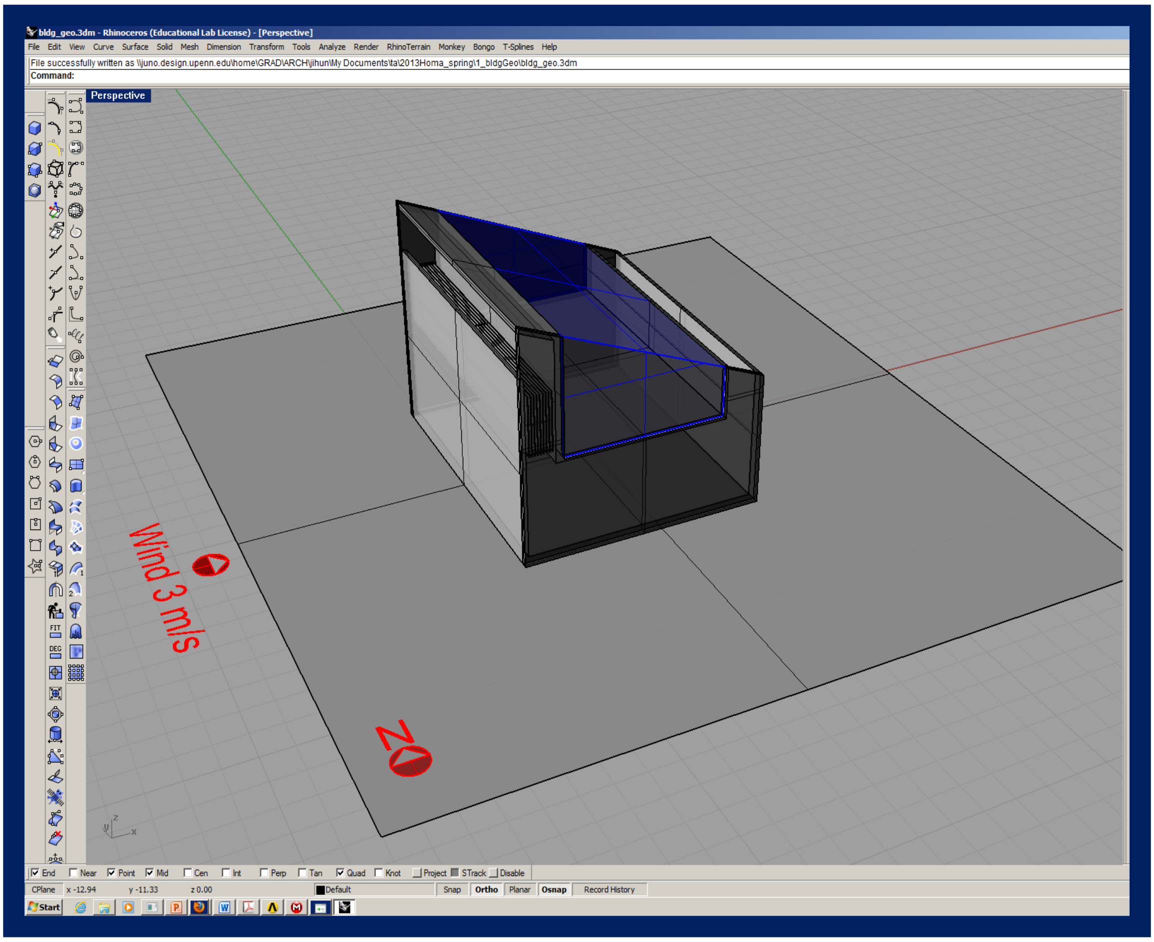
PHOENICS Case Study: HVAC
Air Flow and Solar Radiation

PHOENICS/FLAIR, with a structured computational mesh, was employed, by the T C Chan Centre for Building Simulation and Energy Studies in Pennsylvania, USA, to create the following replication of a pseudo-3D tutorial example for the built environment.



Modelling goals:

1. Outdoor wind flow into indoor space.
2. Heat source to change the inlet temperature of wind as application of modern wind tower.
3. Solar radiation outside as another heat source from natural system → Wind Speed and Distribution of heat to indoor and surrounding.

Although it is straightforward to set up the geometry of such a case using PHOENICS 'primitives', in this scenario the building design in question has been imported from CAD as a DWG format file.

The facility exists within PHOENICS/FLAIR to read weather data prevailing for any particular location from an external data file such as EnergyPlus. In this example, the wind direction and solar gain values are individually specified via the menu. The 2D representation of the building in question is as follows:

Concentration Heat and Momentum Limited, Bakery House, 40 High Street, Wimbledon, London SW19 5AU, UK
Telephone: 020 8947 7651 Fax: 020 8879 3497
Fax: 020 8879 3497
E-mail: phoenics@cham.co.uk, Web site: http://www.cham.co.uk

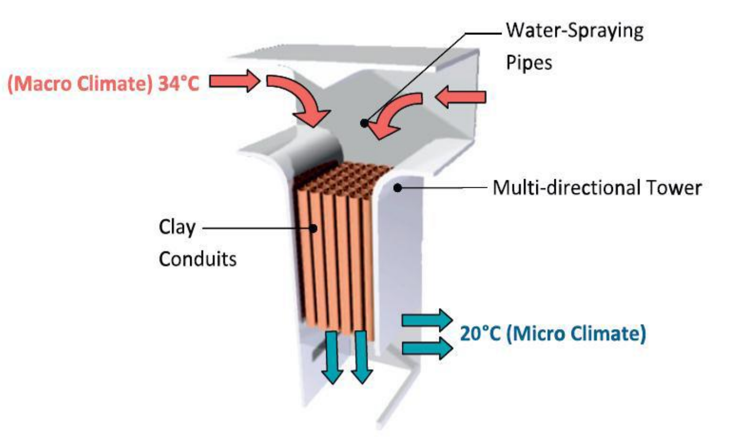
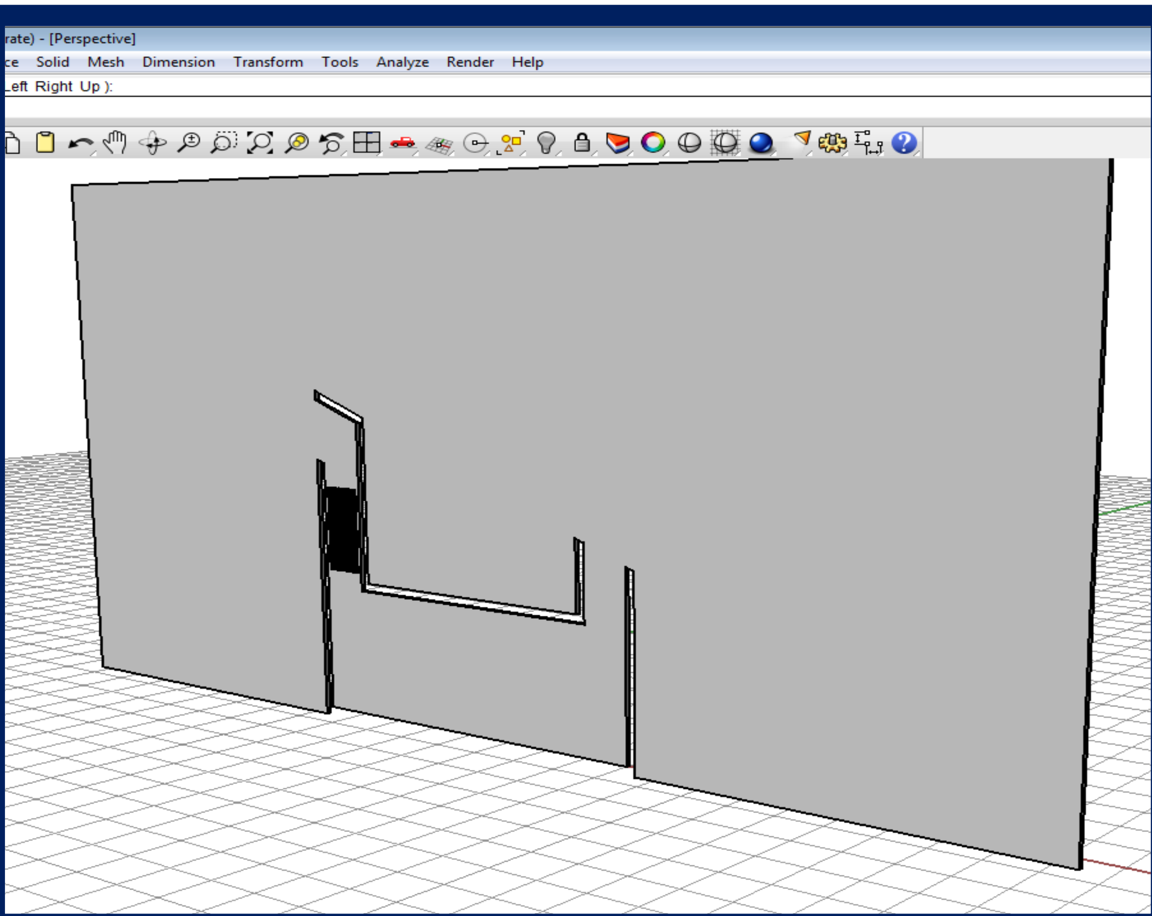


Fig. 16. Wind tower integrated with wetted columns or clay conduits.

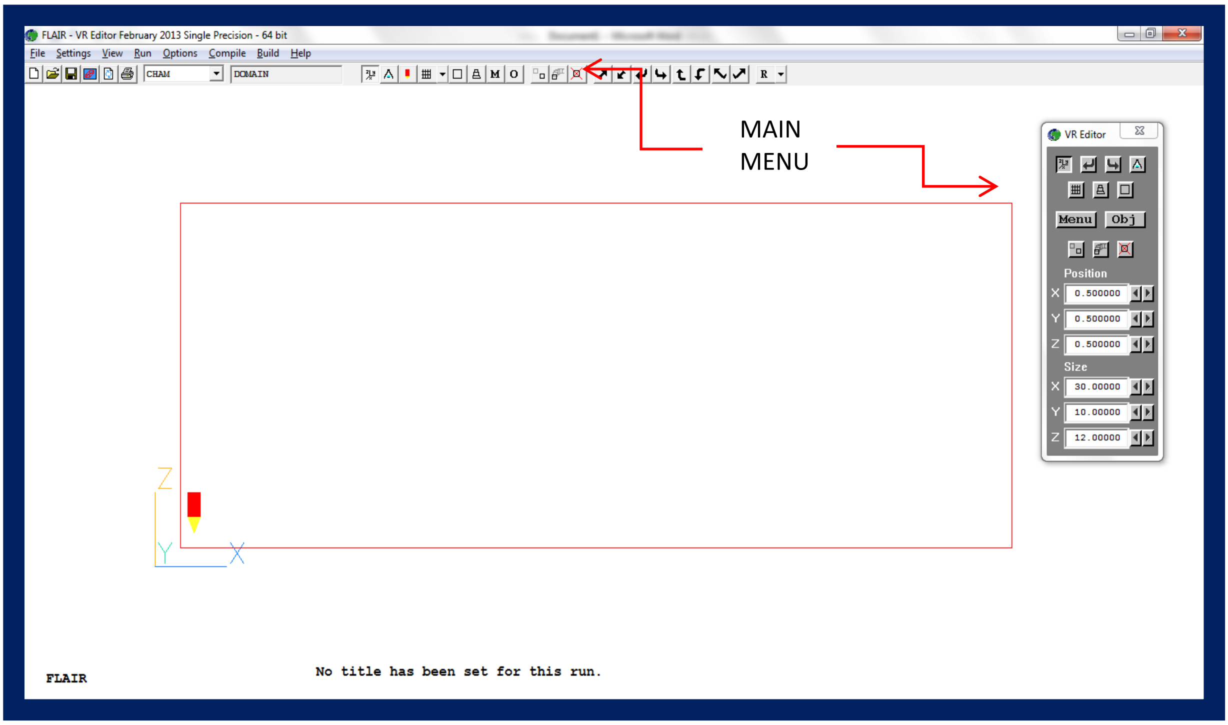


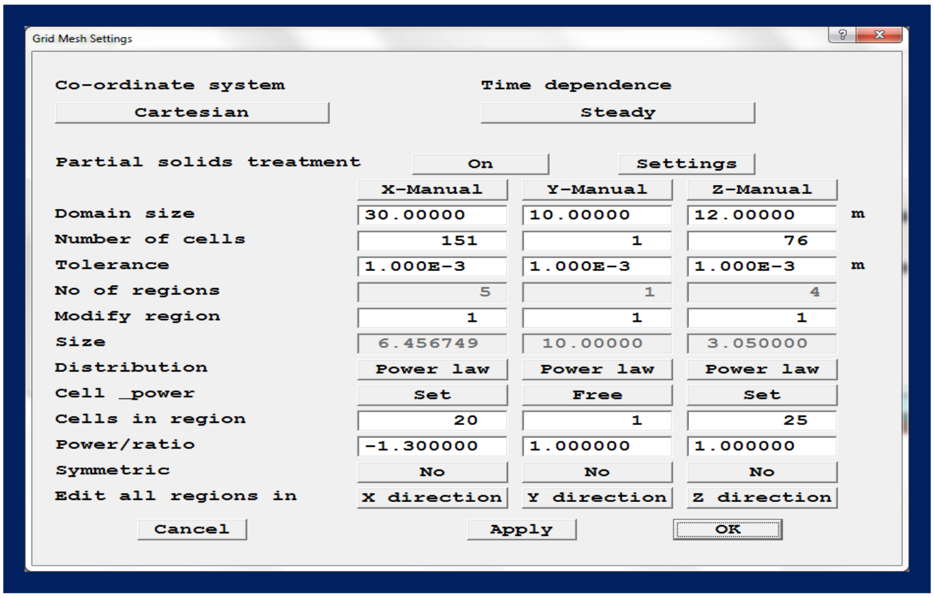
Starting a new model in PHOENICS

In the VR-Editor, go to File > New case. Select FLAIR as the interface best suited for this type of application\simulation.

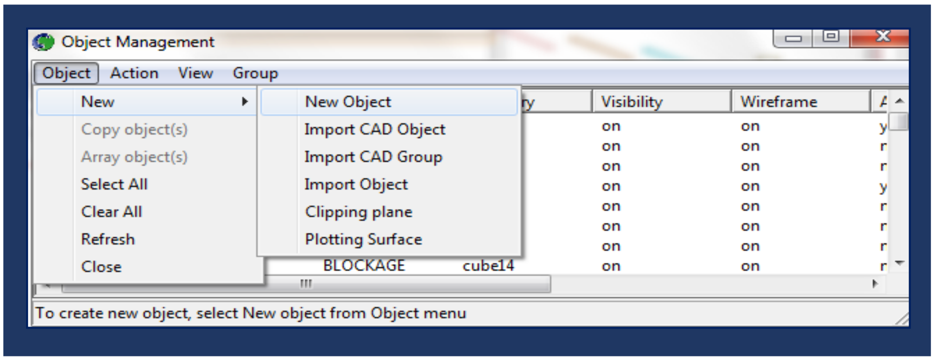
Pre-processing

Set the model domain size through the main menu > geometry. This can be done as indicated below.

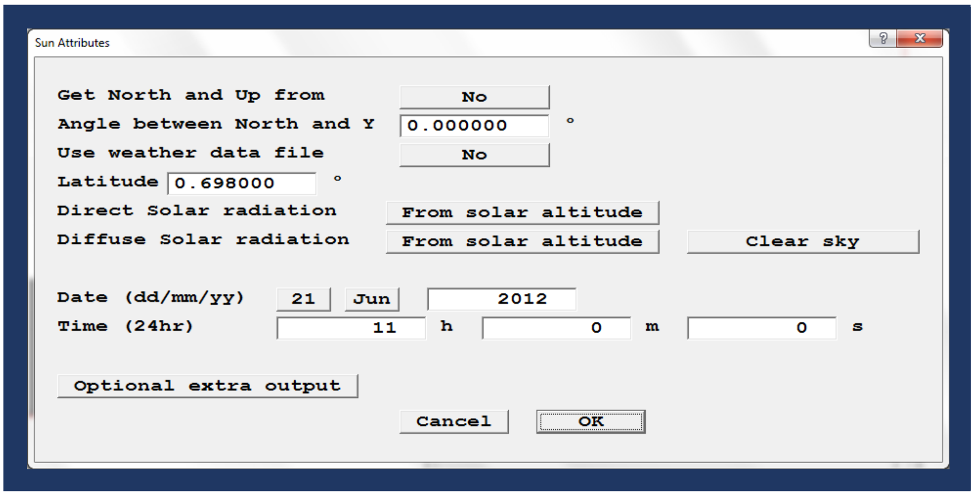




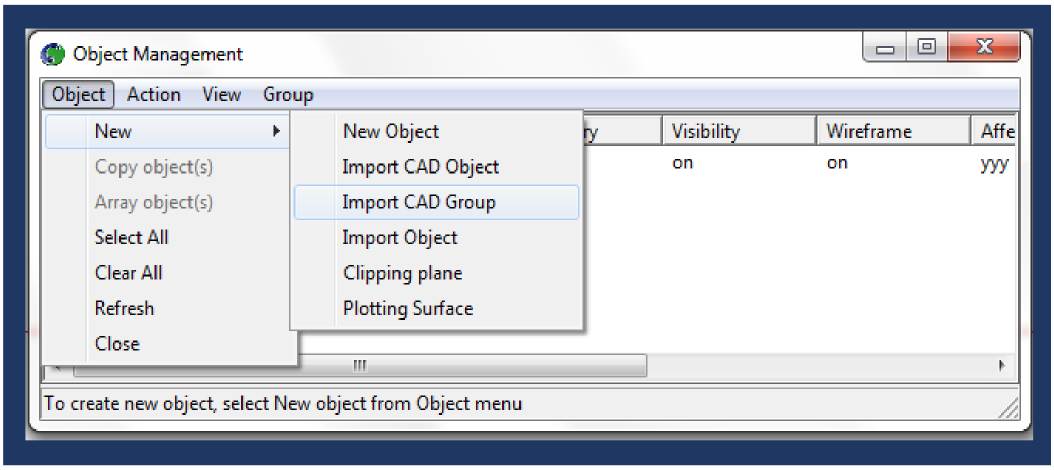
Next, all relevant objects are added to the domain. Clicking *Obj* on the control panel selects the following.



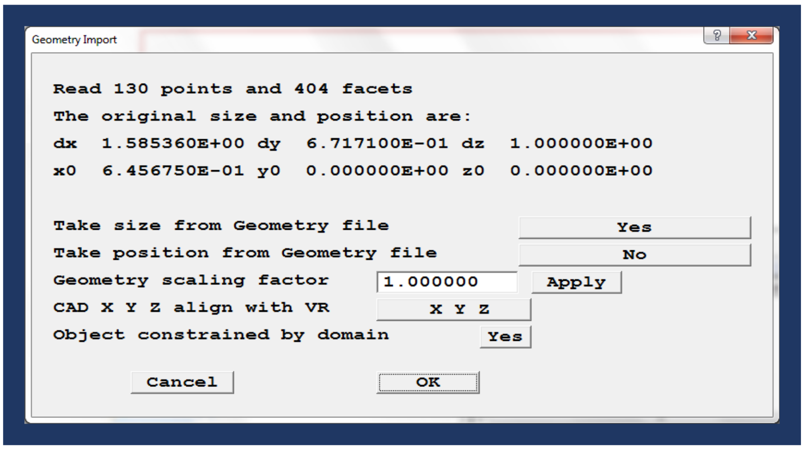
Select *New Object* and set object type to *Sun* object. Then, by clicking attributes, the following panel appears, for which relevant data can be specified.



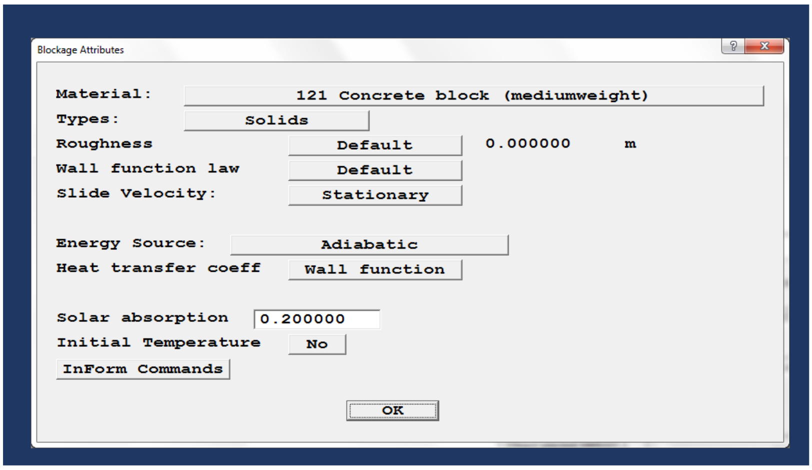
Once this is done, again go to *Obj* > *New,* select *Import CAD Object* from the list and select the desired CAD file.



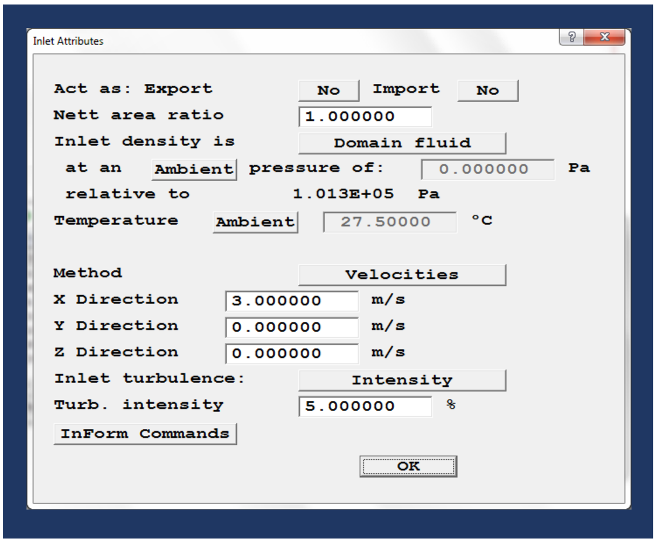
Once it is selected, the following screen will appear. We have the option to scale the CAD object in question, which can be useful if units used for the CAD file are not in metres (PHOENICS default is to use SI units.)



Once imported, clicking on attributes displays the following screen: The desired material can be selected, along with its roughness and its solar absorption factor.

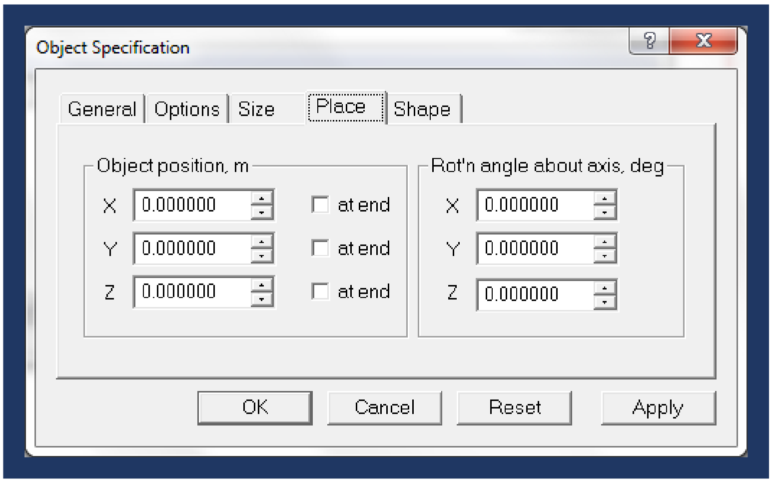


From the same *Obj* menu we can then add the remaining objects. First we will add an inlet. Select *New object* and from the drop-down list and select *Inlet*.

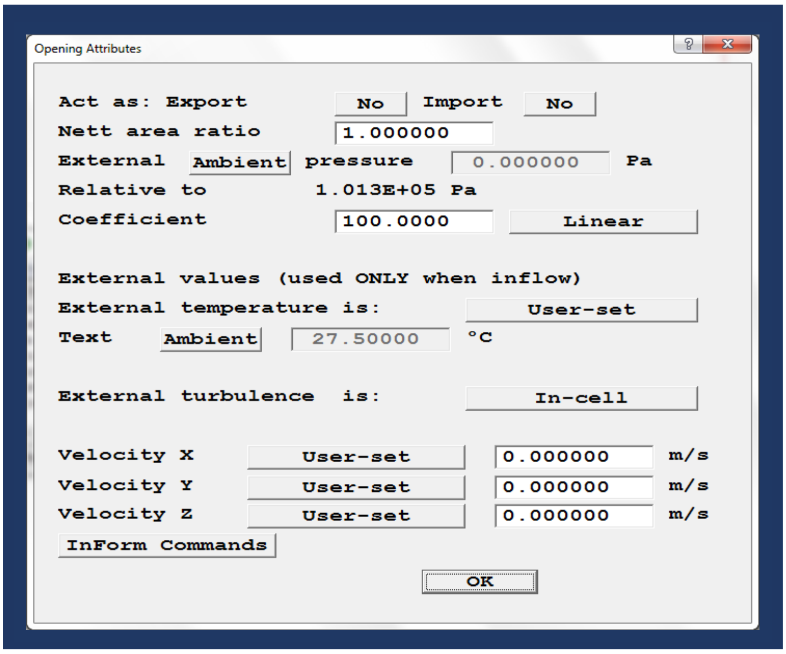


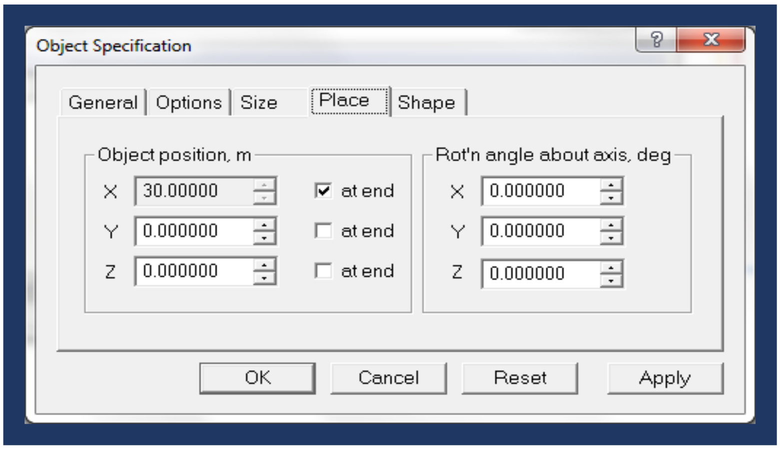
Clicking attributes we can modify the inlet conditions.

Size and place are then modified and set to fill the entire left-side of the domain (using "to end" option), with an object position of (0, 0, 0).

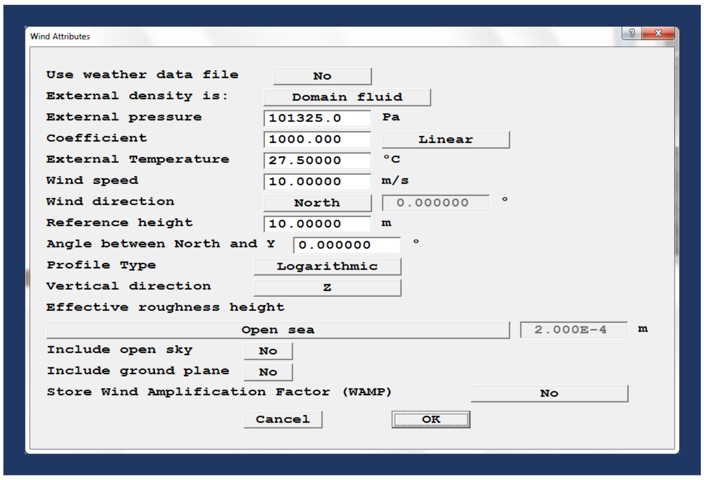


Likewise, an *Opening* (or outlet) object must be added on the far right of the domain. This is a pressure boundary, with flow through it governed by the difference in pressure inside the domain and the external reference pressure (normally set to ambient pressure). The position of this outlet is set to X "at end" or at 30m.



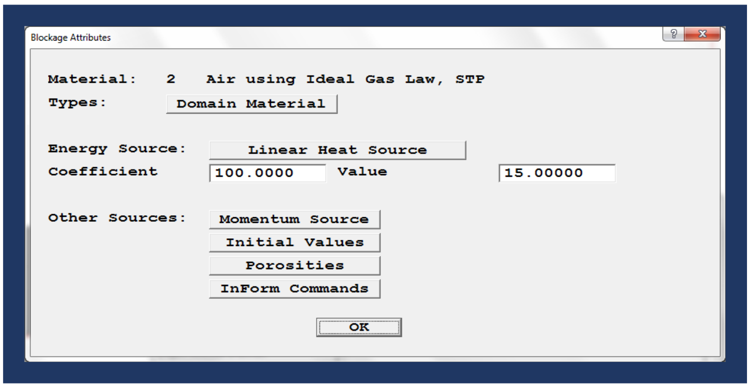


[Note that if this were a 3D simulation, it would be easier to use the *Wind* object to define the external boundary conditions through a single user panel.]

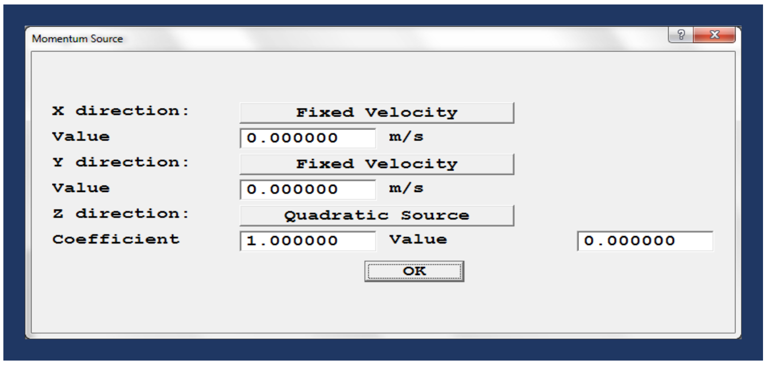


To model the cooling effect of the chimney due to a water spray, a blockage of *Domain* material type is added. This will overwrite any solid regions which happen to overlap, and in this case implies the thin vertical plates within the CAD geometry. Clicking *Attributes* we can change the material of the object to *Domain Material*, which in this case is air.

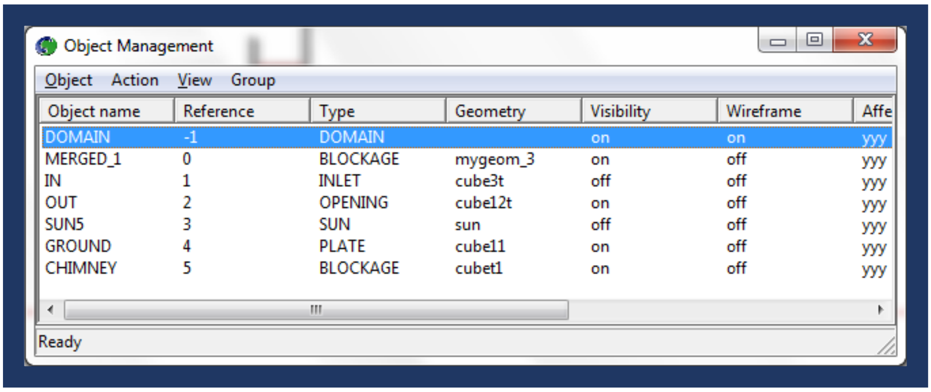
A *Linear Heat Source* source is added with a value (or temperature) set to 15ºC, and a coefficient, which establishes the extent that this temperature is to be enforced, ie a large value of 1.0+E07 would ensure air passing through the patch is fixed at 15º, while a lower value such as 100 will allow more variation, dependent on the coefficient multiplied by the difference between the specified value and local temperature of the air.



In addition to this a momentum source is set to restrict flow to the Z direction only and apply a small sink using a *Quadratic Source* to provide a pressure loss based on velocity squared via an approprate *coefficient*.

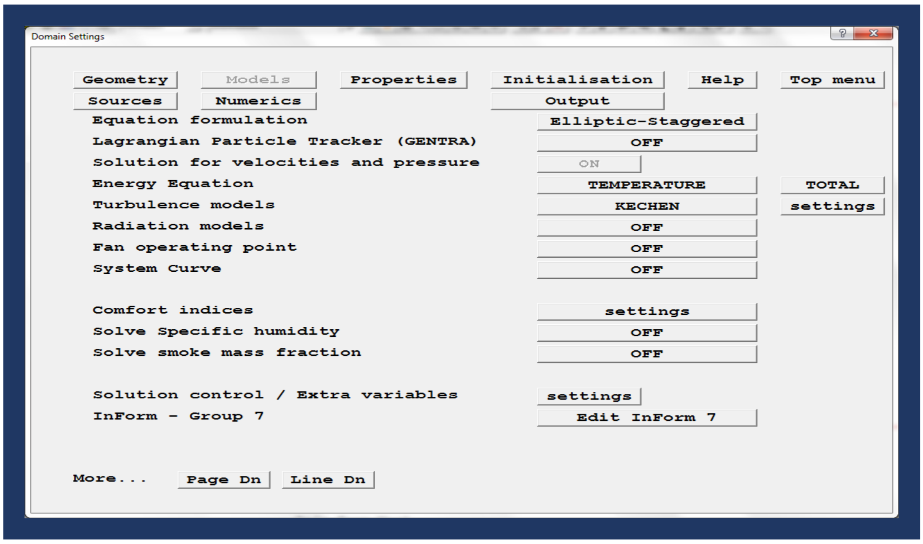


One more object is necessary for this model, and that is the Ground terrain. It is added in the same fashion as the objects above, selecting *Plate* as the type of object, sizing it to cover the entire XY plane and located at (0,0,0). This object will provide the no-slip condition for the ground, and boundary layer development for a smooth surface.

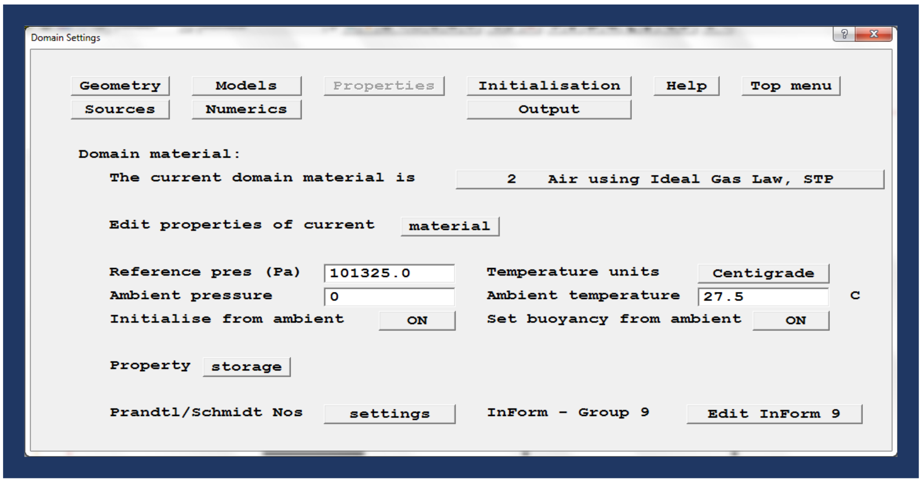


With the geomety set up, the domain settings are applied.

Clicking the *Menu* button on the control panel, and navigating to *Models* allows the turbulence model to be selected, and ensures the energy equation is solved (*FLAIR* default settings are fine).

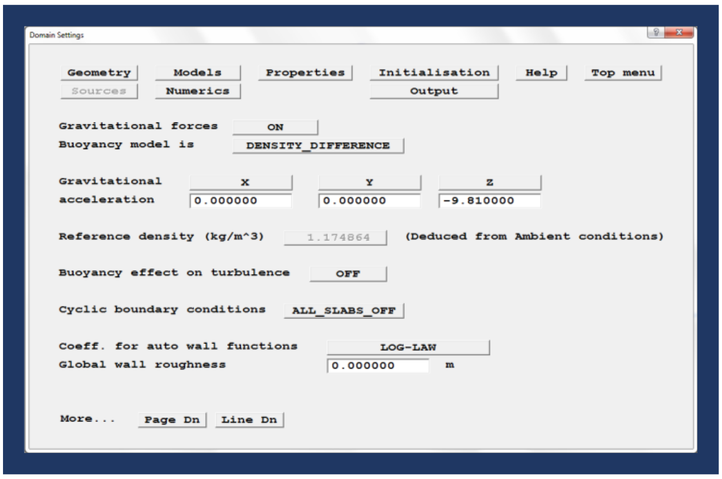


The domain material is set to *Air using Ideal Gas Law*, which allows for temperature and density variations and ambient conditions to be set.

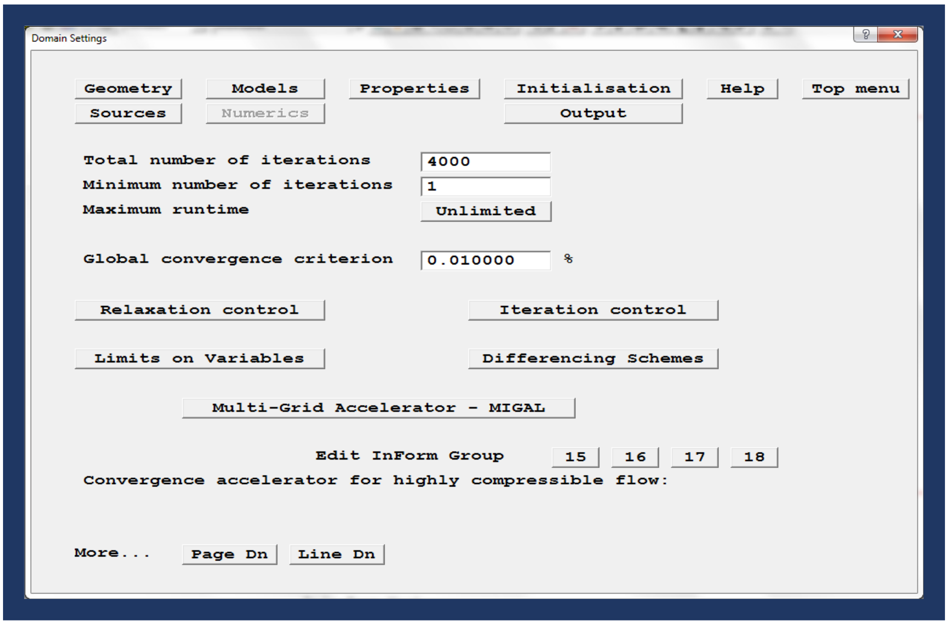


Buoyancy is set in the *Sources* panel. This option is set automatically to the *Density Difference* method if *FLAIR* is used.

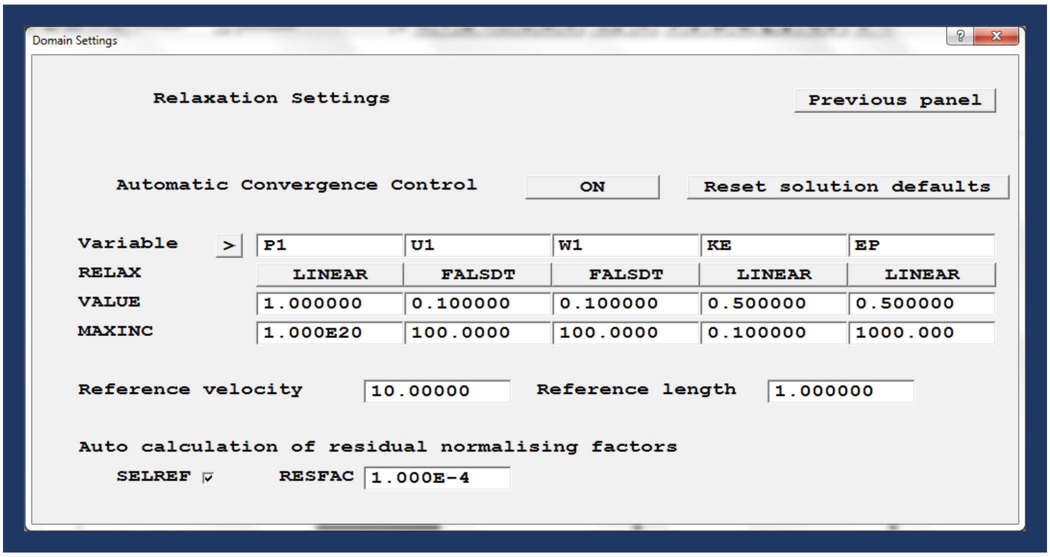
The option to have buoyancy affect turbulence can also be set in this panel (eg for fire/smoke modelling).



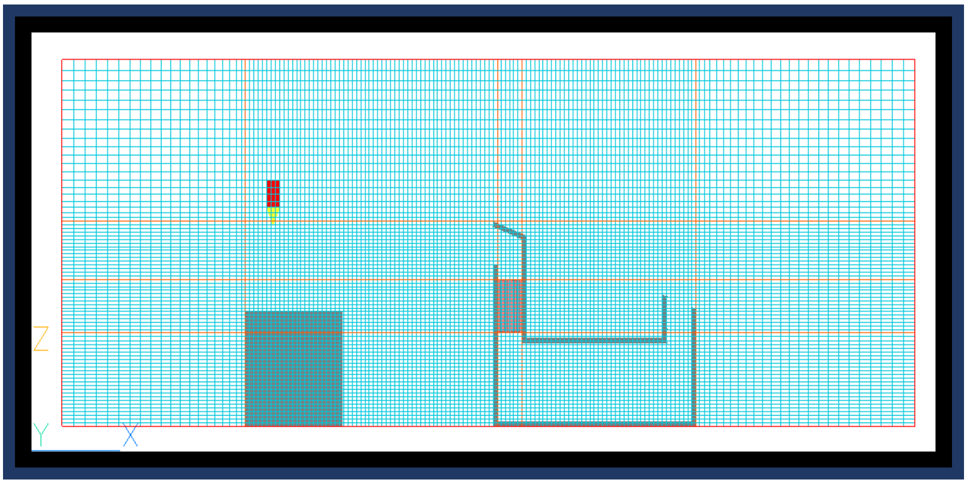
Solver conditions are set under *Numerics*. These include the total number of iterations to be performed, the type of differencing scheme to be used and the settings of any relaxation parameters.



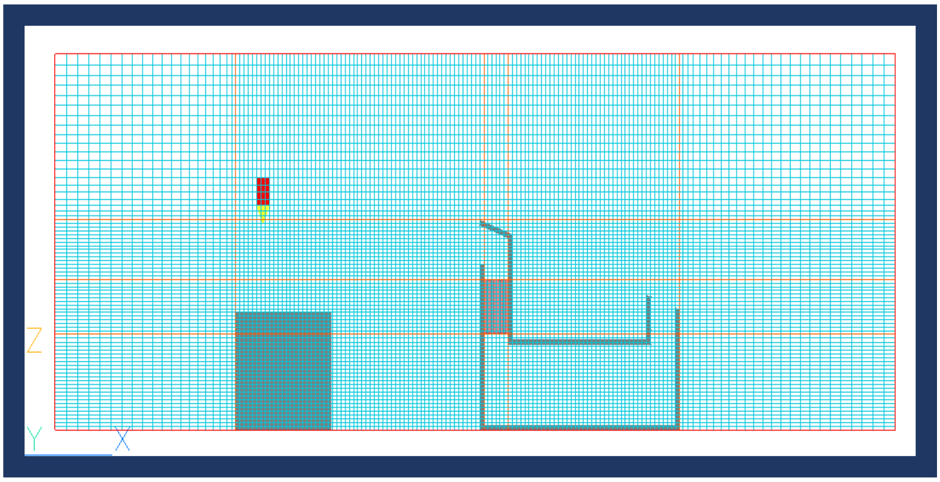
The default option for relaxation is *Automatic Convergence Control* (CONWIZ): On. This is a convergence wizard that automatically sets the relaxation parameters to try to ensure convergence. If it is switched off, experienced users can set parameters manually, which can lead to faster run times using less conservative relaxation values.



PHOENICS creates automatically a Cartesian mesh across the whole domain. It uses a cut-cell algorithm called *PARSOL* (PARtial SOLids) to analyse which parts of a given cell are occupied by fluid and solid regions. This reduces greatly the complexity of generating a mesh for any given case.

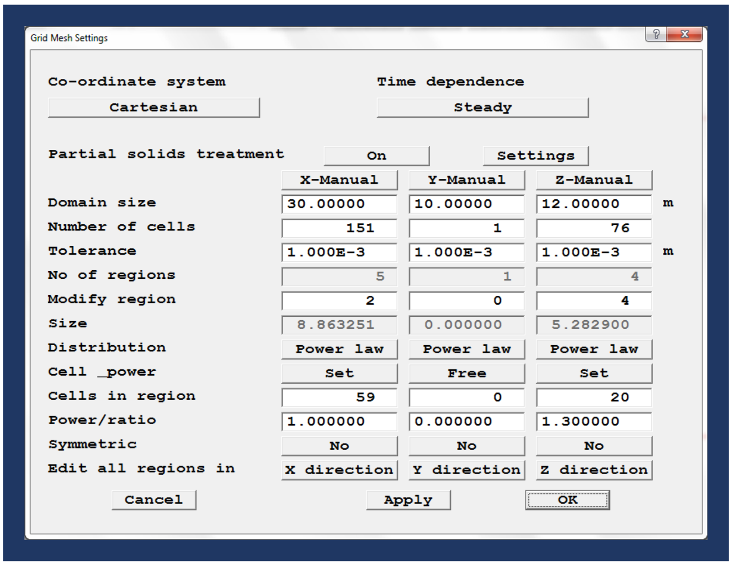


Clicking the mesh button on the control panel displays the mesh overlaid on the domain. Each of the regions (created by objects affecting the grid and denoted by orange lines on the mesh display) can be modified individually by clicking them and changing the desired parameter on the *Grid Mesh Settings* panel.

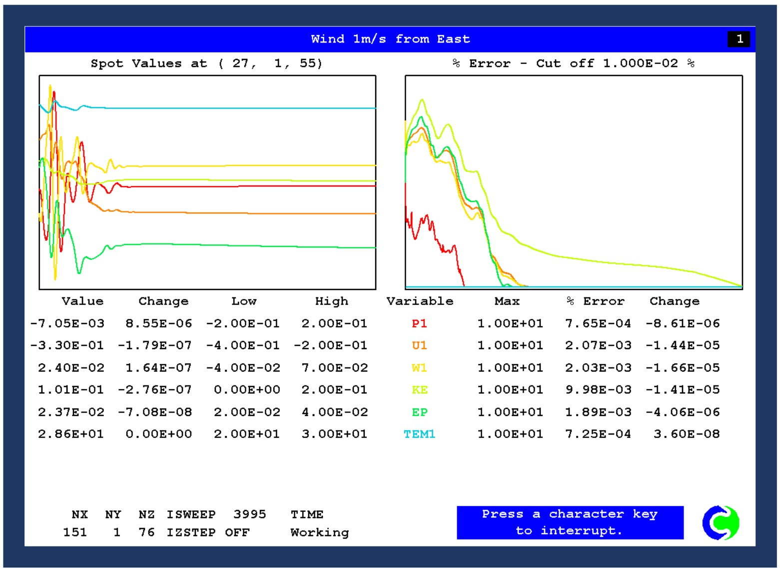


In certain regions it might be beneficial to crush the mesh using non-uniform spacing to reduce the overall cell count. Both *Power* and *Geometric Progression* distributions are available, with power/ratio settings <-1 for diminishing and >1 for increasing mesh spacing.

Once the desired mesh is obtained, the solver can be run. To commence the run, select *Solver* from the *Run* menu tab.

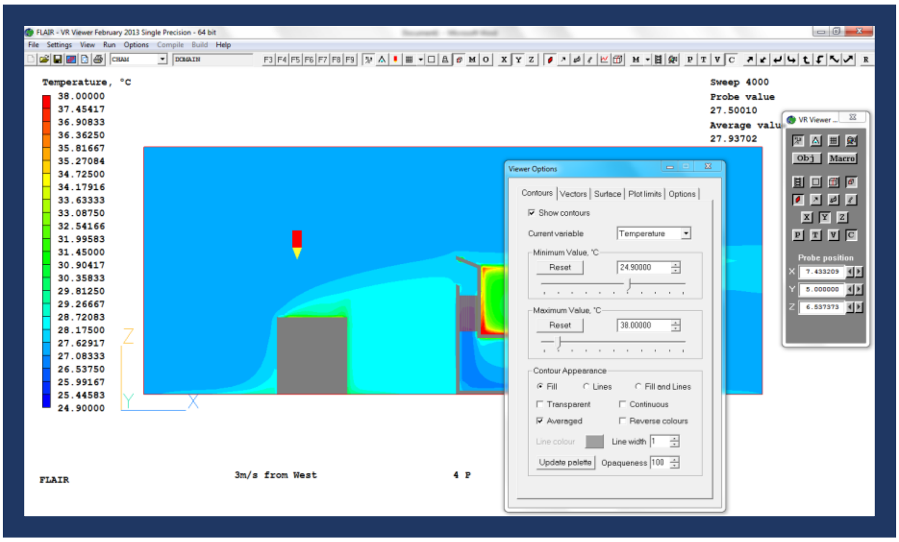


A real-time plot of residuals and spot values for the probe location are printed to the screen as the run progresses (known as a GXMONI plot). This establishes whether the solution is converging or not. As we can see on the right hand side of the plot below, the residuals tend to be very small numbers, while the spot values on the left remain stable as the number of iterations increases - an indication of good convergence.



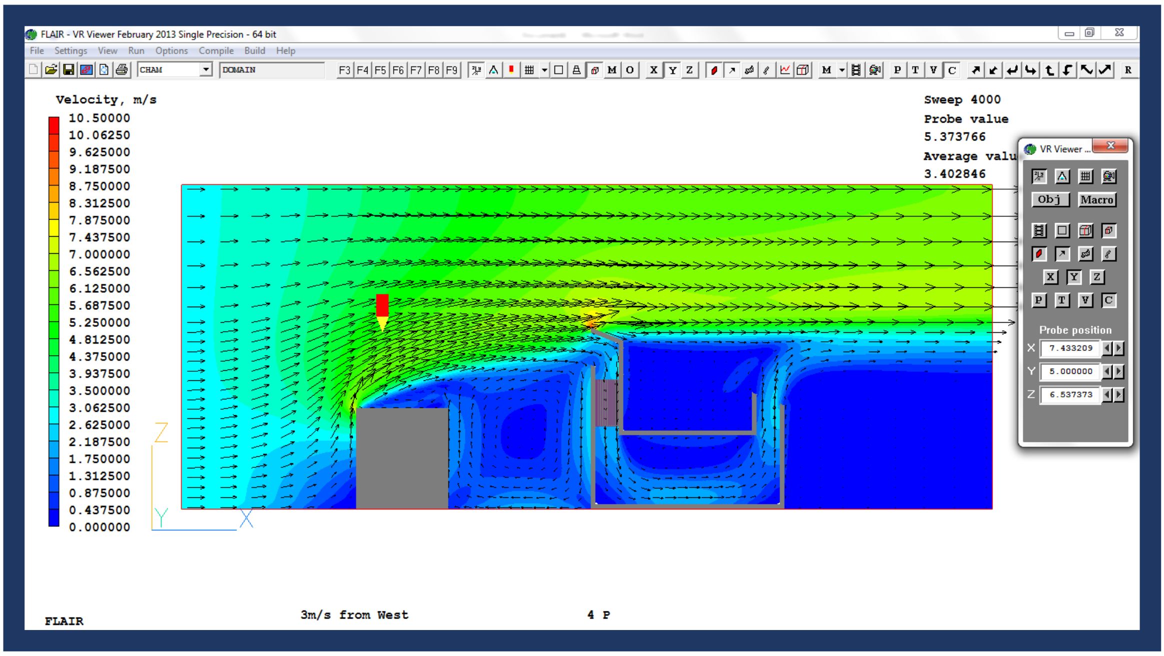
Post-processing

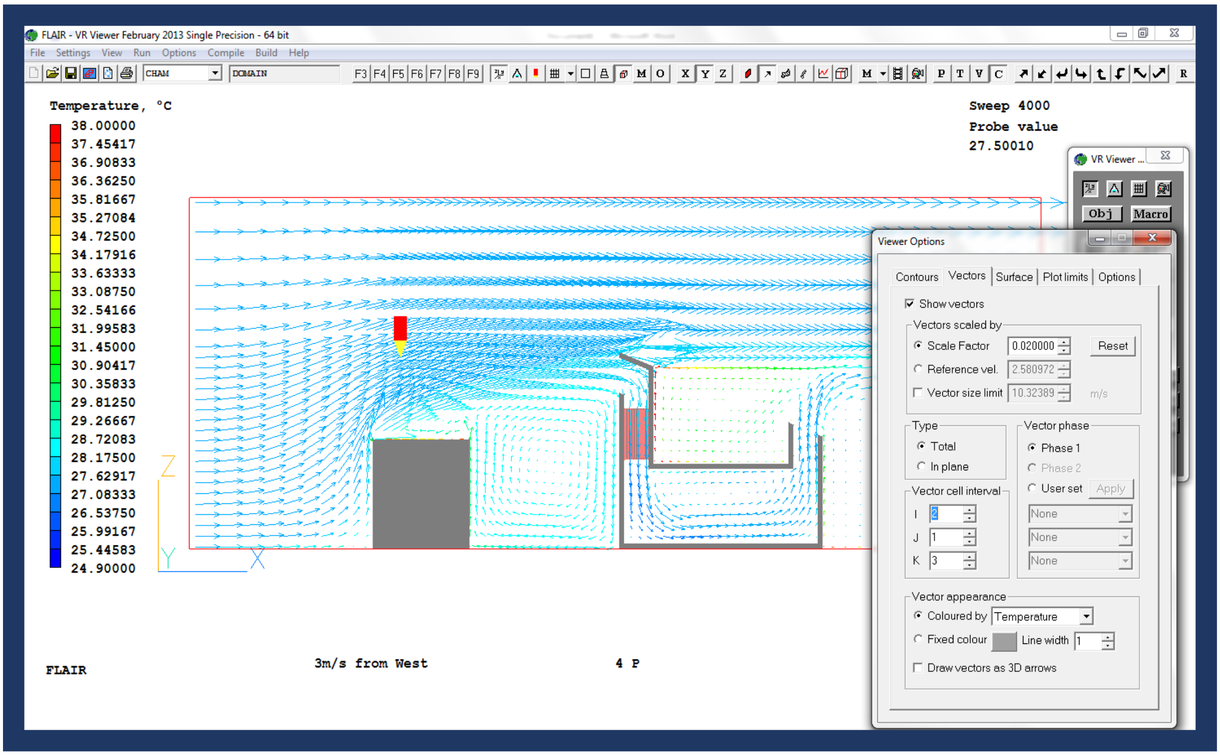
Once the run has completed with satisfactory convergence, select *GUI*, from the *Post-Processor* tab (also under the *Run* menu tab). Clicking the *C* button on the control panel opens the contour plot options.



Vectors can be used in conjunction with, or instead of contour plots by selecting them from the *Vector* tab.

Use of the Sun object provides the option to plot certain related field variables such as #Sol (Solar absorption factor), #QS2 (Total heat source per unit cell) or LIT (Illumination flag).





These can be accessed from the *Current Variable* drop-down menu.

Full post-processing guidance can be found by clicking on the PHOENICS-VR Reference Guide: http://www.cham.co.uk/phoenics/d_polis/d_docs/tr326/tr326top.htm