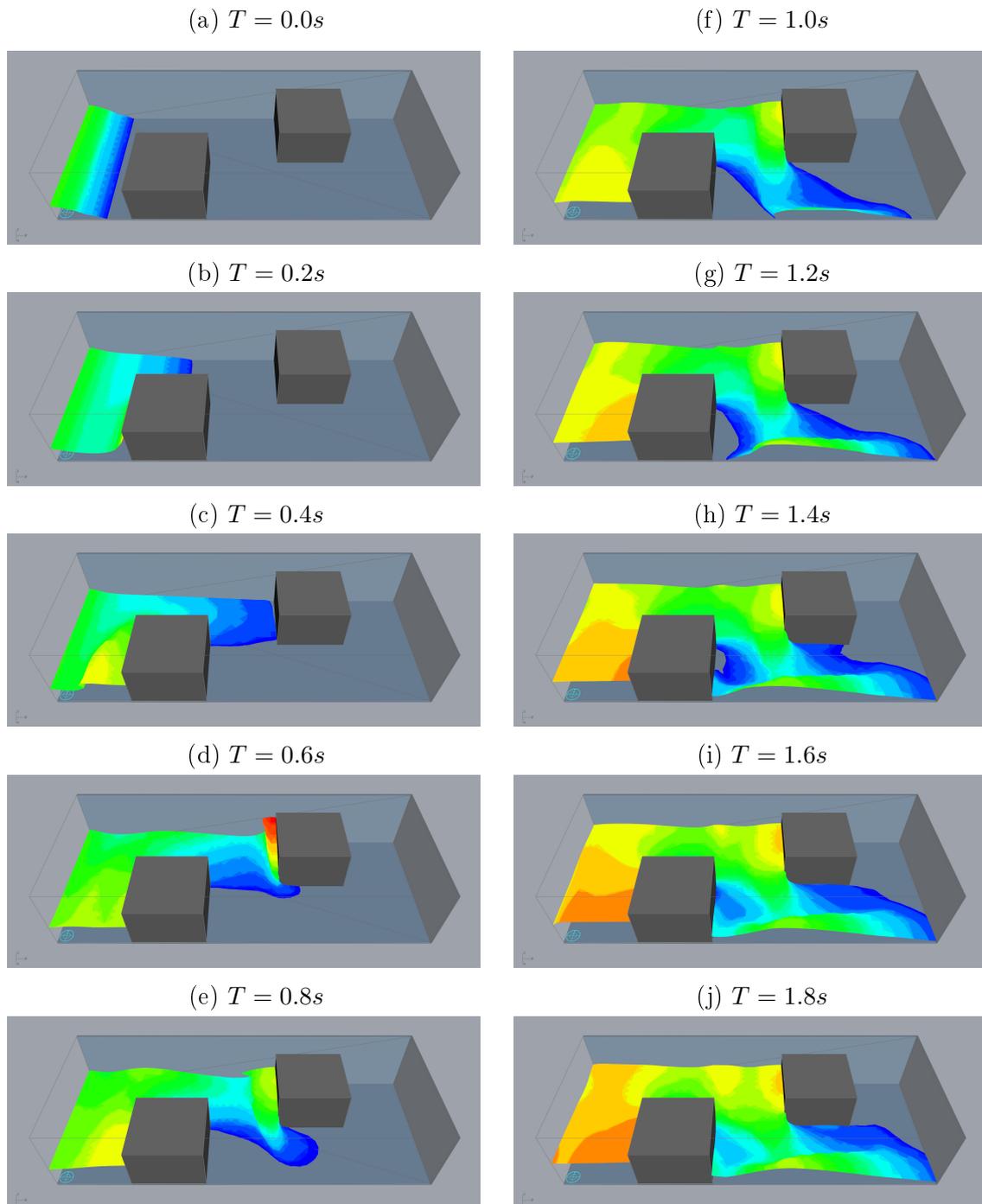


Figure 15: Iso-surfaces of  $SURN = 0.5$  coloured by the water height above the floor

```

result - Notepad
File Edit Format View Help

-----
Integrated force on object: BLOCK1
-----
Fx = 3.067605E+00 (Pressure= 3.070882E+00, Friction= -3.277220E-03)
Fy = 2.422701E+00 (Pressure= 2.419926E+00, Friction= 2.775660E-03)
Fz = -9.810328E-03 (Pressure= -1.004495E-02, Friction= 2.346207E-04)
Pressure force on West side: 3.070882E+00
Pressure force on East side: 0.000000E+00
Pressure force on South side: 4.973505E+00
Pressure force on North side:-2.553580E+00
Pressure force on Low side: 0.000000E+00
Pressure force on High side: -1.004495E-02
Force unit vector: 7.847673E-01 6.197855E-01 -2.509719E-03

Total moment about X axis = 8.554459E-02
Total moment about Y axis = -7.036455E-02
Total moment about Z axis = 1.494973E-01

Moment of Fx about Y axis = -6.810284E-02 at distance Z = 2.220066E-02
Moment of Fy about X axis = 8.329054E-02 at distance Z = 3.437920E-02
Moment of Fz about X axis = 2.254020E-03 at distance Y = 2.297599E-01
Moment of Fx about Z axis = 7.008546E-01 at distance Y = 2.284697E-01
Moment of Fy about Z axis = -5.513575E-01 at distance X = 2.275796E-01
Moment of Fz about Y axis = -2.261731E-03 at distance X = 2.305459E-01
Total exposed vertical area = 4.500008E-02
Sum vert. area with P1 > 10.00 = 3.408321E-02
Ratio area P1 > 10.00/Totalarea = 75.74 %

-----
Integrated force on object: BLOCK2
-----
Fx = -2.823836E-01 (Pressure= -2.854745E-01, Friction= 3.090895E-03)
Fy = 4.094348E+00 (Pressure= 4.091418E+00, Friction= 2.930114E-03)
Fz = -7.533860E-03 (Pressure= -7.743984E-03, Friction= 2.101235E-04)
Pressure force on West side: 0.000000E+00
Pressure force on East side: -2.854745E-01
Pressure force on South side: 4.106660E+00
Pressure force on North side:-1.524201E-02
Pressure force on Low side: 0.000000E+00
Pressure force on High side: -7.743984E-03
Force unit vector: -6.880555E-02 9.976284E-01 -1.835699E-03

Total moment about X axis = 1.081146E-01
Total moment about Y axis = 2.525151E-03
Total moment about Z axis = -4.434057E-01

Moment of Fx about Y axis = 3.095910E-03 at distance Z = 1.096349E-02
Moment of Fy about X axis = 1.040945E-01 at distance Z = 2.542395E-02
Moment of Fz about X axis = 4.019993E-03 at distance Y = 5.335901E-01
Moment of Fx about Z axis = -1.556028E-01 at distance Y = 5.510334E-01
Moment of Fy about Z axis = -2.878028E-01 at distance X = 7.029269E-02
Moment of Fz about Y axis = -5.707578E-04 at distance X = 7.575901E-02
Total exposed vertical area = 4.500011E-02
Sum vert. area with P1 > 10.00 = 1.699777E-02
Ratio area P1 > 10.00/Totalarea = 37.77 %

Ln 1255, Col 2

```

Figure 16: Snippet from Results file showing forces on Blocks 1 &amp; 2



---

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