RhinoCFD Tutorial

Heat transfer in a Heat Sink

RhinoCFD Official document produced by CHAM
October 13, 2021

Introduction

This tutorial demonstrates how to set up a simulation involving heat transfer between solids and fluids. It also demonstrates how RhinoCFD can be used to simulate conductivity through solids.
The geometry required for this tutorial can be be found here or be created quickly in Rhino.

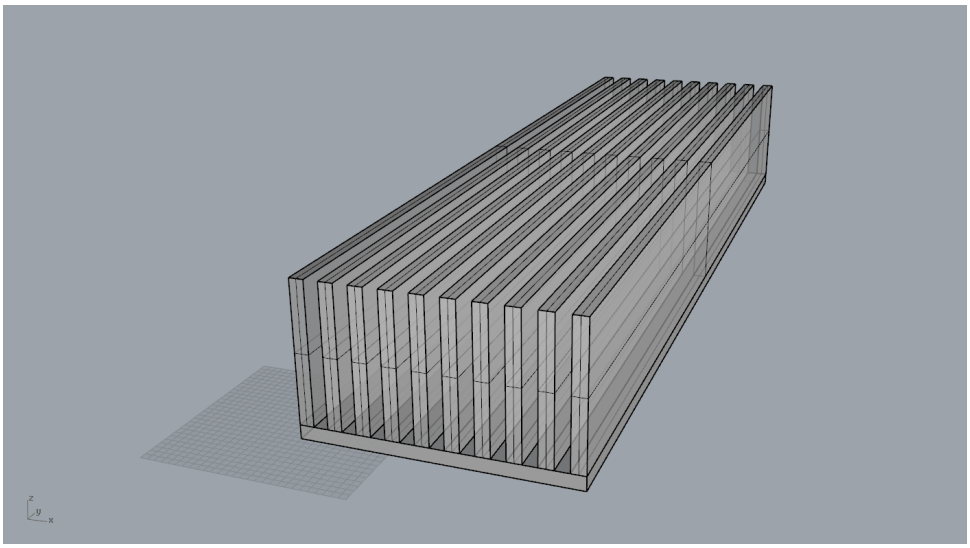


Figure 1: Heat sink geometry. Ten fins attached to a solid base

The resolution in this tutorial has been chosen to fit within the maximum cells per direction permitted in RhinoCFD Lite (i.e. 60 x 60 x 40). Other users may wish to increase the resolution to get more detailed results, in particular between the heat sink fins, but should keep in mind that the simulations will take longer to run.

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 FLAIR as the main option and then the desired working directory. FLAIR is optimized for heat transfer simulations and automatically adjusts some internal settings and parameters to make setting up the case easier.

To get a bit better resolution around the fins, resize the domain to be about $x = 300mm$, $y = 1100mm$ and $z = 250mm$, and centre the geometry in the domain.

Figure 2: RhinoCFD tool bar

To set up the boundary conditions of the simulation, right-click the second button on the tool bar (Edit Domain Edge Conditions). Specify the 'Ymin' face to have a Flow condition with a velocity of $1m/s$ in the Y-direction and the opposite face (Ymax) to be Open (this implies that RhinoCFD will automatically determine how much flow will go in or out of this end based on the reference pressure set for the outlet and the calculated pressure in the domain). 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.The 'Wall Condition' applies roughness to these faces, which helps the turbulence model converge. If these conditions aren't set, the boundaries are treated like mirrors, which reflect the flow pattern near them. These menu settings are shown in Figure 3.

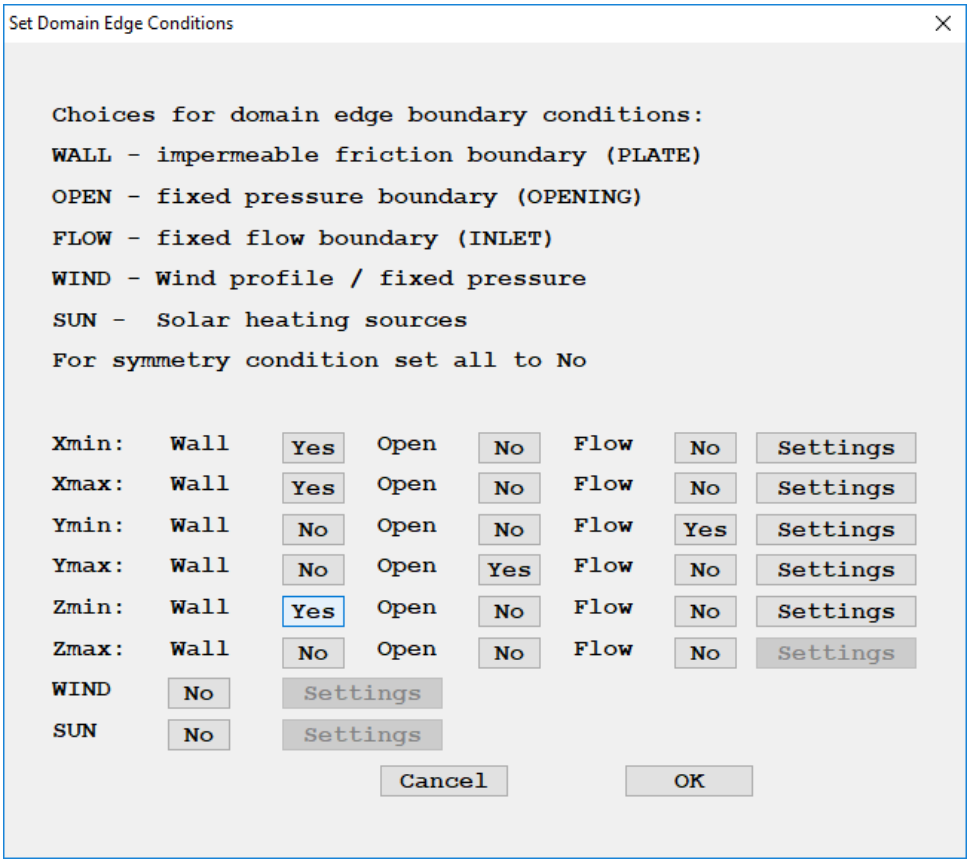


Figure 3: Domain Edge Conditions

Main Menu

Models

In the Models section, click on Turbulence Models and select 'LVEL'. This is a proprietary turbulence model which was specifically developed for narrow flow channels, such as heat exchangers, electronics components or flow in pipes. It is also ideally suited for heat transfer scenarios.

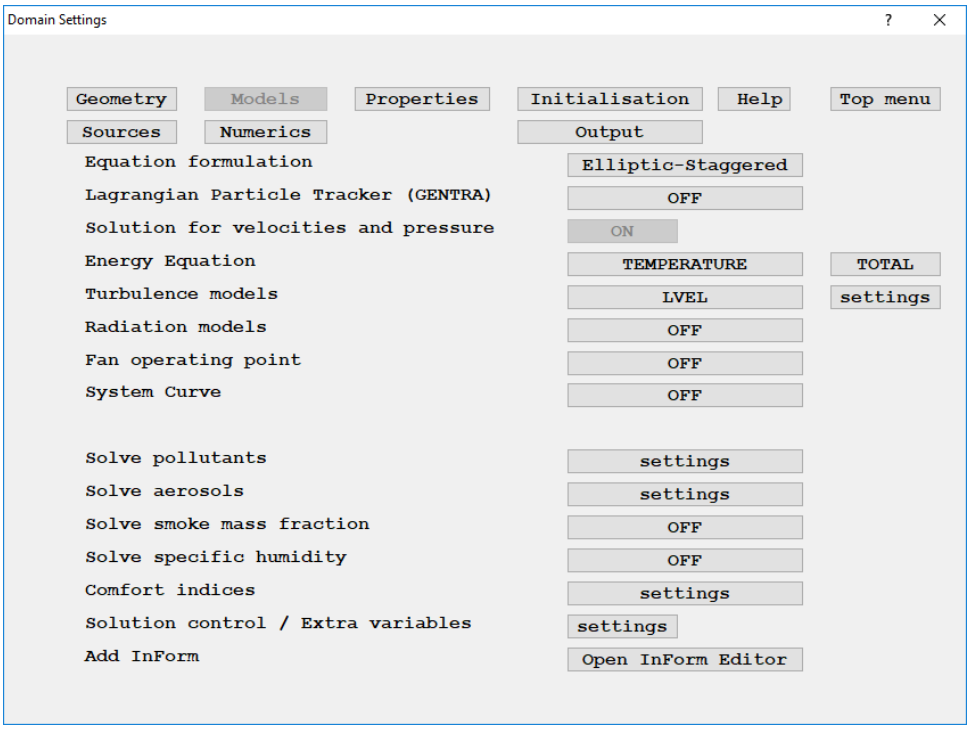


Figure 4: Model settings

Properties

In the Properties section FLAIR will have automatically set the fluid settings to the Ideal Gas Law for Air. This setting is necessary whenever temperature is being calculated, as it allows the fluid to change it's density (which in turn allows it to be affected by buoyancy -so hot air rises-). You can see the buoyancy settings by clicking on the 'Sources' panel.

Sources

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

Numerics

In the Numerics section set the number of iterations to 500. Select the Relaxation control button and edit the values for reference velocity and reference length (Figure 6). These should be set to characteristic values of the flow, the velocity could be set to the inlet velocity and the reference length to the height of the fins. Manual relaxation can also be used. In general it is a little quicker than the automatic setting, and can be set up by following a few 'rules of thumb':

- Relaxation on momentum (U1, V1 and W1 velocities) should be relaxed as FalsDT to a value of reference velocity divided by an average cell size. 0.1 for all three would work in this case
- When conducting solids are present, set TEM1 (temperature) relaxation to Linear = 0.3

See the RhinoCFD basics Convergence 4-minute video for further information on relaxation and convergence.

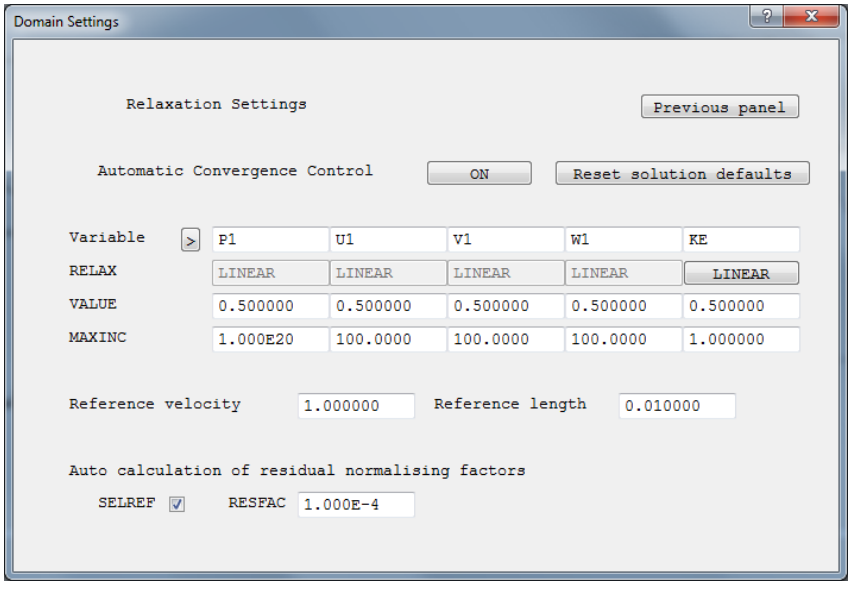


Figure 5: Relaxation settings using Automatic Convergence Control, given a Reference Velocity and Reference Length

Object Properties

Double-click on the base plate and then go to attributes. Click on 'Other Types' and select Steel. You can then set it as an energy source. In this case select 'fixed temperature' and type in 70^oC. The initial temperature isn't strictly important for steady state simulations, but setting it to 70^oC might help it converge slightly faster.

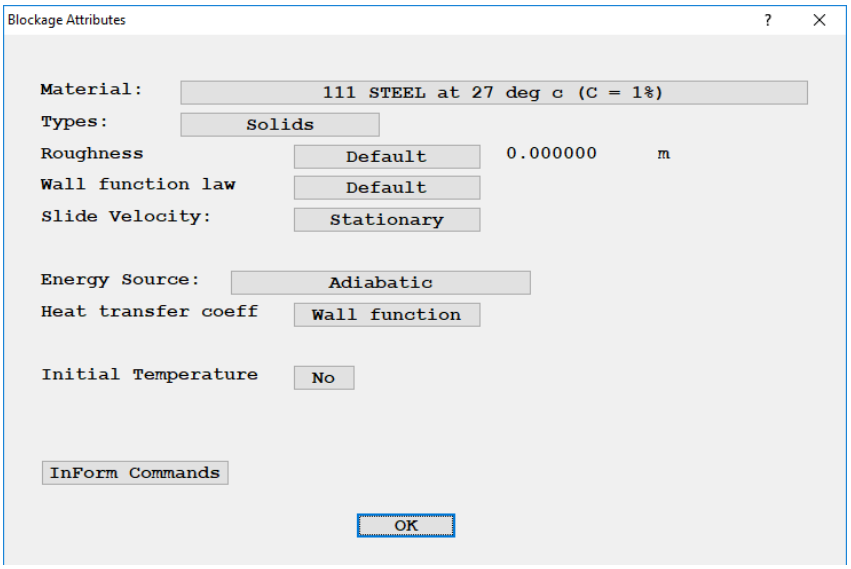


Figure 6: Attributes panel to set individual material and temperature properties to the base and fins

Repeat this process with the vertical fins. In this case though, change the material to Copper, and leave the Energy Source as Adiabatic. This will allow them to conduct heat, but will not pose a source of heat themselves. Feel free to set various material types to see how they conduct heat differently.

Finally, locate the probe using the RhinoCFD tool bar by left-clicking the sixth tool bar button Show Probe, then positioning it in the airspace just behind the heat sink. This way the simulations monitor plots will show changes in quantities at this point, which is likely to be slightly turbulent. When looking at the monitor plot as the simulation is running, we ideally want to see these values stabilize and the lines flatten out.

Meshing

The CFD grid is created automatically based on the geometry in the domain. However, we can refine it to make sure we have the best distribution possible for accuracy and simulation time. To do this, click on the 4th icon on the toolbar to bring up the Mesh menu. The region we are interested in the most is the X-Direction, as we want to make sure there is enough grid to accurately capture the fins and the gaps between them. Clicking on the X-Direction we can set the Auto Grid Settings to 'Off'. Ideally we want a minimum of 3 cells between each of the fins and either 1-2 cells across the thickness of each fin. We can change this by setting the number of cells in the relevant region. You can tell which one you're editing as the cells will turn dark blue when you click apply.

NOTE: The maximum amount of cells allowed in RhinoCFD Lite for FLAIR is 60x60x40. Ensure your total number of cells is under these values if you're using RhinoCFD Lite.

After applying these setting the grid should look like this:

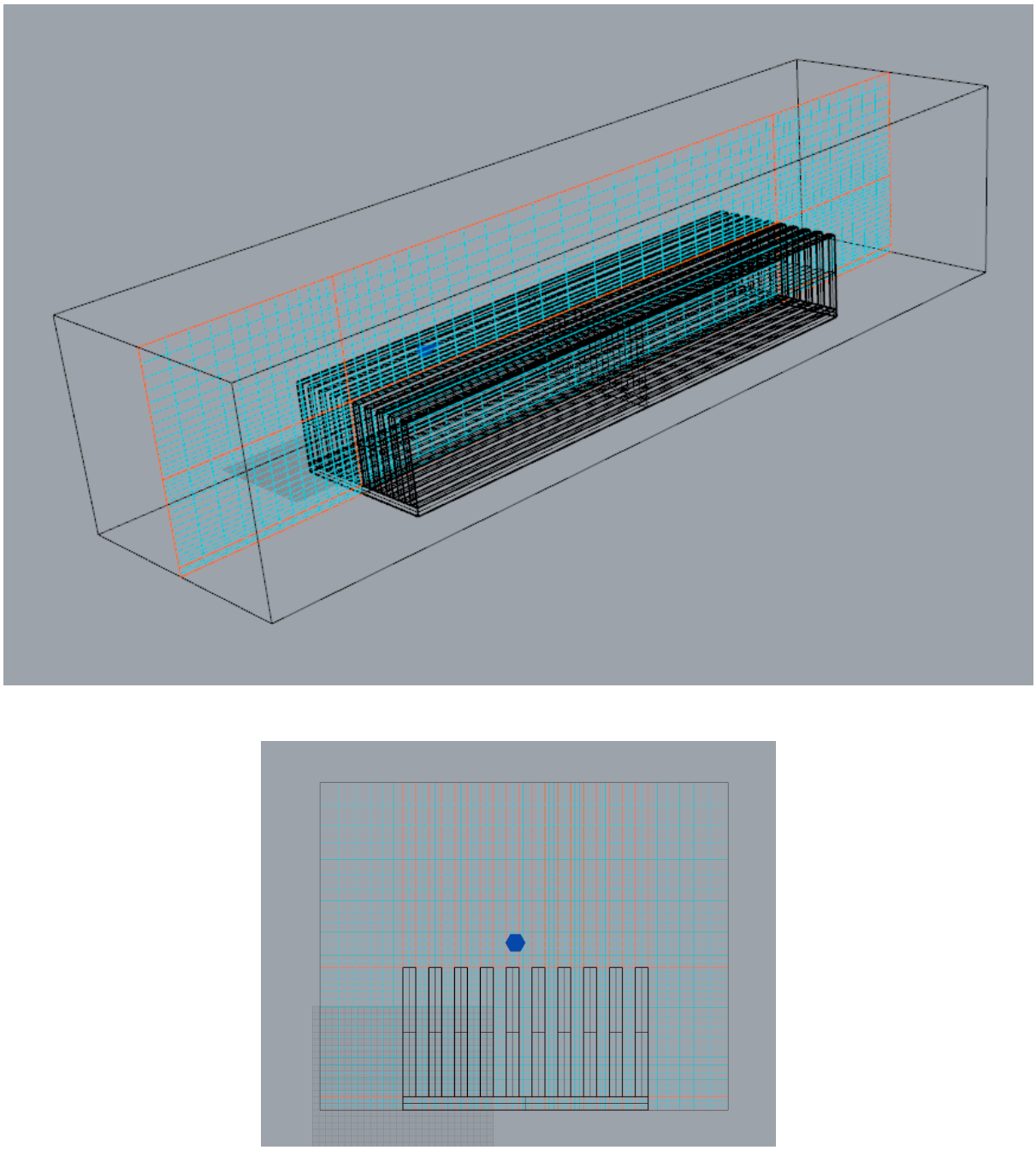


Figure 7: Grid

Results

On the tool bar click 'Load Results' and click OK. You'll be presented with a cutplane probe which you can move around and rotate with the Gumball. On the results panel that appears, you can select which variable is being plotted. The default variable is pressure. Here we can see that it increases in front of the heat sink as it encounters resistance.

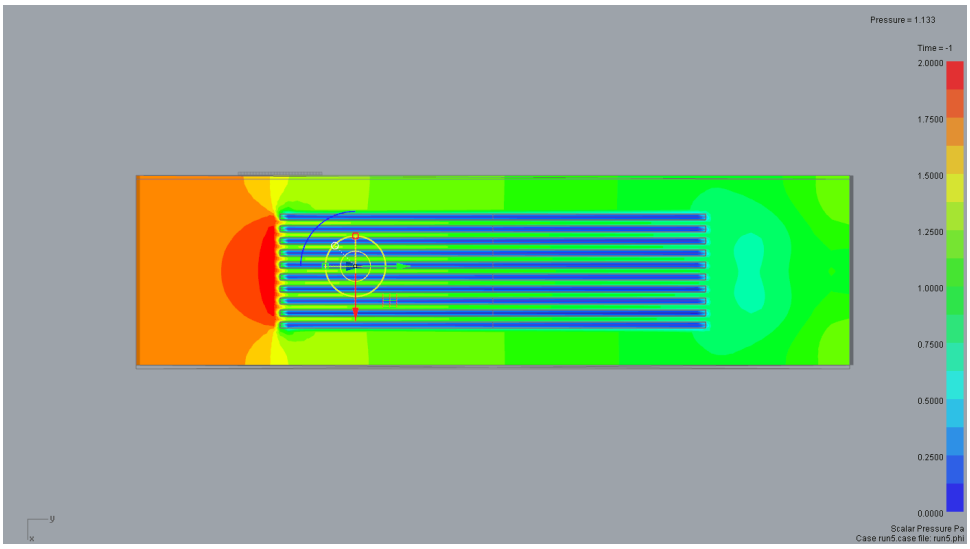
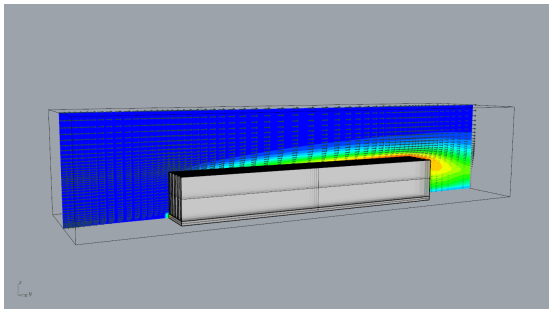


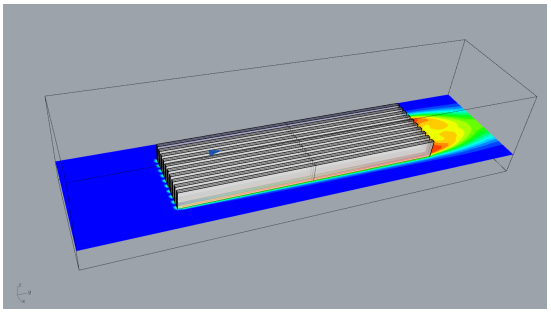
Figure 8: Grid

Changing the displayed variable to temperature and playing with the orientation of the cutplane we can now see the evolution of temperature both in the air and within the fins. If you've selected different materials for each fin, you'll notice a less homogeneous result.

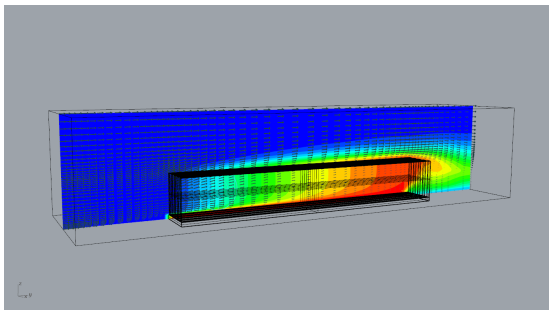
(a) Temperature in Y



(c) Temperature in Z



(b) Temperature in Y (wireframe)



(d) Temperature in Z (wireframe)

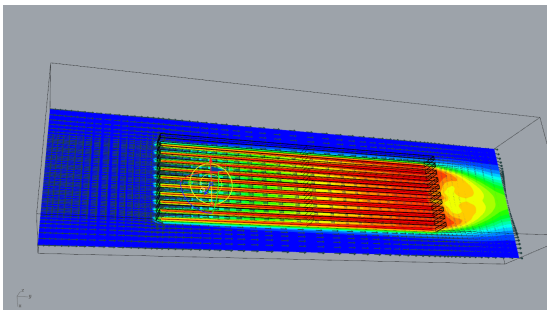


Figure 9: Contour plots of temperature through the heat sink

If we now open the result file by right-clicking on the third-last icon on the toolbar,

scrolling to the bottom of the file we can see the sources of temperature. The base of the heat sink is indicated as object B17. The value of Tem1 indicated is given in Watts. Above this, we can see the amount of heat in watts that is brought in by the inflow, and the amount that exits from the outlet. The values in brackets are the average temperatures in Centigrade of the flow entering and exiting the domain through the boundaries. Here we can see that the average temperature out (at DOM_YMAX_O) has increased from 20^oC to 25^oC

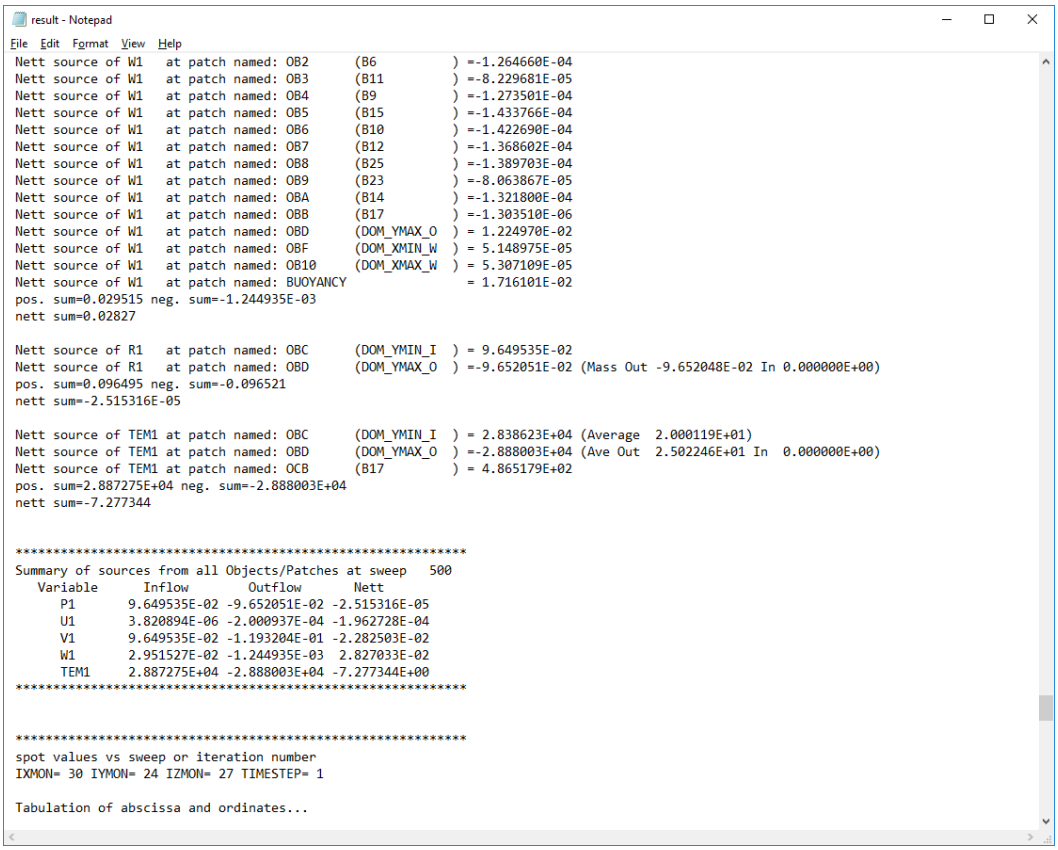


Figure 10: Heat released from heat sink

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