



RhinoCFD Tutorials

HVAC Tutorial



RhinoCFD Official document produced by CHAM
September 26, 2017

Introduction

This tutorial is designed to get you up to speed on some of the capabilities of RhinoCFD's FLAIR extension. In particular, it aims to show how it can be used to easily simulate HVAC scenarios with heated flow inside an apartment. The .3dm file "Internal Flow" can be downloaded from [RhinoCFD Tutorials](#) for use in this case.

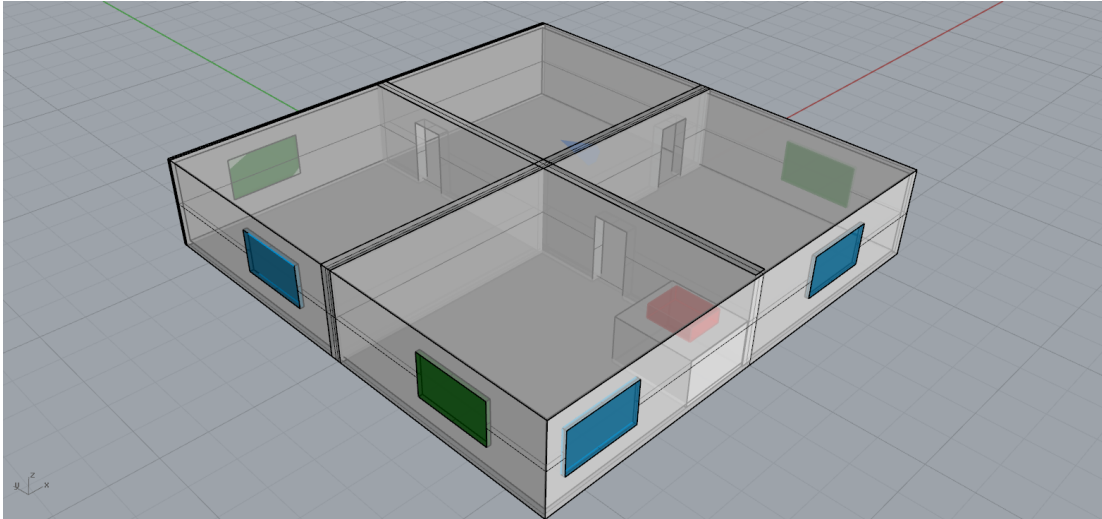


Figure 1: Internal flow geometry

The internal flow geometry contains a floor plan for the ground floor of a building, with several interconnected rooms with windows, and a "stove" located in one of the rooms, which is used to simulate a source of temperature and "smell" which will travel through the building. The flow pattern through the rooms is shown in the following figure. Air enters through the two windows on the right side of the floor plan, then travels through the rooms until it exits through the single open blue window on the left side.

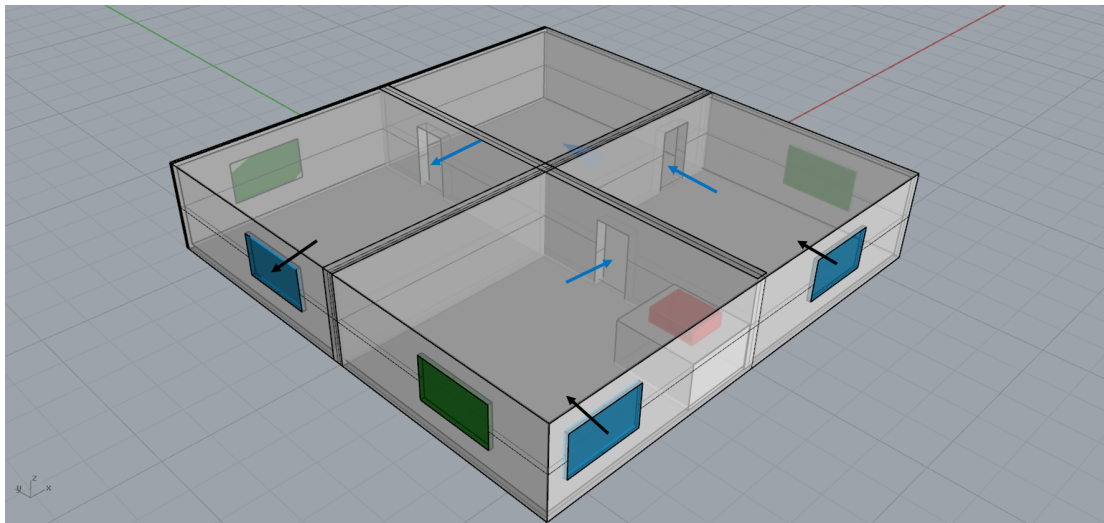


Figure 2: Flow pattern through the geometry

CFD Analysis

Open the .3dm file in Rhino CFD and create a domain by clicking on the first RhinoCFD toolbar icon.

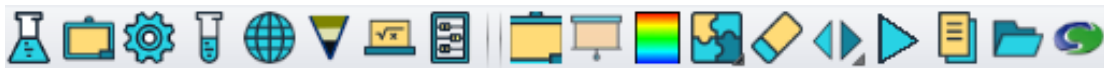


Figure 3: RhinoCFD Toolbar

Select the desired working directory and then select the menu option FLAIR when prompted.

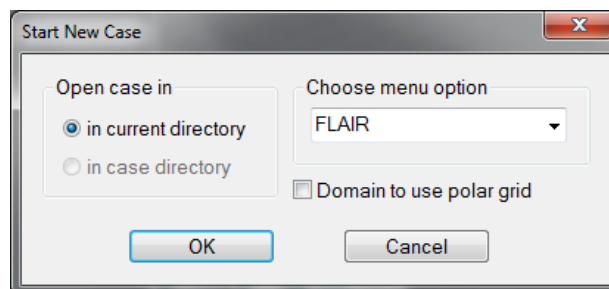


Figure 4: Menu Option: FLAIR

This will automatically change some of the default menu options from that seen in 'Core' in order to make setting up internal and external flow problems easier. Using the Gumball position the domain so that it is just inside the outside walls, roof and floor.

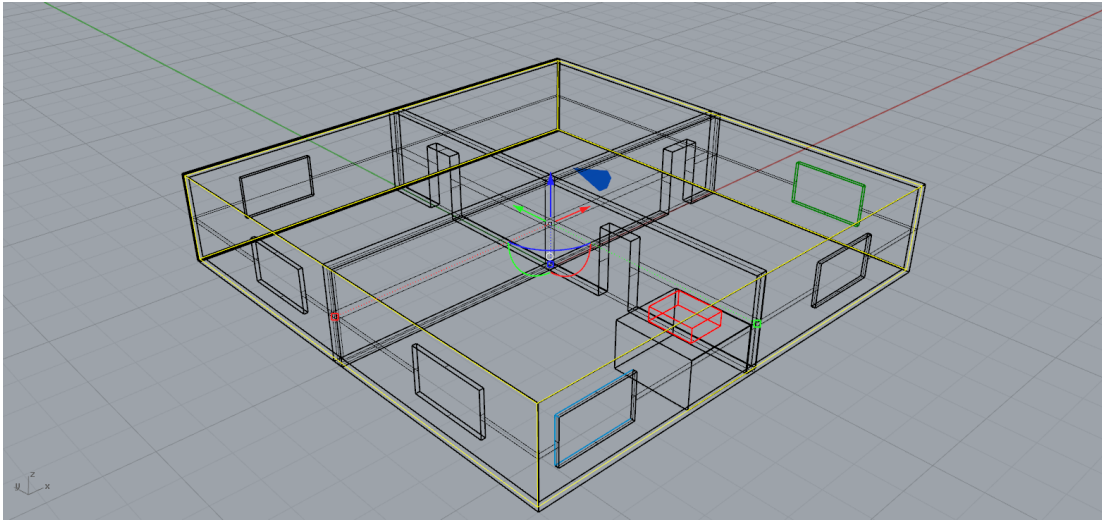


Figure 5: Domain sized to the apartment

The (C1) scalar variable

Scalar variables can have a multitude of uses, but in general serve as a tracker or marker device. They don't have any influence on the flow, but rather get convected and dispersed by it, thus making it a useful tool to establish how well a flow is mixed, track the spread of a particular contaminant or in this case, the smell coming from a stove.

To create a scalar variable go to the Main Menu → Models and then click on Settings next to Solution Control/Extra Variables. On the right hand side in the input box labelled 'Solve' type **C1** and click 'Apply'.

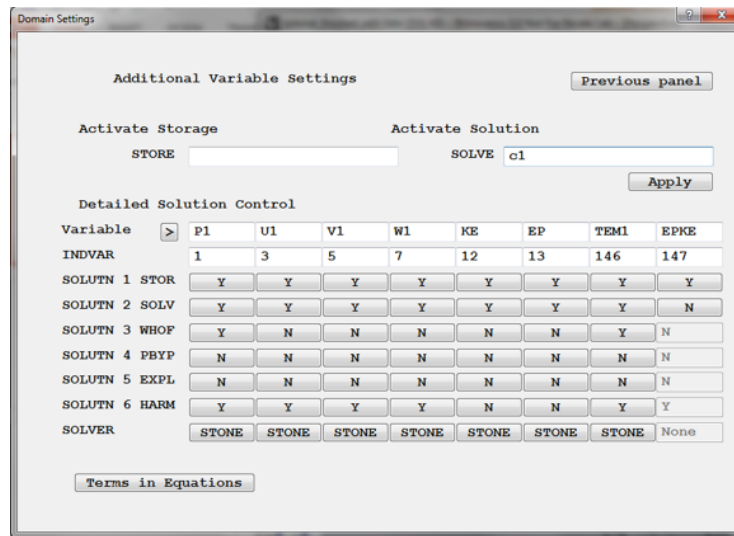


Figure 6: Adding C1 scalar variable to be solved

This will add **C1** to the list of variables below. It will have a Y next to 'Store' which will store all values of the variable across the domain and a Y next to 'Solve' which will cause the CFD solver to calculate the value of the variable across the domain. Ensure that the third button for this C1 scalar (in the WHOF row) is also set to Y. This is necessary to ensure proper convergence of the solution.

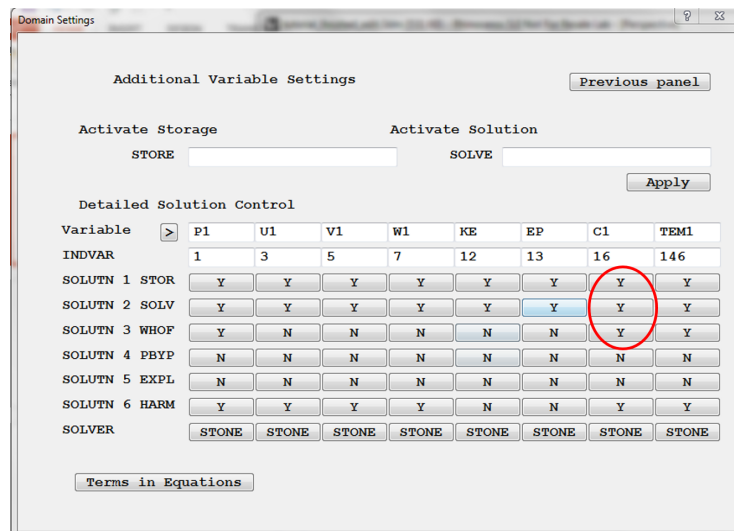


Figure 7: C1 Variable added to variable list.

Setting up flow boundary conditions

To complete the set-up of the simulation, we now need to input the boundary conditions. In particular, this refers to setting the windows to be inflows or outflows as well as setting a temperature and value of C1 to the stove.

Flow conditions at windows

- Select the extrusion called “IN1” in the room where the stove is located. Then click on the Cog in the toolbar (the third button along) to bring up its CFD properties.
- Change the object type to Opening. This represents a fixed-pressure boundary, which is most akin to a window. When asked which plane this object lies in select the Y plane.
- In attributes, set the exterior pressure to 2Pa. This means that if the pressure inside the room is less than 2Pa, flow will enter through the window. If it were above 2Pa, flow would actually exit through this window
- Repeat this process for the window in the next room called ‘In2’.
- Next edit the window called ‘OUT’. Change the type to Opening, then when prompted select X plane and then keep the attribute settings as default. The default external pressure will be set to zero. This means that the overpressure produced by the first two windows will drive the flow towards and out of these last two.
- The remainder of the windows do not need to be edited. They are simple blockages representing closed windows.


Stove boundary conditions

The stove will be modeled as a block of air that produces a certain amount of power (in Watts) and also be a source of C1. This is a good engineering approximation as we are mostly interested in how the flow is affected away from the stove in the other rooms.

- Select the block called "Stove" and access its CFD properties as mentioned above
- The object type should be set as Blockage. In Attributes, change the material type to Domain Material (this is the main fluid that is being solved in the simulation, which is defaulted to Air)
- We can now change the temperature source to be a Fixed Heat Flux, and set the power to 3500W

- Finally, set the Fixed Value of C1 to be 1. This means that when viewing the results, they will all be a fraction of this value, and in effect, represent the percentage of C1 in each area. Thus, if we think of the smell being 100% at the stove, when viewing the results, we can assume the values to be a percentage of whatever it was at the stove

Meshing

The next step is to ensure the mesh is adequate to resolve all the features in the simulation. Of prime concern are the interior walls, which are thin. A secondary concern is to ensure the mesh "looks good", that is, that there aren't sharp differences in size between one cell to the next. More information on RhinoCFD grids can be found in our [.Meshing User Guide](#) To modify the mesh, click on the fourth button on the toolbar .

RhinoCFD has an automeshing mechanism, so you will see a grid appear automatically. Its parameters can be modified manually by clicking on "X/Y/Z Direction".

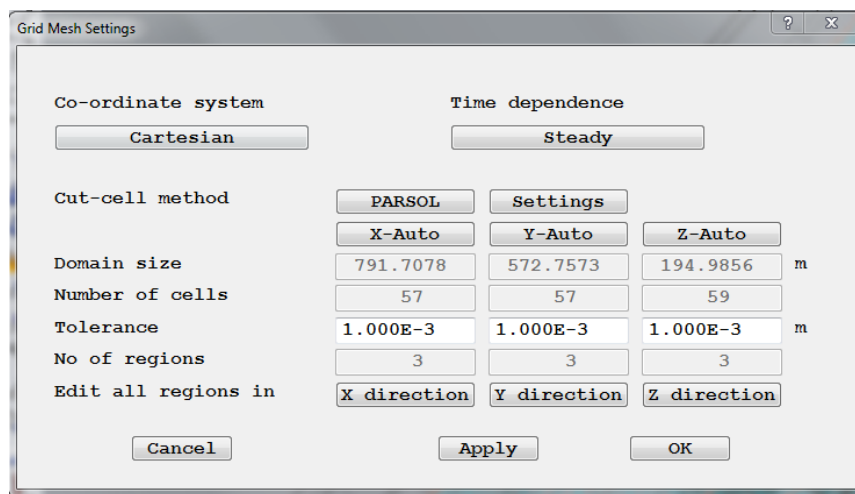


Figure 8: C1 Variable added to variable list.

Alternatively, the grid can also be modified simply by first displaying it (by clicking on the 5th button on the toolbar) and then double clicking on the mesh. This brings up a slider bar which can modify the mesh automatically in each direction.

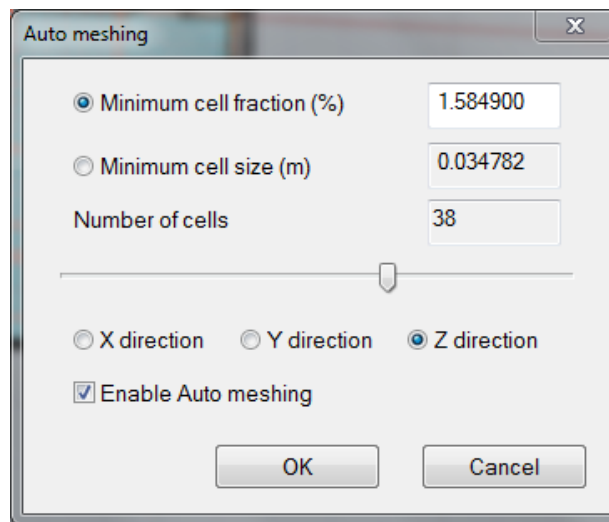


Figure 9: C1 Variable added to variable list.

To ensure this simulation runs quickly, use the slider or the mesh menu to get approximately 60 cells in the X and Y direction and 40 in the Z. (NOTE: If the slider is not allowing your mesh to go low enough, then enter the mesh menu enter any direction's menu and click on 'set default').

Running the Simulation

Next enter the Main Menu, by clicking on the second icon on the RhinoCFD toolbar. Click on Numerics → Relaxation and turn "Automatic convergence control" off. Set the relaxation to a false time step of 0.1 for the momentum components (U1, V1 and W1). These values are based on the average cell size and the average flow velocity. A general rule of thumb is to set the manual relaxation of momentum to the nearest order of magnitude to $\frac{Cellsize}{FlowVelocity}$.

For temperature, a good rule of thumb is to set the FALSDT relaxation value to two/three orders of magnitude larger than for momentum, so 100 in this case. C1 on the other hand works best with a value of Linear 0.5 or slightly lower.

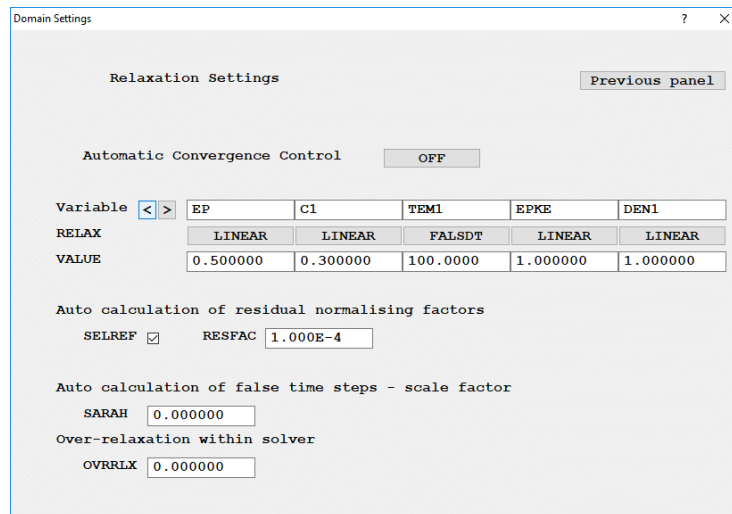




Figure 10: Relaxation Settings

For more information on relaxation parameters watch the [RhinoCFD Basics video](#) or read the [RhinoCFD User Manual: Convergence and Relaxation](#).

We can then commence running the simulation by clicking on the abacus on the toolbar (the last button before the dividers) .

Visualizing Results

Once the simulation is complete, we can visualize the results. These are loaded by clicking on the first icon on the toolbar after the dividers. A viewer options panel will appear where we can change the way the results are displayed, and add new result probes such as streamlines, iso-surfaces, line-graphs, and surface contours. These can also be added via the link on the toolbar (4th button after the divider) .

For more details on results visualizations see [Flow Past a Cylinder Tutorial](#) or the [RhinoCFD User Manual: Viewing Results](#).

All results shown in this tutorial have been plotted at the mid point height of the windows. Looking at the contour for velocity, as we would expect, there is an increased velocity through the doors, creating a draught region that connects the window in the kitchen with the one in the room farthest away. The doorway connecting the first room to the second shows a lower velocity than the other two door ways as only half the mass flow is travelling through. It also shows regions of poor circulation with more stagnant air, identified by the dark blue in the third and final rooms.

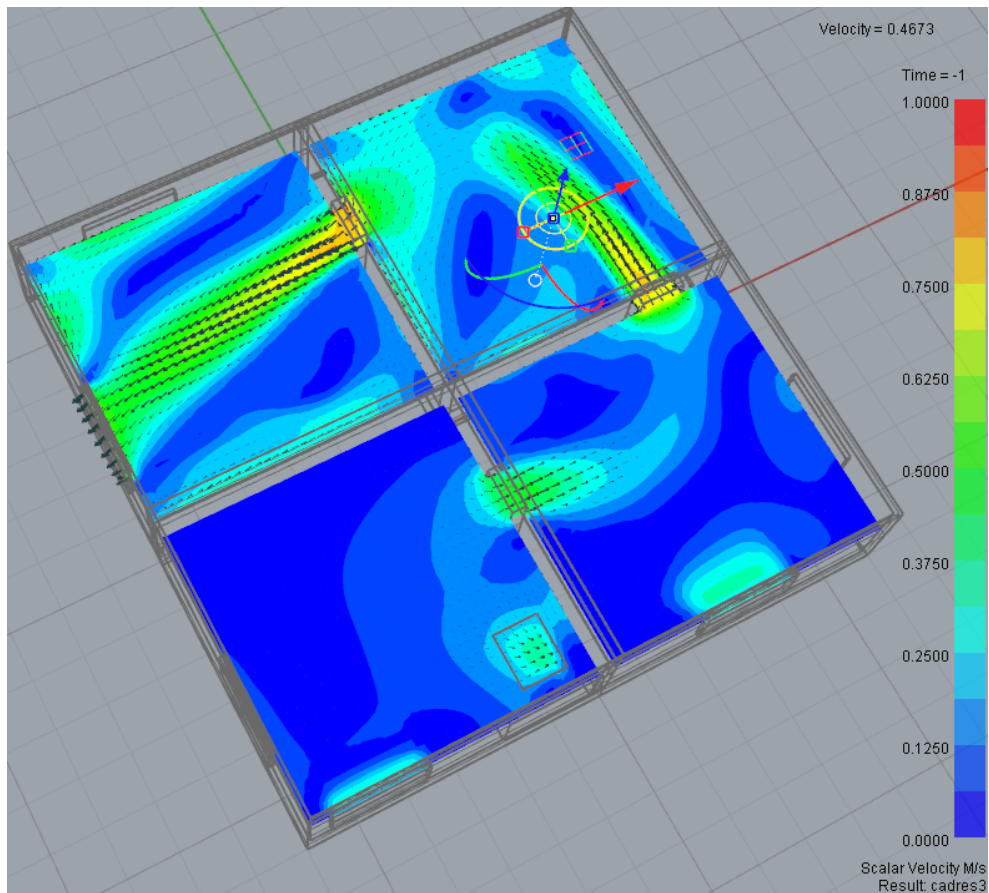
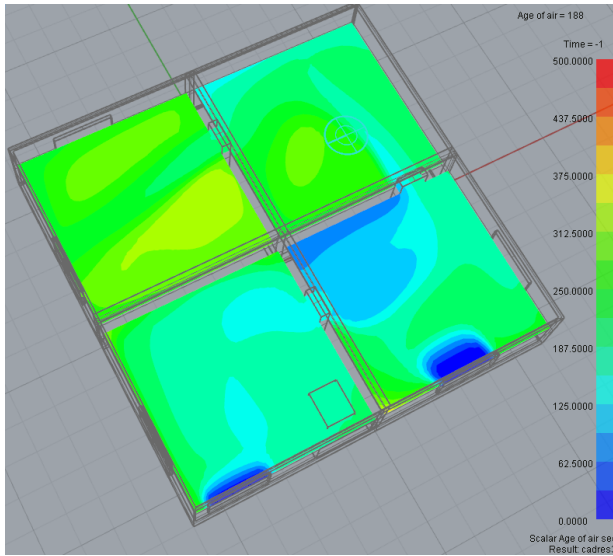
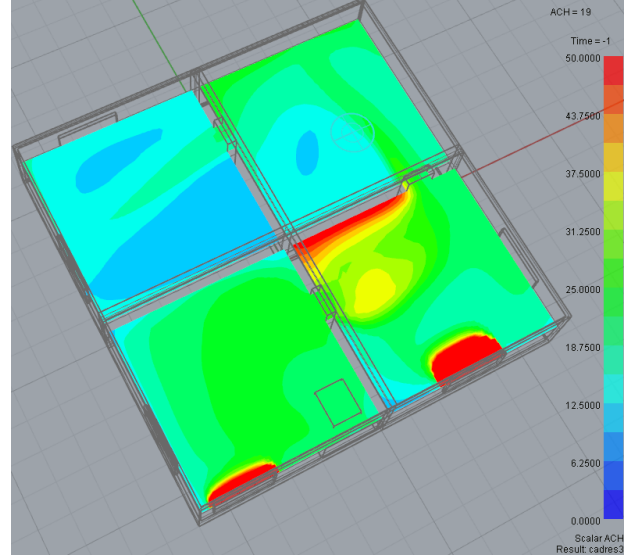


Figure 11: Velocity contours across main building with different scales

To see the consequences of this velocity distribution we can plot the Mean Age of Air (MAA) and Air Changes per Hour (ACH). The Mean Age of Air is essentially how old the air in each area of apartment is, and the Air Change per Hour is the number times the air is completely changed/recycled per hour. These values are essentially the inverse of each other i.e. where MAA is high the ACH is low. High values of MAA (and low ACH) correspond to regions where the air is stagnant. These regions can be seen in the final room to the left of the door way, as well as just to the left of window in the second room. Regions where the air is recycled regularly and have a higher velocity correspond to the low MAA (high ACH). These regions can be seen at the open windows and around the doorways.




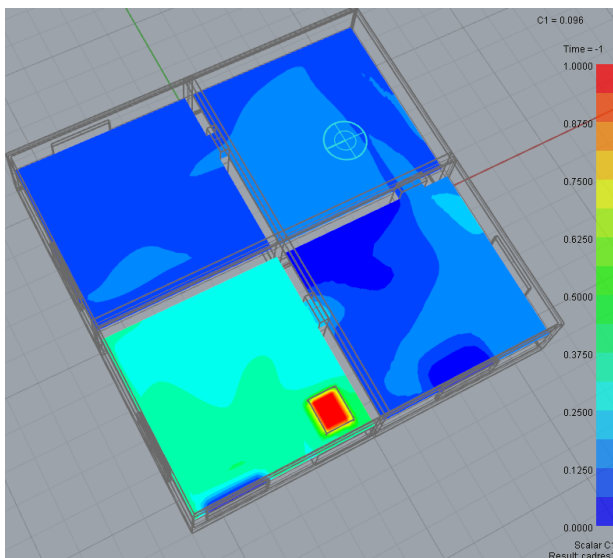
(a) Mean Age of Air



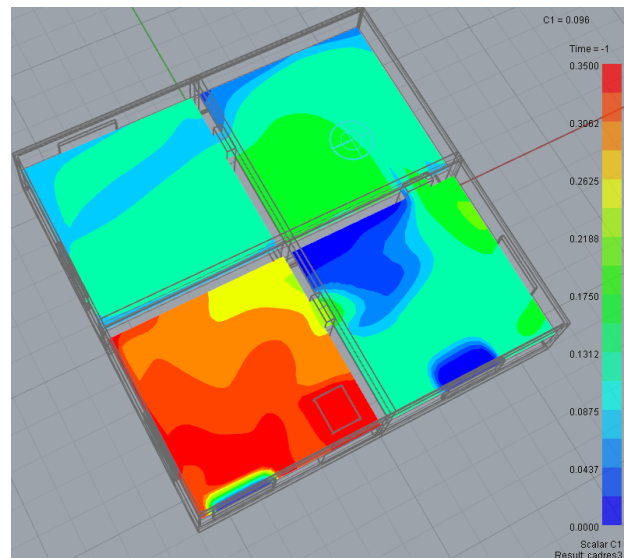
(b) Air Changes per Hour

Figure 12: Mean Age of Air and Air Changes per Hour

Likewise, if we now plot C1, we can see the areas where the "smell" propagated through the house. It is best to change the scale slightly so the maximum value is 0.3-0.5 or 30-50% of the initial value. This will give us a more visible gradient scale as can be seen in the following figure. To show the scale key on the plot click on the relevant toolbar button .



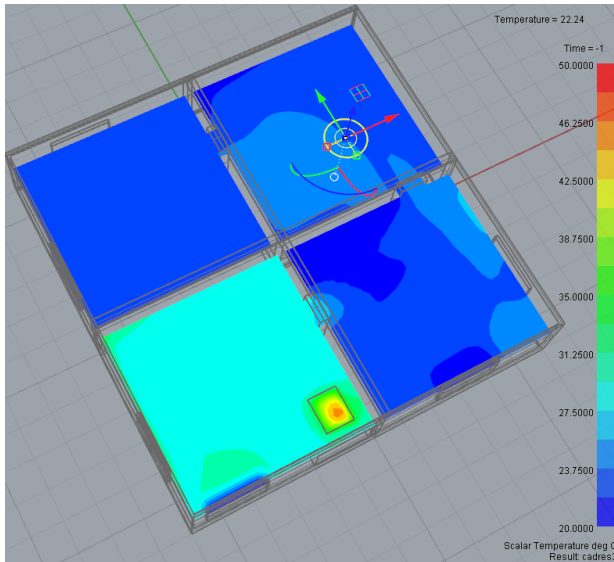
(a) Maximum C1 Scalar Variable



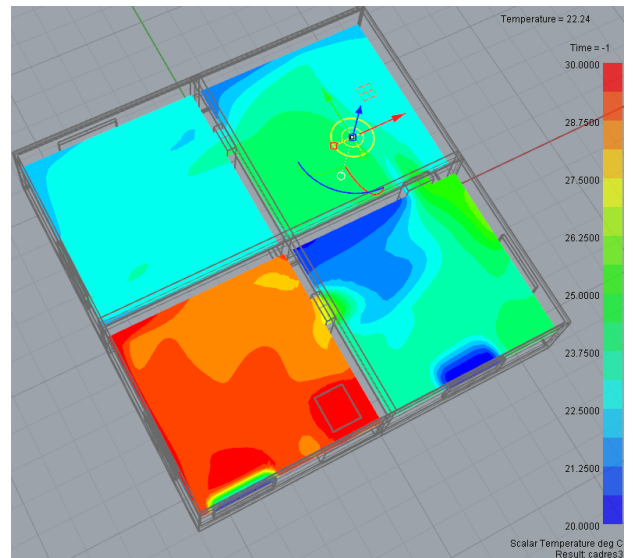
(b) Scaled C1 Scalar Variable

Figure 13: C1 Scalar variable with different scales

Finally, we can change the variable to temperature and find its distribution. The heat source is shown to predominately heat the room which it is in, but heat is also convected into the second and third rooms, shown by the green regions. In the fourth room the air is only just above room temperature and has not been greatly affected by the heat source.



(a) Maximum Temperature



(b) Scaled Temperature

Figure 14: Temperature Plots with different scales



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