RhinoCFD Tutorial

FLAIR - Modelling a Trombe Wall

Introduction

A Trombe wall is a construction method that can be used to passively heat (or cool) a house using only direct sunlight. It generally consists of a sun-facing masonry wall covered with ventilation gaps at the top and bottom and a glass panel spaced a few centimetres away. Sunlight passes through the glass and heats up the wall, using it's thermal mass to store the energy which it will then radiate into the room over time. It also drives a convective current of air between the glass and the wall: the sunlight heats the air which rises and draws cooler air in through the bottom vent to be heated and recirculated back into the room.
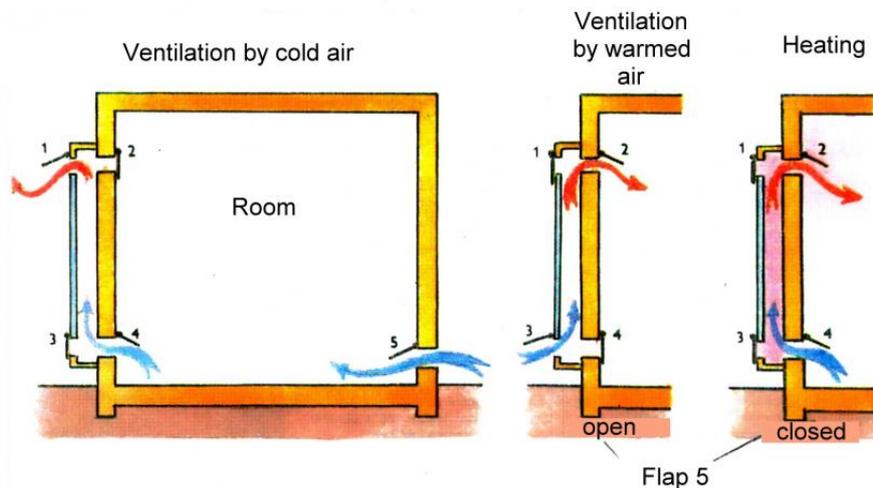
Figure 1: Description of a Trombe wall used to heat a room.

This tutorial demonstrates how to set up the models necessary to run this sort of simulation, including using the Sun and radiation modelling. 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. This tutorial assumes that you already understand how to set up a transient simulation. See the RhinoCFD basics Transient videos for more information.

CFD Analysis

Domain creation

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. Because the case we're interested in is related to the built environment, select FLAIR as the Menu Option. When prompted, select the desired working directory.

Figure 2: RhinoCFD tool bar

Resize the domain, using the BoxEdit or the Gumball tool, to be slightly within all of the geometry. Ensure the x axis of the domain is within both sides of the roof, rather than fitted to the window, as this corresponds to the outside of the house. Note in Figure 3 that the CFD domain is highlighted in yellow.
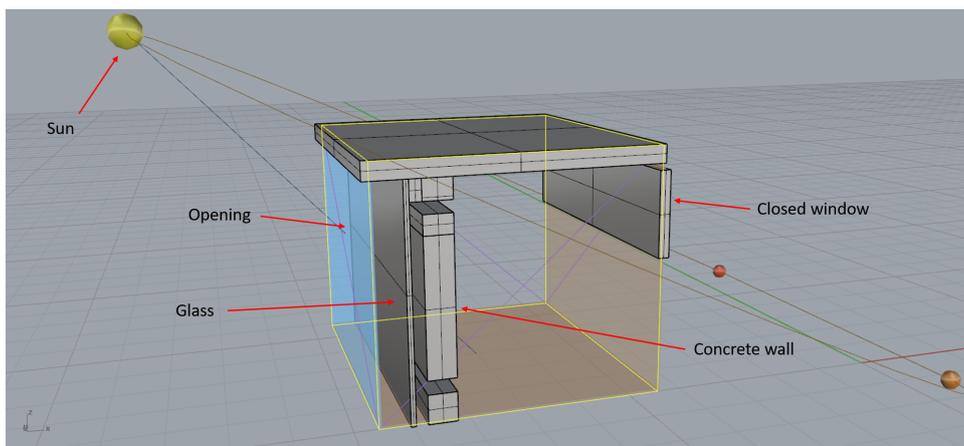
Figure 3: Geometry and domain definition

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 6000 seconds in 120 steps (Figure 4). Due to the slow changing flowfield expected in this type of simulation, these settings should be enough to achieve convergence at each timestep.
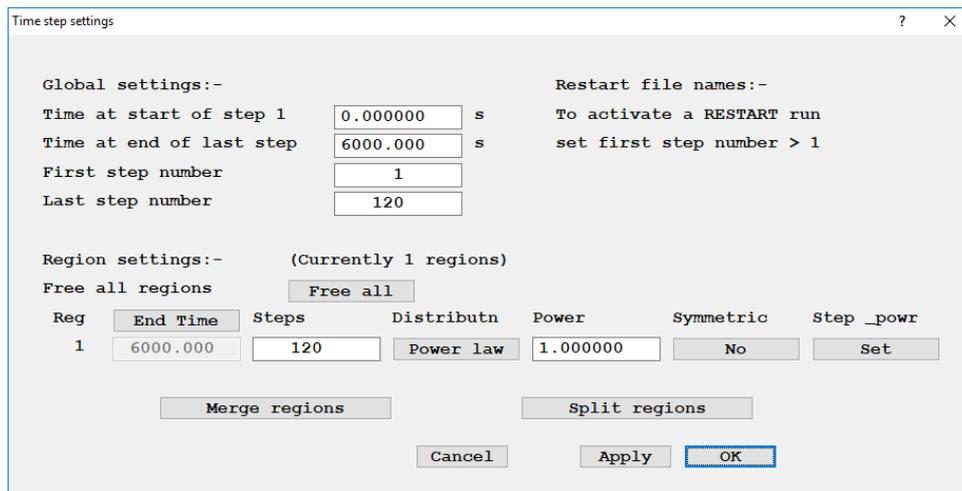
Figure 4: Time Step settings

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

Properties and Sources

In the properties section reduce the ambient temperature from 20 to 15 degrees to simulate a cold room. As we are relying on the temperature difference between the walls and the air to create the ventilation in the room, we will need to enter the Sources menu and change "Buoyancy effects turbulence" to "on". Leave all the other settings as default.

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).

Under the Relaxation panel the Automatic Convergence Wizard should be switched on. Simply edit the values for reference velocity and reference length (Figure 5). These should be set to characteristic values of the flow. In this case, a typical flow velocity for natural convection flows is in the order of $0.5m/s$ and a reference length of 1m should be suitable. See the RhinoCFD basics Convergence video for further information on relaxation.
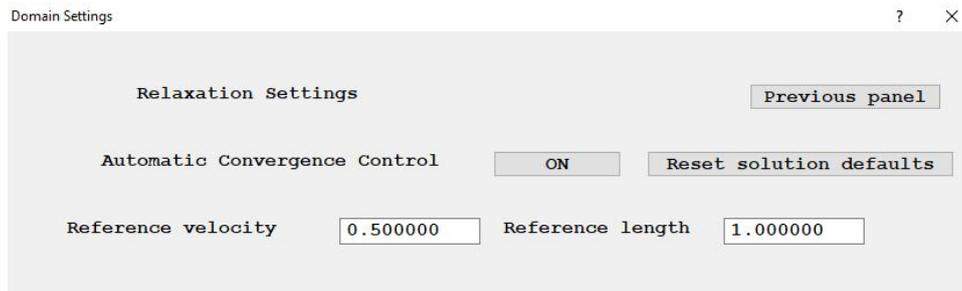
Figure 5: Relaxation setting using Automatic Convergence Control

Output

In the Output section under "Field dumping" set results to be output every other step (2). This should allow for the creation of a smooth animation of the flow in post-processing, but will mean less storage compared to storing every step; however, for a smoother animation you can decrease this to every step (1).

Boundary Conditions

Return to the top menu (by clicking on previous menu, then Top Menu) and click on "Domain faces". In this menu we can select the boundary conditions to be set on each of the domain faces. In this case we want to set the following:

- Min X Boundary: Opening - This provides an "outside" to the world

Technically this is the only domain boundary condition that is required. However, it is in general good practice to set relevant domain boundaries as walls, as they provide a source of friction that helps the simulation converge.

- Min Z Boundary: Wall - This sets a floor to our room

- Max X Boundary: Wall - This sets a floor to our room

Sun Object

Technically this is a boundary condition as well, which applies to the full domain. It can be accessed in the same "Domain Faces" panel we saw above. Activating it and clicking on settings brings up the following panel:

Figure 6: Sun Settings

The sun object is defined by several parameters. The first part is to "locate" our building. This is done by setting the latitude and the angle between where we want to define north and the y-direction in our domain. In this case -90 degrees and 65N for latitude work perfectly. We can also define the exact time of day we want our model to run at. This will determine it's position in the sky. In a transient simulation it will be the starting point of the simulation and will change as time progresses in the simulation.

The second step is then to define the intensity of the solar radiation. $800W/m^2$ is a typical value for direct sunlight. This will be applied to all the surfaces illuminated by the sun in our current configuration. This is calculated automatically. In this case we want to set the diffuse solar radiation to 0. This refers to indirect sunlight, the sort that comes from clouds and reflections from other places. It is applied to all surfaces in the model, but really is applicable only in external cases, such as if you're modelling sun on the outside of a house or city.

Clicking on "Optional Extra Output" brings up another panel which allows us to store information about the sun object. In this case we want to output LIT, which indicates what parts of our model are impacted by direct sunlight, and #QS2, which shows us the total heat per square meter that is applied on the illuminated surfaces.
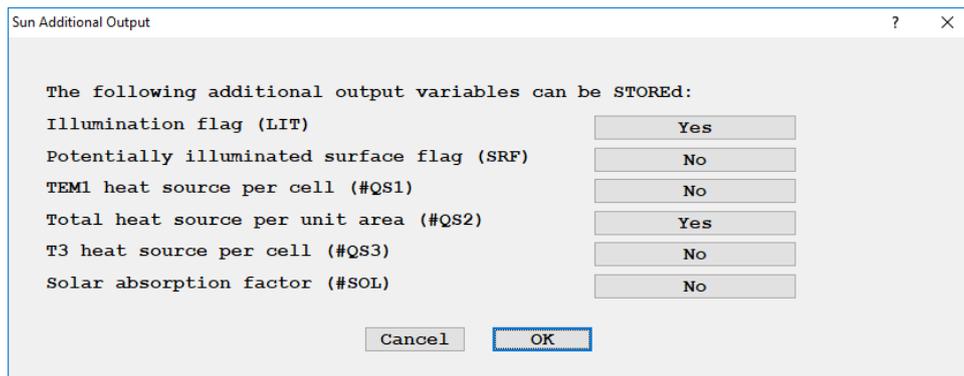
Figure 7: Sun Additional Outputs

Object Properties

At this point, we need to define the different materials that are involved in our model. In particular we want to set the front panel to be glass (that is, transparent to the solar radiation), and the wall behind it to be a conductive material, such as brick or concrete. By default, all objects are set to a non-participating material to start with, meaning they have no interaction at all with any heat sources.

Wall

To change the material of the thick walls (behind the glass), double click on each and click on attributes. Then set the material to Cast Concrete. Note that we can also set the emissivity and absorptivity. Set those to the values below:
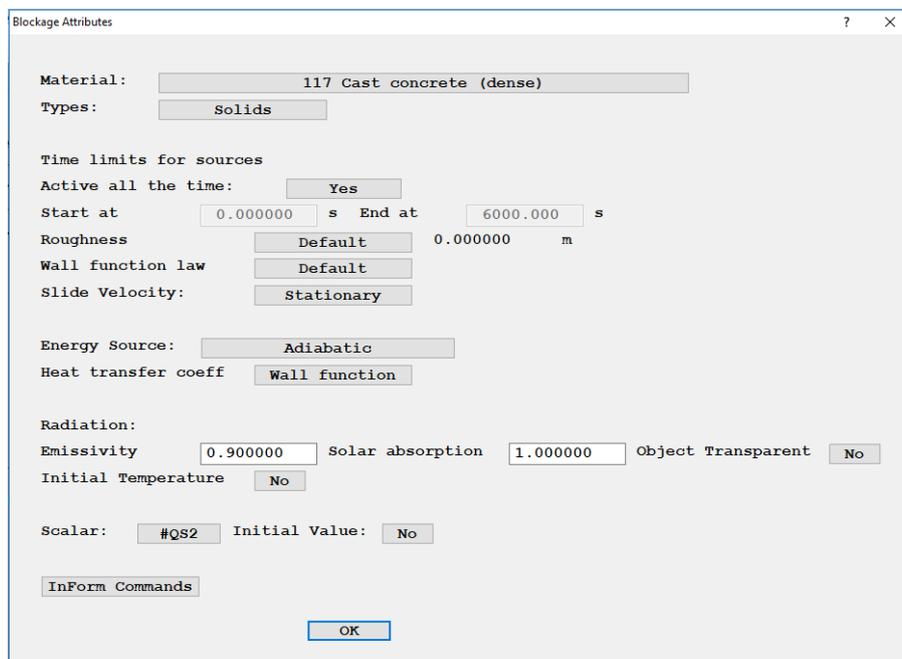
Figure 8: Trombe wall properties

Glass

The second object that needs changing is the glass panel. Double clicking on it we can change its material to glass so that it can conduct heat through to the outside world. We also want to set the object as transparent so that the sun can shine through it. Change 'No' to 'Yes' next to Object Transparent.

Back window

The final object we need to edit we should be aware of is the window at the rear of the building. This will remain closed for this winter heating scenario, but can be opened for summer cooling.

Probe

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

Meshing this scenario is quite straightforward, as we will only have to edit the grid the x and z axis.

The key concept here is to concentrate mesh where it is needed. In particular, that refers to the air gap between the glass and the concrete wall, and in the ventilation gaps around the concrete wall. In general, we want at least 5 cells in these areas to allow them to be represented correctly.

The limitations on RhinoCFD Lite in terms of meshing are 60x60x40 cells in the X, Y and Z directions respectively.

In this case we will use 57x1x40 in the following distribution (to bring up the panel, click on the third icon on the toolbar and then switch to manual meshing. You can edit individual regions by clicking on X-direction, Y-direction and Z-direction):
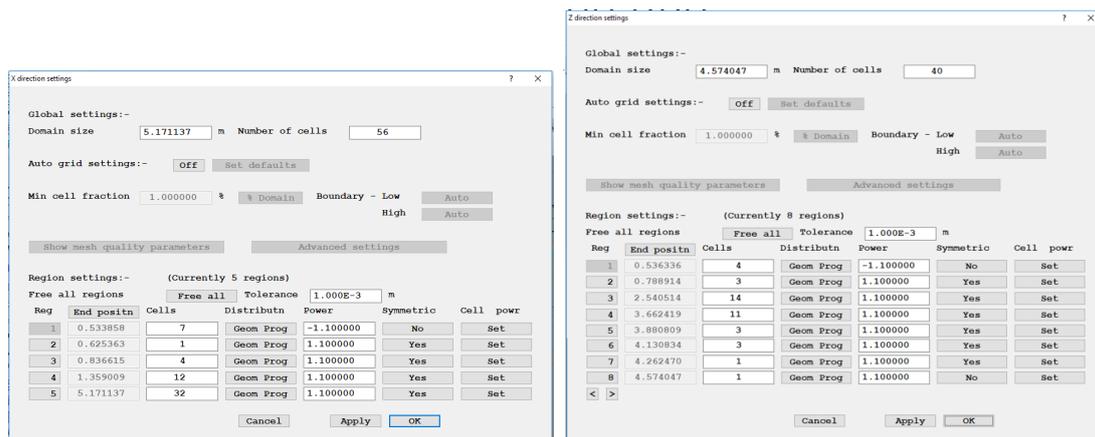
(a) Grid in X-plane   (b) Grid in Z-plane

Please note that if you have a greater number of regions in the x, y or z axis, then you will need to resize your domain so that it matches Figure 3. Users are welcome to use their own grid settings, ensuring the basic principles outlines about in our meshing video are followed.

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. This simulation will take under 5 minutes on a single processor.

Results

On the tool bar click 'Load Results' (9th icon) and click OK. Rotate the cut plane so it is in the Y plane and click on 'Play time series' (3rd from last icon) to play the results from all time steps. Figure 10 shows the heat patches applied by the sun on the external wall and can be selected by changing the variable to "heat source". The value displayed is in $watts/m^2$.
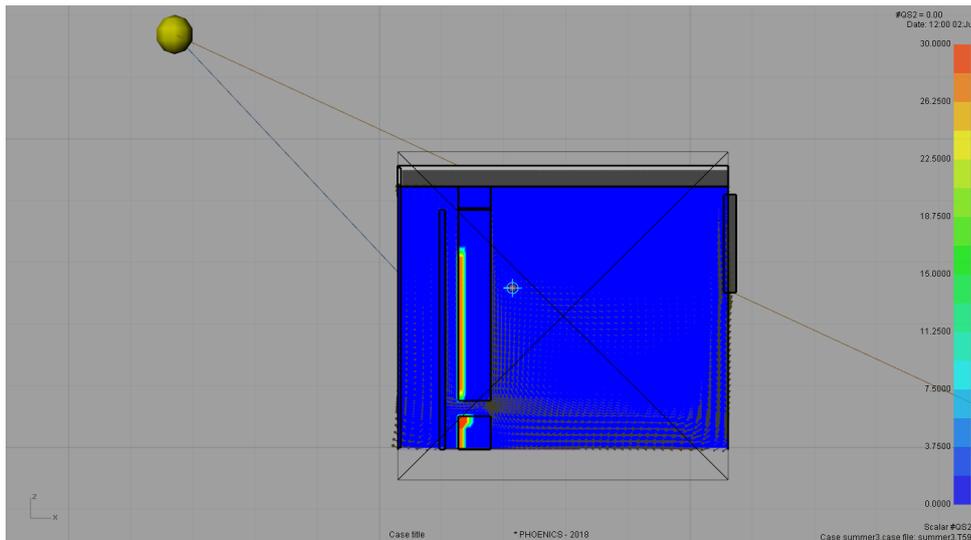
Figure 10: Heat per meter squared applied to relevant areas illuminated by the sun

We can also plot a contour of temperature inside the room. Here we can see that the heat from the sun creates an upwards current in the air-gap and heats the air which then returns to the room. Change the "Plane as" drop down to "Vectors and fill" to produce the aroows and colour plot.
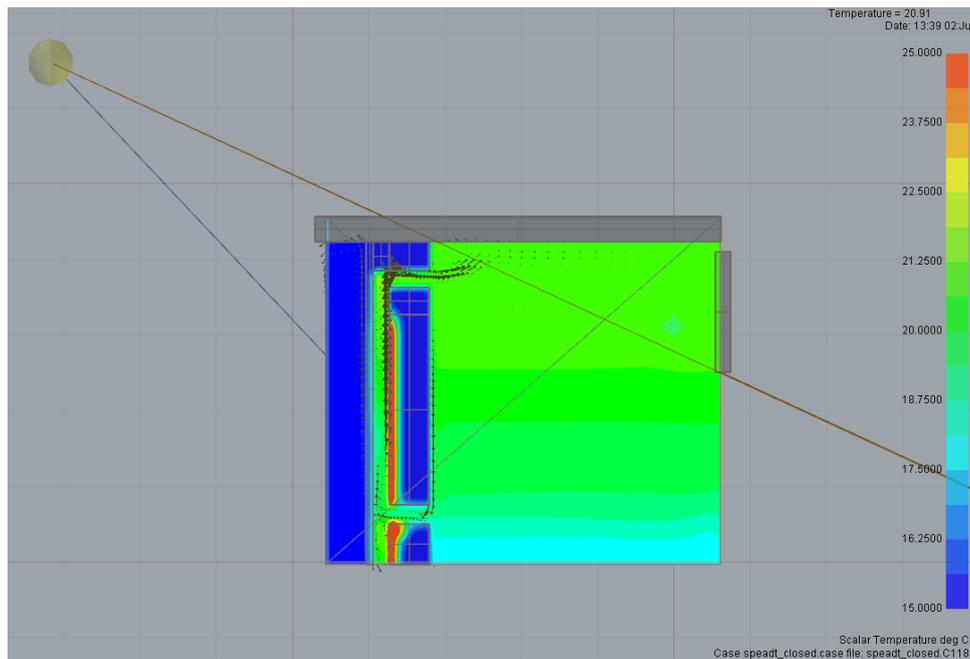
Figure 11: Temperature distribution in the room

Try changing the scalar to different variables such as velocity, radiant temperature and sun lit area to investigate the results further.

It is possible to create a video of your results. To do this right click on the 10th toolbar icon, enter a file name, select "Full Frames" and then OK to start recording. Click on the "play timesteps" icon to cycle through the timesteps and when these reach the end, right click again on the 10th tool bar icon to end the video.

Summer

Just as a note, it is possible to use the Trombe Wall design for heating in Summer as well. In this case, you change the window at the rear of the room to an "Opening", close the top gap in the concrete wall and open a window/gap in the glass. Geometry for this can be found in the Tutorials section of our website (called Summer Trombe Wall) and can use approximately the same grid settings as the winter case. You will also need to change the properties so that the ambient temperature is 25 and edit the rear window so that the outside temperature is 15 degrees to simulate air drawn in from a cooler room or shaded region outside.
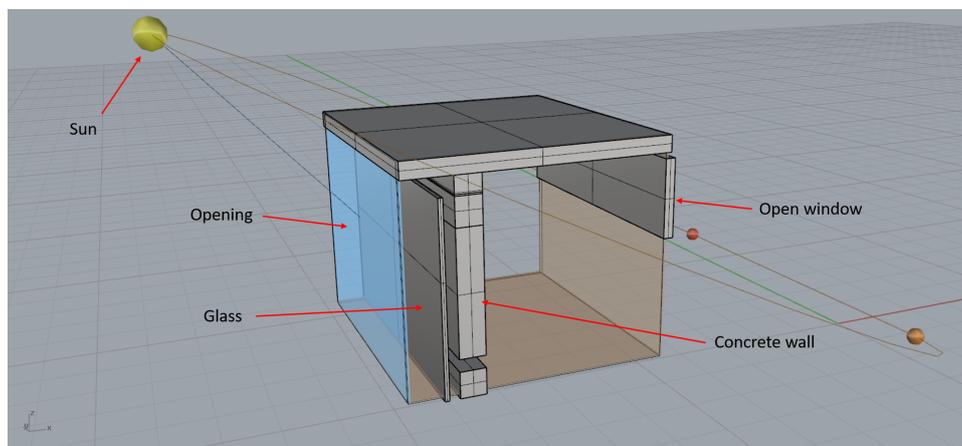
Figure 12: Set up for Summer simulation

Running the simulation as for the winter case will give the following results.
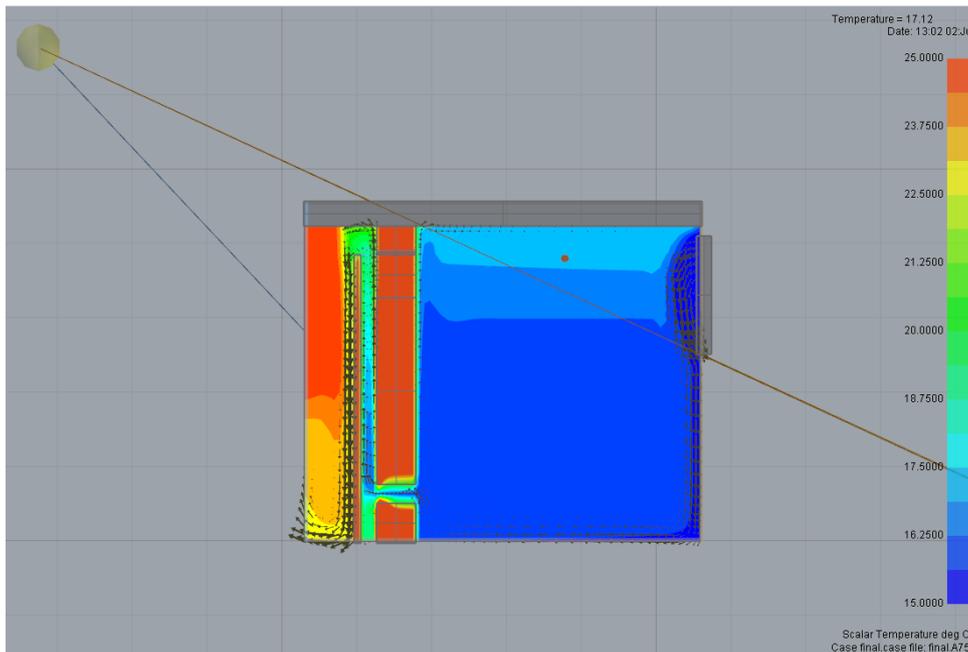
Figure 13: Temperature inside the room

We can also plot the radiant temperature (T3). This shows us how the concrete wall absorbs radiation from the sun and re-emits it into the room, contributing to it's heating as well.
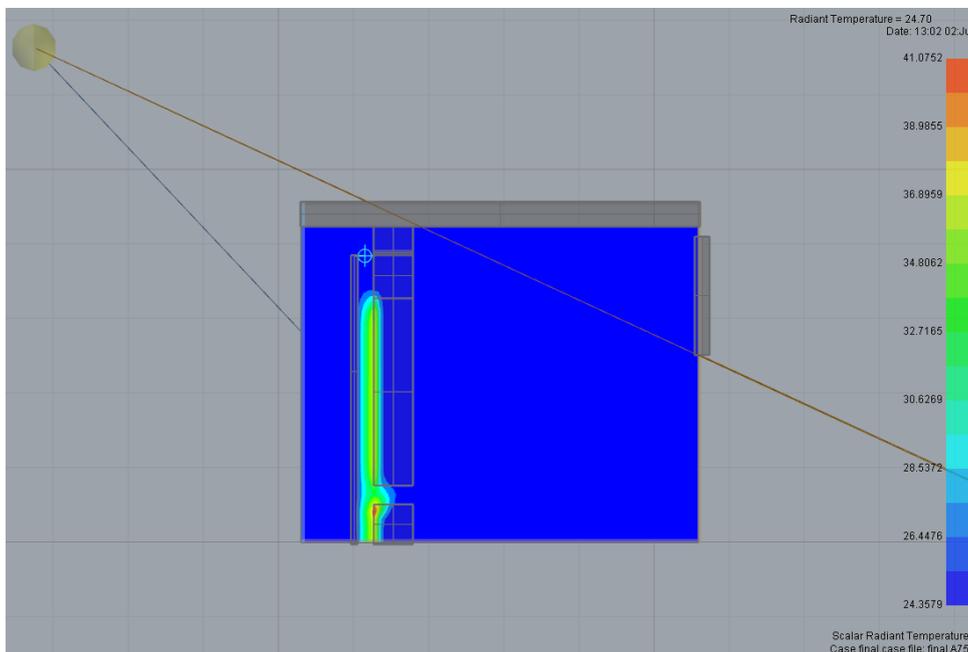
Figure 14: Radiant temperature inside the room

Finally we can look at the velocity in the in room:
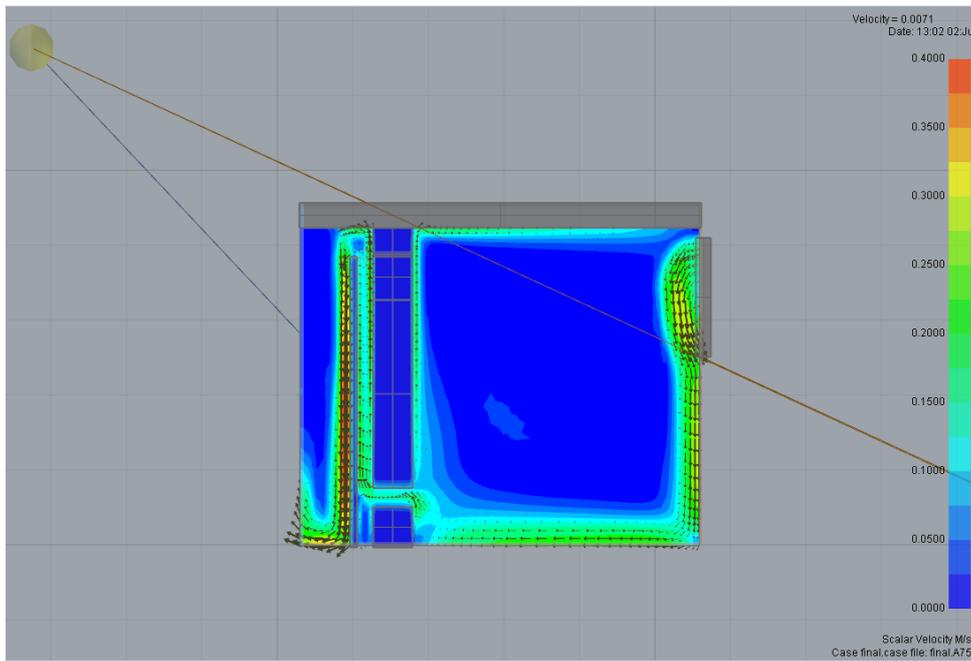
Figure 15: Velocity inside the room

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