RhinoCFD Tutorial

Free Surface - Molten Metal Filling a Mould

RhinoCFD Official document produced by CHAM
October 13, 2021

Introduction

This tutorial demonstrates how to set up a free surface model of isothermal flow in a mould. It is also suitable for simulations of free surface flow inside a confined space. It is designed to get you up to speed on how to apply Volume Of Fluid (VOF) models 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 individually in Rhino or a geometry file can be downloaded from RhinoCFD Tutorials.

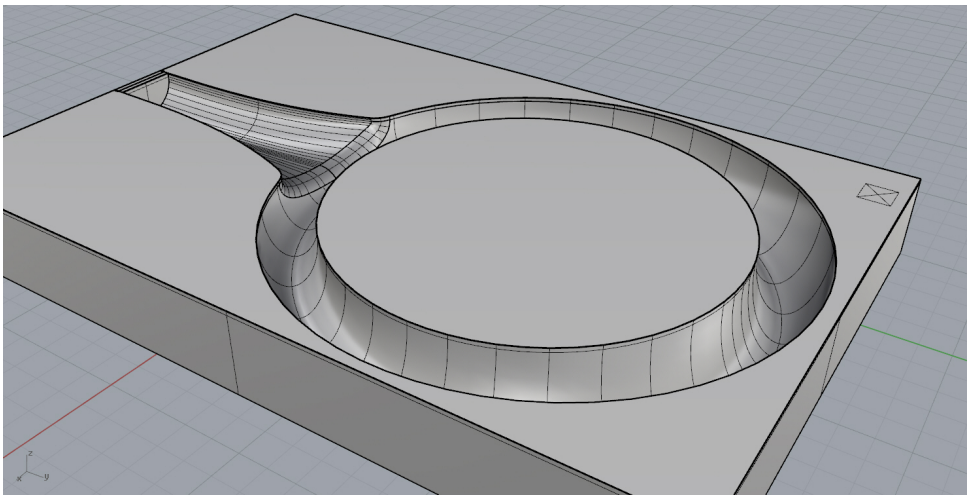


Figure 1: View into the interior of a mould showing the bottom half using a clipping plane. The simulation set up has one inlet on the left of the image and an outlet at the far end of the flow path, furthest from the inlet (not visible here).

The resolution required for this tutorial is relatively high and does NOT fit within the maximum cells per direction permitted in RhinoCFD Lite (i.e. 40^3). Users may wish to manually adjust the resolution to fit within the restrictions, others may increase it further to obtain more detailed results, 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 liquid metal flowing through air. VOF 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 and transient method and also more computationally expensive.

Geometry Set Up

Inlet

The flow into the domain is modelled using an 'Angled-In'. Angled-Ins are 3d objects that intersect other solid objects and specify inlet conditions on the surface of the intersection. In this tutorial the mould form is open at the location at which fluid would flow in. We create two 3d objects, one a solid used to block flow out of the mould, the other a Angled-In which overlaps the block, to specify inlet conditions on its face. Both objects are cylindrically shaped, the blockage has a radius larger than the channel radius and the Angled-In block has the same radius as the channel. A top-down or side-on, wire-frame view of the objects is shown in Figure 2.

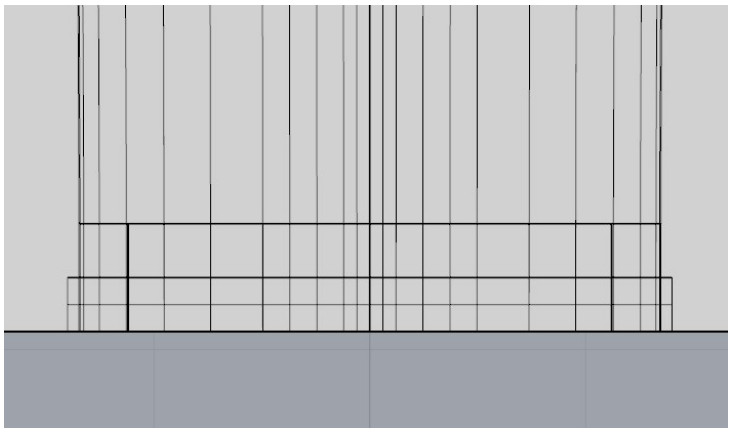


Figure 2: Close up of the objects used to block off the inlet area and specify inlet conditions. (The inflow direction will be upwards on the page.)

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

Outlet

An Angled-Out is used on the far end of the ring to allow air to escape the mould as it fills. Angled-Outs create outlet surfaces where they intersect with solid objects. A small cylinder can be used as the angled-out, intersecting the top surface of the inside of the ring furthest from the inlet (see Figure 3).

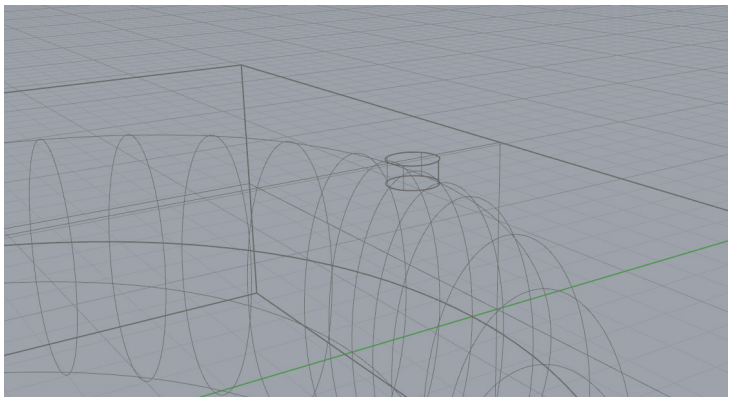


Figure 3: Location of the object used to specify an outlet area.

CFD Analysis

Domain and Boundary Conditions

To start the set up, 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 4: RhinoCFD tool bar

Resize the domain, using the BoxEdit tool, to be slightly smaller than the mould in all three directions. RhinoCFD defaults to making the domain twice the size (in each direction) of the box bounding all geometry objects and placing the centre of the domain at the centre of the bounding box.

Domain Edge Conditions will not be used in the Tutorial and all boundary conditions will be applied through the Object Properties menu.

Using the Properties File

Liquid metals are not part of the standard fluids included in the RhinoCFD/PHOENICS materials library. It is therefore necessary to add a new material and its properties to the Properties File. For this tutorial we assume the liquid to be molten gold at $T = 1100^oC$ with the required properties listed in Table 1. For the simulation run in this tutorial only the first two properties are actually required, but for completeness all properties are included.

Property	Density	Viscosity	Specific Heat	Thermal Conductivity	Thermal Expansion	Compressibility
Units	km/m^3	$Pa.s$	J/kg^oC	$W/m.K$	$^oC^{-1}$	–
	15900	$5.13e-3$	150	260	$1.45e-5$	$1.43e-11$

Table 1: Properties of liquid gold at $T = 1100^oC$ required for full specification in the Properties File

The Properties file is located in the folder where Rhino stores Plug-in data:
%APPDATA%\McNeel\Rhinoceros\5.0\Plug-ins\RhinoCFD (...\2018...\d_earth
The AppData folder is generally hidden and can be accessed by typing `%appdata%` into the address bar of the Windows Explorer and then navigating to the above location from there.

Before editing the Properties file, make a backup of it, just in case.

To edit the Properties file, open it with a text editor such as Notepad. Find entry number 69, and in the same format as the other entries, enter a new entry below it: number 70. After adding the new material using the properties listed in Table 1 it should look like the following Listing 1:

```
   <69  >   Air at 20 deg C, 1 atm, treated as incompressible
69       1.225       1.45E-05    1005.0    0.0258   3.41E-3    0.0

   <70 >  Gold Liq at 1100 deg C
70     1.59E4     5.13E-3     150.0     260.0     1.45E-05    0.0
```

Listing 1: New Properties File entry for liquid gold at 1100^oC

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 1 second in 1000 steps (Figure 5). A general guideline for VOF simulations is that information should only travel about half a cell or less 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 desired CFL number; in this tutorial we estimate the time step with an approximate length scale of $L_{ref} = 1mm$ and a estimated peak velocity of $V_{ref} = 0.5m/s$, as $dt = 0.5 \times 0.001m \div 0.5m/s = 0.001s$.

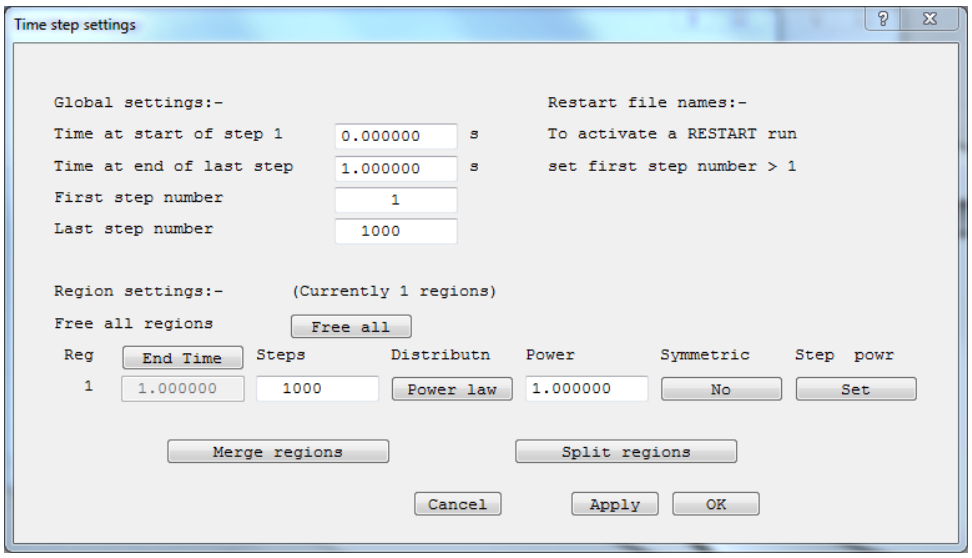


Figure 5: Time Step settings

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 6.
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.

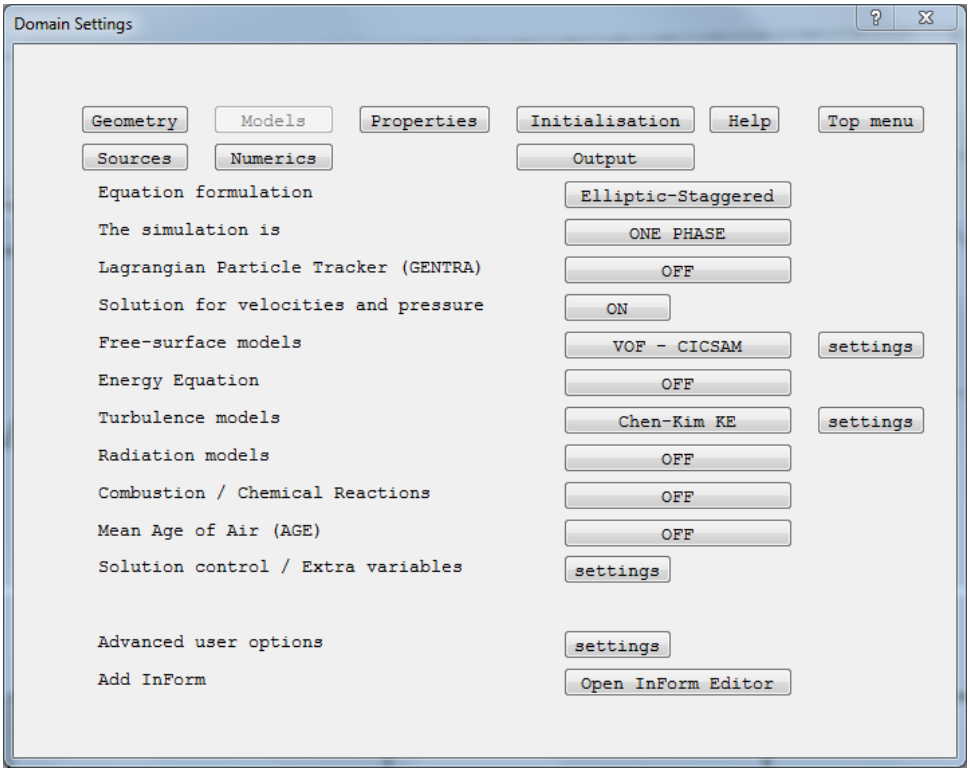


Figure 6: Model settings

Properties

In the Properties section the settings default to air for the light fluid, water for the heavy fluid and initialisation of the domain as the light fluid (Figure 7). To change the Heavy fluid click on the Heavy fluid button to bring up the selection menu, then select Liquids and the newly created "Gold Liq at 1100 deg C" material as shown in Figure 8.

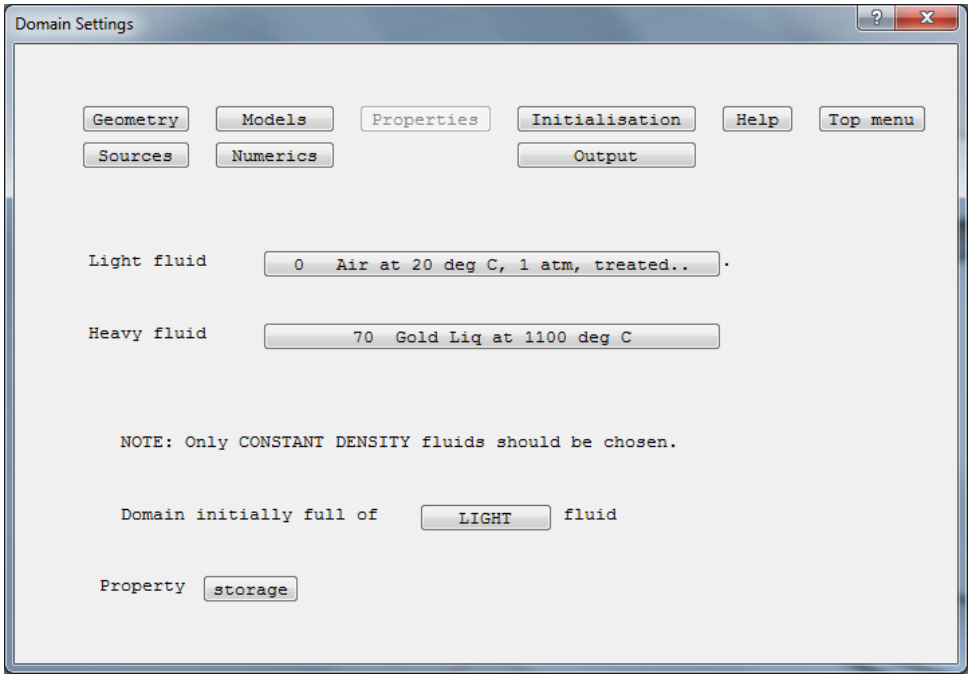
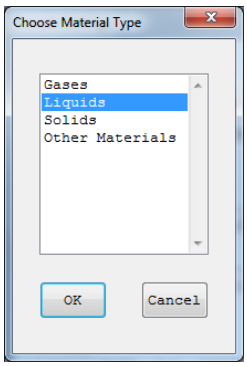


Figure 7: Properties settings

(a) Choose Material Type



(b) Select Material

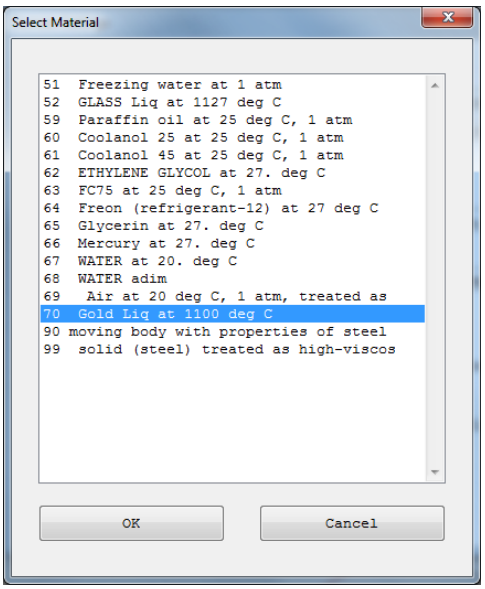


Figure 8: Menu windows to select a different Heavy fluid from the Liquids category.

Sources

In the Sources section enable gravity with the constant buoyancy model. By default gravity acts downwards in the z-direction, $gz = -9.81m/s^2$. For this Tutorial we are going to assume that the ring mould is at an angle, as it would be on a tree of many moulds. Assuming an angle of 30^o to the horizontal plane, set the gravitational acceleration in y to $gy = 4.905m/s^2$ and in z to $gz = -8.496m/s^2$ (Figure 9).

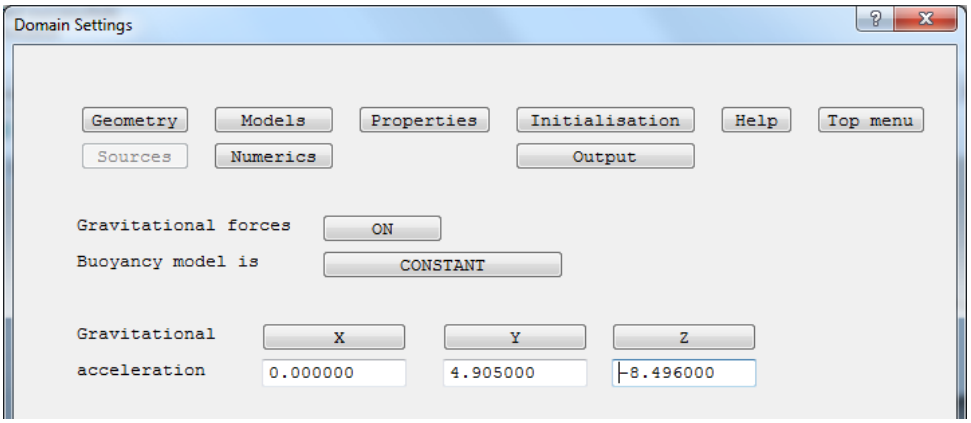


Figure 9: Sources settings (Additional options below Gravitational Acceleration will be left unchanged for this tutorial.)

Numerics

In the Numerics section reduce the number of iteration for each step to 50 (If a transient simulation requires many more iterations per time step, it is recommended to reduce the time step size). We will also increase the tolerance to which each time step is solved from 0.01% to 0.1%, as it is not necessary to solve every time step to the same high accuracy as a steady run.

For this tutorial we are going to use the Convergence Wizard (CONWIZ) to deal with relaxation of the computations. CONWIZ is enabled by default, but must be supplied with two pieces of additional information to work effectively: Select the Relaxation control button and edit the values for reference velocity and reference length (Figure 10).

These values should be set to characteristic values of the flow, the velocity could be set to e.g. the inlet velocity and the reference length to e.g. the radius of the inlet or the width of a section of the ring mould. In some places in the domain the gold will accelerate to velocities faster than the inlet, and near walls the characteristic length of the flow may be smaller than the inlet radius, therefore in this simulation set the reference velocity to $v_{ref} = 0.2m/s$ and reference length scale to $l_{ref} = 0.001m$.

See the RhinoCFD basics Convergence video for further information.

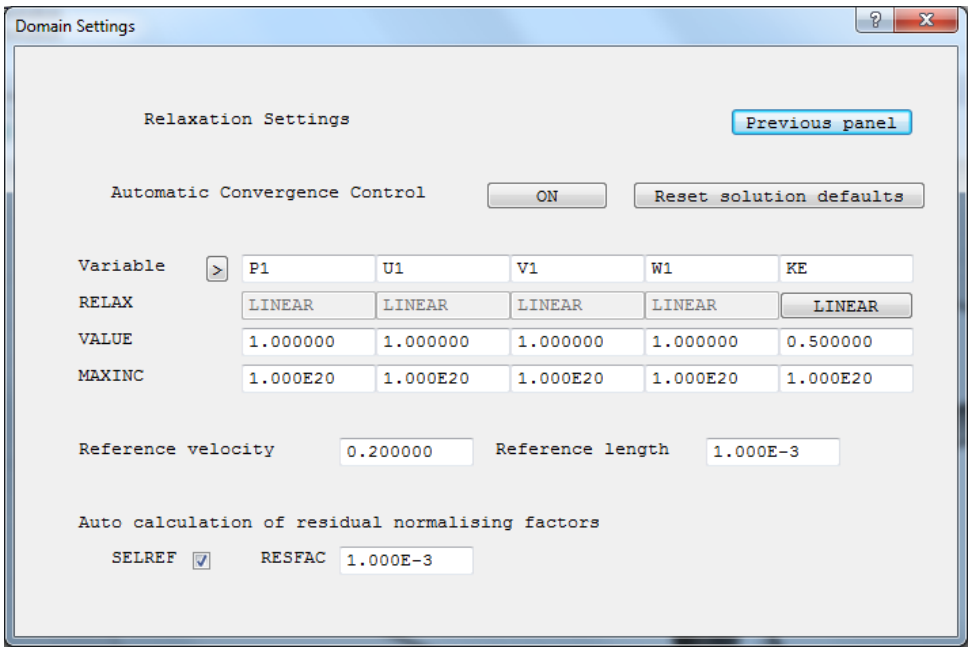


Figure 10: Relaxation setting using Automatic Convergence Control, given a Reference Velocity and Reference Length

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 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.

Object Properties

The next step is to set up the inflow boundary conditions. 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 Inlet properties to have the inlet density of the Heavy liquid (Gold) and a velocity of $v = 0.1m/s$, as shown in Figure 11.

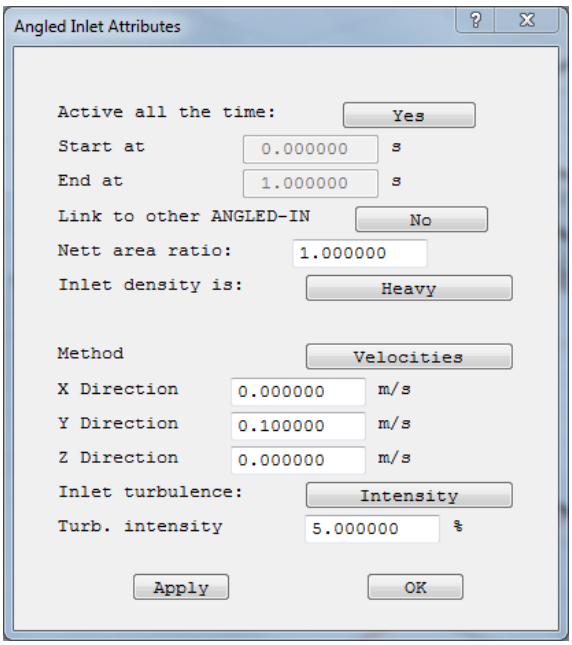


Figure 11: Water Inlet Properties

Ensure that the Mould and Inlet Block are set to "Blockages", the Inlet is set to "Angled-In" and the Outlet is set to "Angled-Out". Also, set the Inlet Block and the Outlet to not affect the grid in any direction.

Finally, locate the probe using the RhinoCFD tool bar by left-clicking the sixth tool bar button (Show Probe), then positioning it at the end of the inlet channel before the ring and close to the floor. This way the simulations monitor plots will show changes in quantities allowing to assess if gold is inflowing correctly.

Meshing

The features of the mould geometry are very small and in order for them to be captured properly by the grid we need to tighten the tolerances in the Grid Mesh Settings menu. By default the tolerance is set to 0.001m, but the features of the mould geometry are only a couple of millimetres in total, therefore we set the tolerances in each direction to $1e{-}6$m.

We will try to refine the mesh near geometry so that their effect on the flow are accurately simulated. 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 tool bar icon. Ensure that each direction is set to manual.

Enter the x-direction menu by clicking on 'X-direction' and edit regions 1 and 3 to each have 28 cells with non-symmetrical Geometric Progression distributions with a power of -1.02 and 1.02 respectively. Set region 2 to have 11 evenly spaced cells.

In the y-direction set the first (tiny) region to have 1 cell and the second region, spanning the whole length, to have 80 evenly spaced cells.

In the z-direction set the first regions to have 10 cells, the second 11 and the last one 12. Set the first and last regions to have non-symmetrical Geometric Progression distributions with of -1.02 and 1.02 respectively.

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

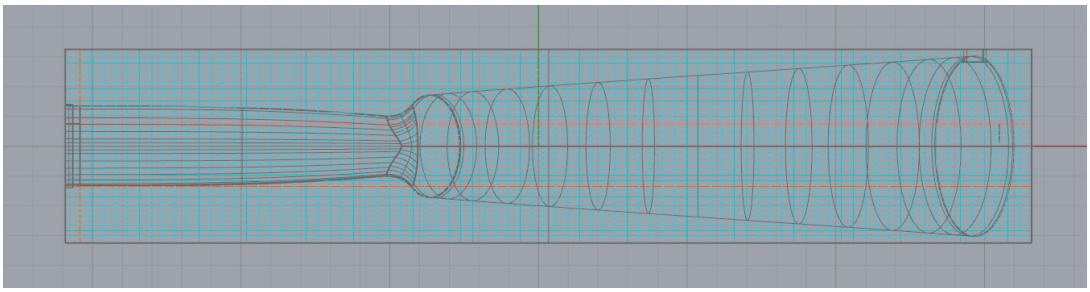


Figure 12: Grid in Y-Z-plane

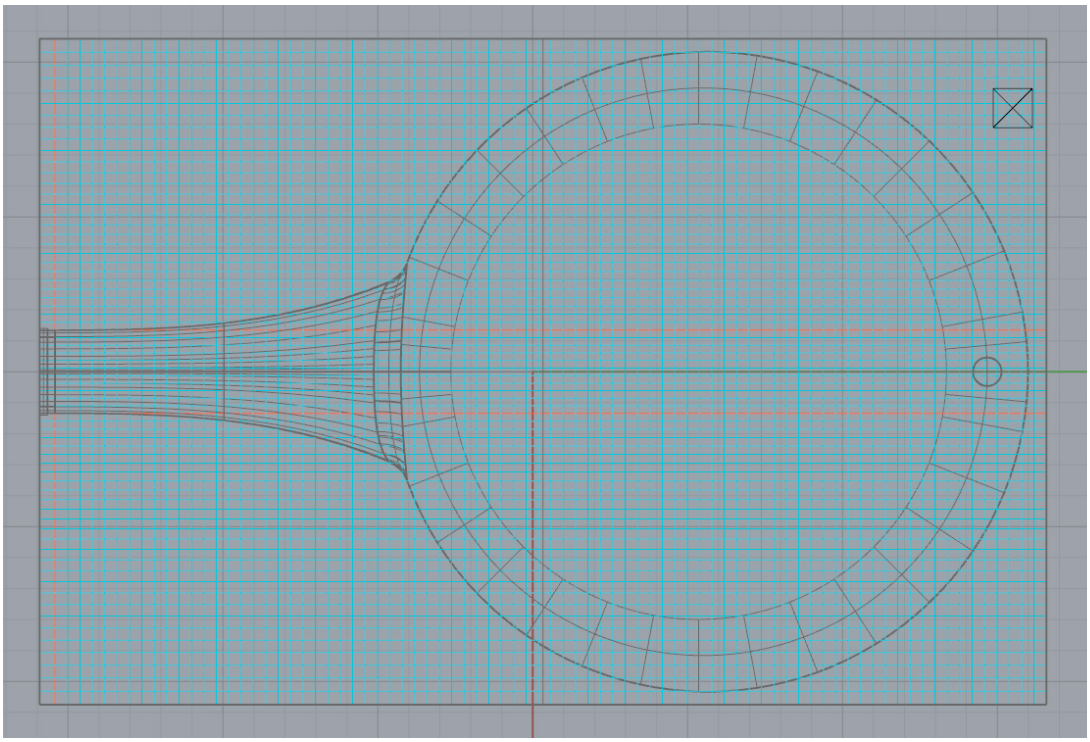


Figure 13: Grid in X-Y-plane

Running Simulations

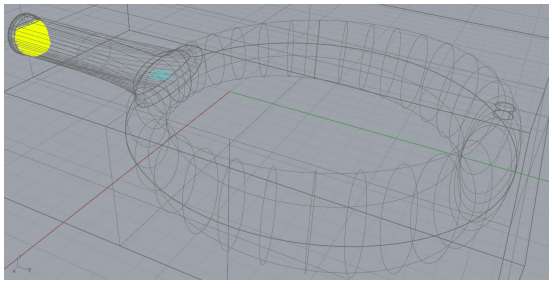
Once you are happy with the set up of all of the above, to run the simulation click on the 8th icon of the tool bar (Run Solver). Depending on the grid settings/number of cells and models used, simulations can take quite long to run. It is recommended to start with a rather coarse representation of the problem and gradually refine the grid as required and if computationally affordable.

Results

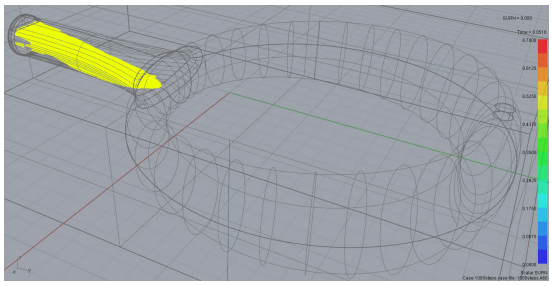
On the tool bar click 'Load Results' (9th icon) and click OK. Set up a cut plane or iso-surface of a variable of interest and click on 'Play time series' (3rd from last icon) to play the results from all time steps. Examples of iso-surfaces of $SURN = 0.5$, which denotes the interface between gold and air are shown in Figure 14

To bring up the 'Results' file, right click on the third to last tool bar icon  . As the name suggests, this file contains all the information resulting from the simulation including convergence, object properties, settings and derived variables and forces.

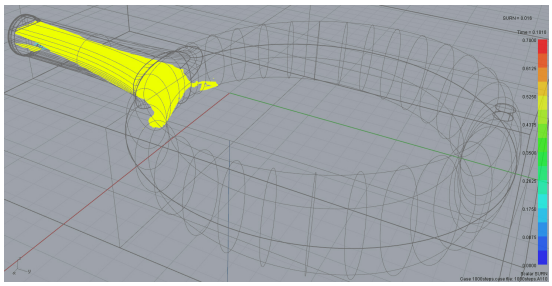
(a) $T \approx 0.00s$



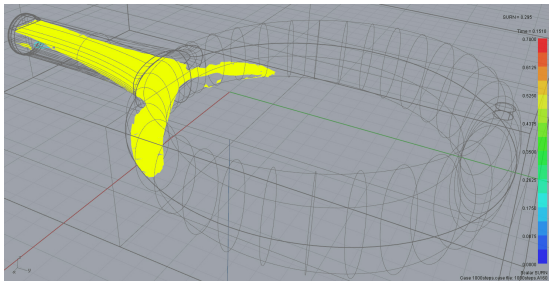
(b) $T \approx 0.05s$



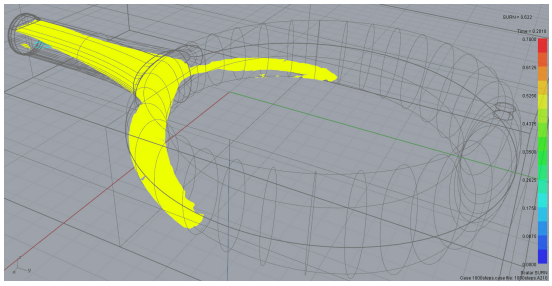
(c) $T \approx 0.10s$



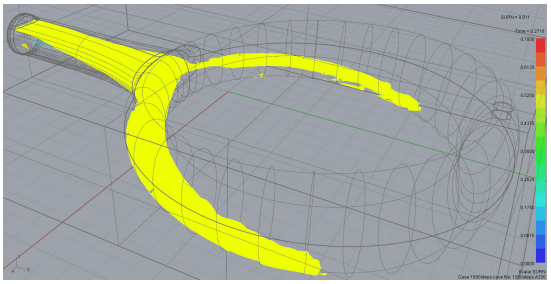
(d) $T \approx 0.15s$



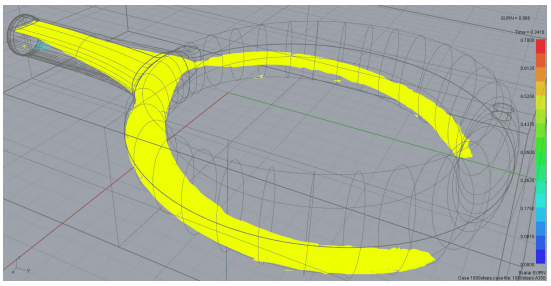
(e) $T \approx 0.20s$



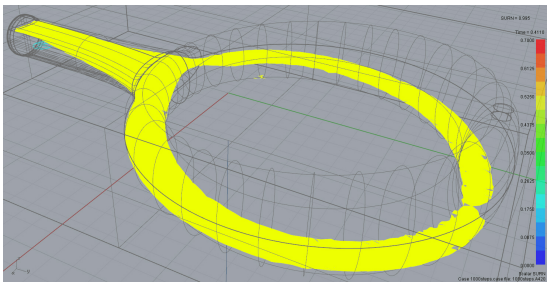
(f) $T \approx 0.25s$



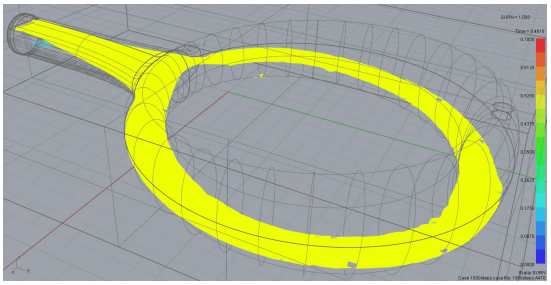
(g) $T \approx 0.32s$



(h) $T \approx 0.40s$



(i) $T \approx 0.50s$



(j) $T \approx 0.60s$

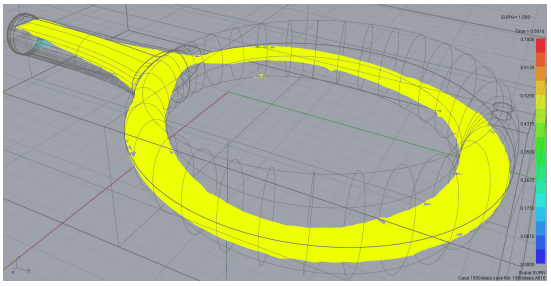


Figure 14: Iso-surfaces of $SURN = 0.5$ coloured by the water height above the floor

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