



RhinoCFD Tutorials

External Flow Tutorial



RhinoCFD Official document produced by CHAM
October 13, 2017

1 Introduction

This tutorial focuses on external flow around city structures. It is designed to get you up to speed on some of the capabilities of RhinoCFD's FLAIR extension. The .3dm file "external flow" can be downloaded from [RhinoCFD Tutorials](#) for use in this case.

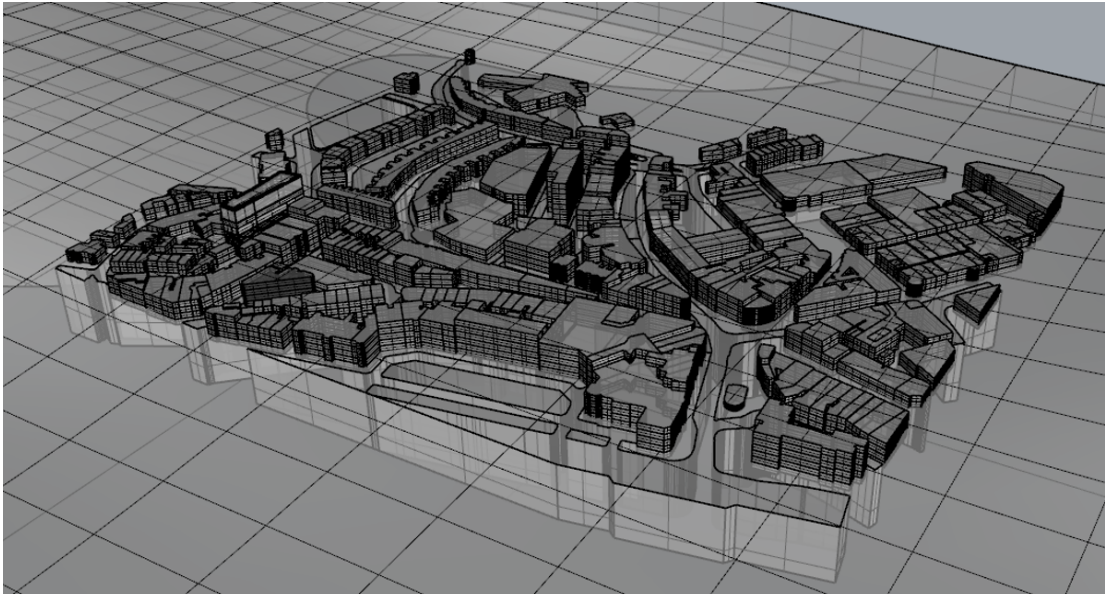


Figure 1: External Flow Geometry

The external flow geometry contains a host of buildings and terrain. In order to keep simulation time down, we will concentrate on the buildings.

CFD Analysis

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

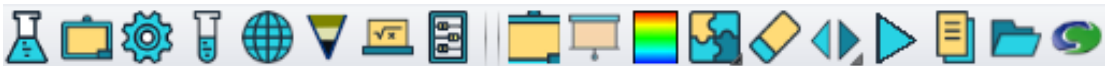


Figure 2: RhinoCFD Toolbar

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

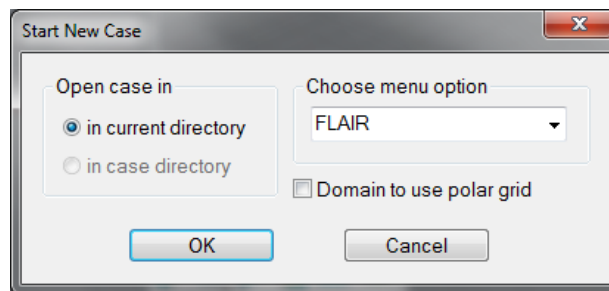


Figure 3: Menu Option: FLAIR

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 covers the buildings completely and extends downstream. Ensure that the minimum Z location is exactly at the bottom of the geometry. Your domain should look similar to that shown in the following figure:

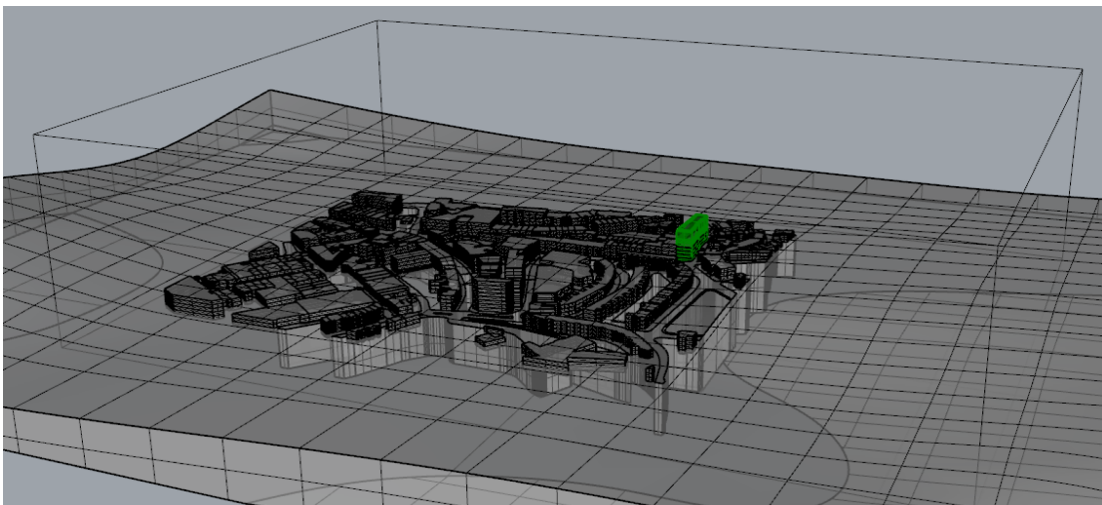


Figure 4: Domain size and placement

The WIND Object

In this tutorial we will use a 'WIND' Object which has been specifically designed for this application. The function of the WIND object is to provide boundary conditions representing the atmospheric boundary layer due to the wind at all faces of the domain. This means that you can easily prescribe a wind speed (at a given height) and a wind direction and the software will calculate the wind profile for you.

The velocity profile to be used can be:

- a logarithmic profile

- a power-law profile
- a profile read from a data file (see appendix for more information)

Creating the WIND object

The wind object can be easily created by right clicking on the second icon on the RhinoCFD toolbar.

The domain faces menu will appear. The last option on this menu is for the wind object, click 'no' to change the button to 'yes' to activate it.

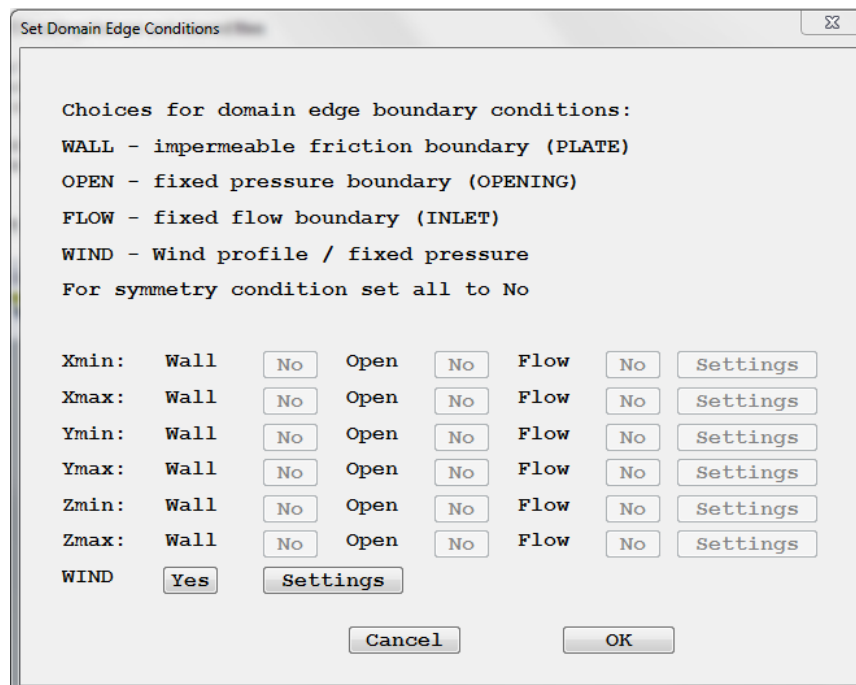


Figure 5: Domain Faces Dialog

Next click on 'settings' to bring up the following menu.

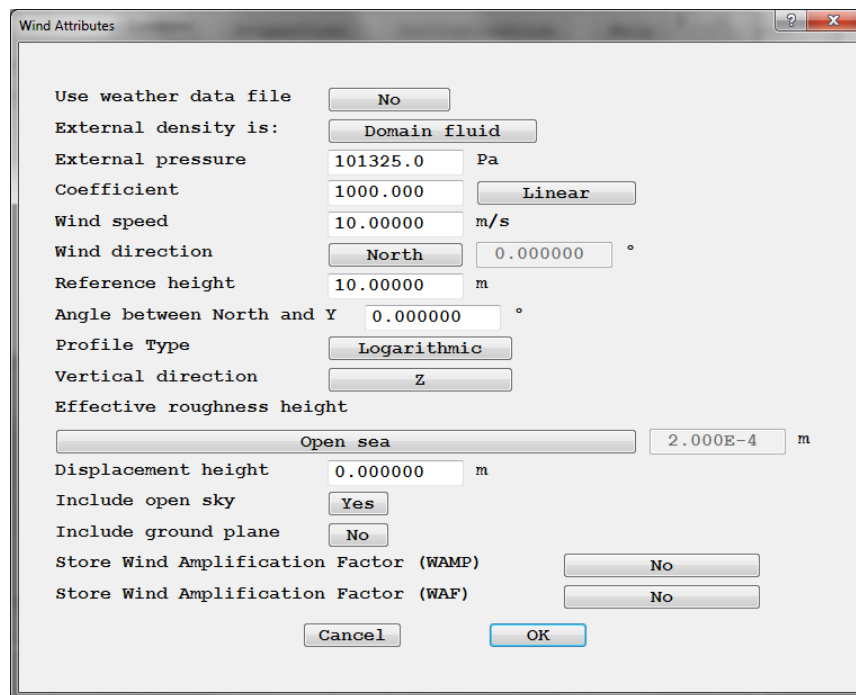


Figure 6: Wind Settings Dialog

Change the settings of the following options:

Option	Setting
Wind Direction	E-S-E
Effective roughness height	Open flat terrain
Open sky	Yes
Store Wind amplification	No

NOTE: You can click on the "?" in the top right corner of the window and then on any of the buttons to bring up extra help and information on that specific button.

Click OK and OK again to exit the domain faces dialog and the Wind object will appear.

Meshing

The next step is to ensure the mesh is adequate to resolve the flow near the building of interest. To modify the mesh, click on the fourth button on the toolbar.



This will bring up the mesh modification panel. The automeshing should be switched on for each of the three directions. The mesh can be modified manually by clicking on "X/Y/Z Direction".

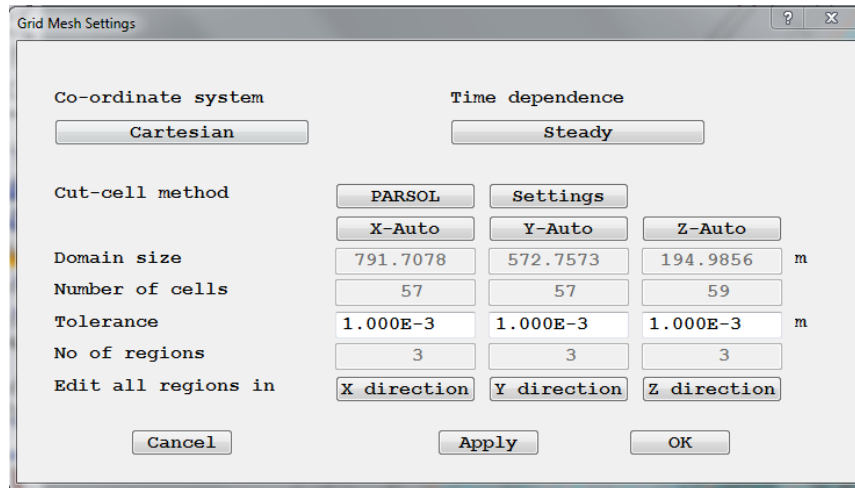


Figure 7: Main Meshing Dialog

Alternatively, the mesh 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.

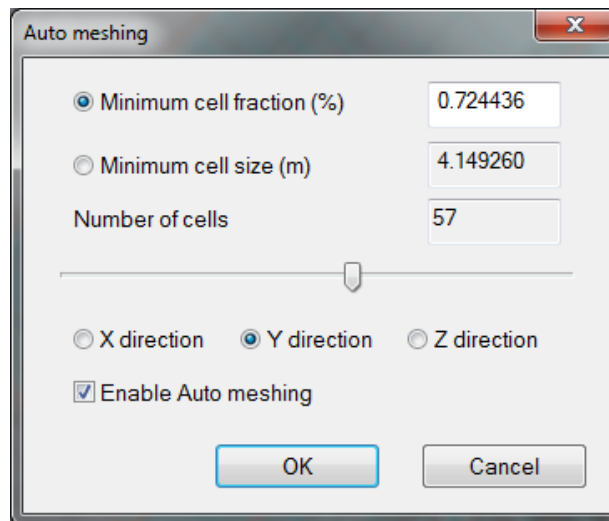


Figure 8: Automesh Panel

To ensure this simulation runs quickly, use the slider or the mesh menu to get approximately 60 cells in each direction. (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").

Next inspect the mesh around the green building by viewing the geometry from the "top" and then the "front". There should be roughly 50 cells (10 by 5) covering the green building in each of these views.

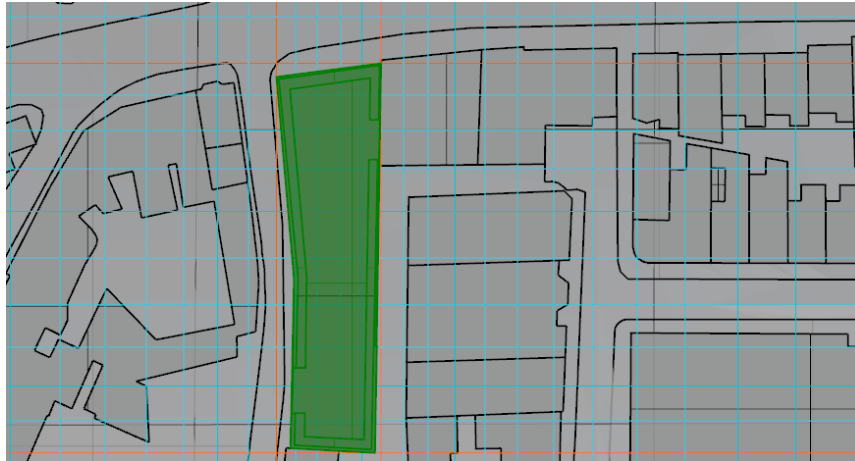


Figure 9: Approximate Mesh over the Green Building (Top View)

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 0.01 for the momentum components (U1, V1 and W1). These values are based on the average cell size and the average windflow velocity. A general rule of thumb is to set the manual relaxation of momentum to the nearest order of magnitude to $\frac{Cellsize}{FlowVelocity}$

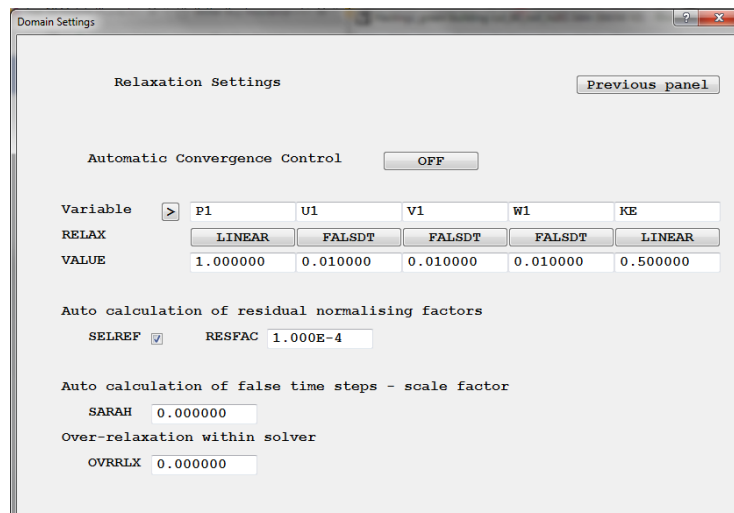


Figure 10: Relaxation Settings

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

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



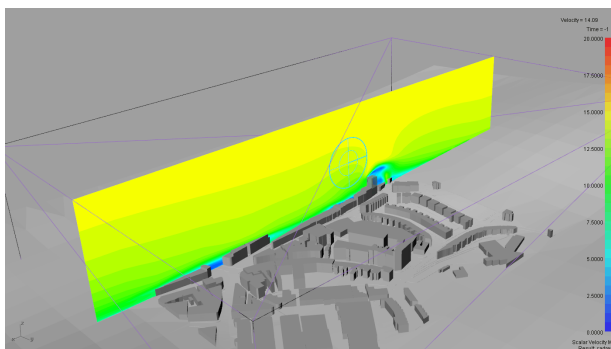
Visualizing Results

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).

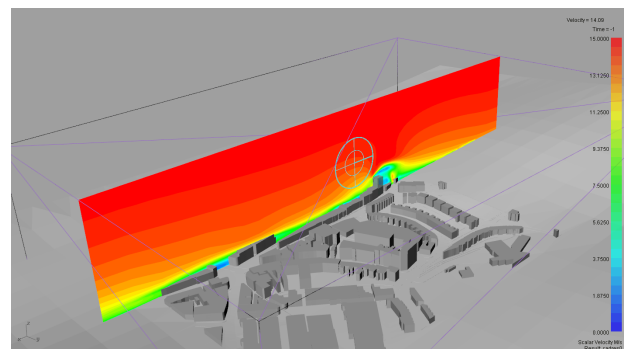


For more details on results visualizations see [Flow Past a Cylinder Tutorial](#) or the [RhinoCFD User Manual](#).

In this case using a cutplane and rotating it by 90 degrees we can see the flow rising over the top of the tallest building, leaving behind it an area of slow recirculating flow. Changing the range of the variable in the viewer panel can alter the amount of detail we get from a particular cutplane.



(a) Velocity contours



(b) Velocity contours

Figure 11: Velocity contours across main building with different scales

We can also look at the pressure contours on the building itself by selecting the surface-contour probe from the toolbar and then selecting our object of interest. The values of pressure (or any other plotted velocity) can then be exported to a .csv file by clicking on the export button on the viewer options panel.

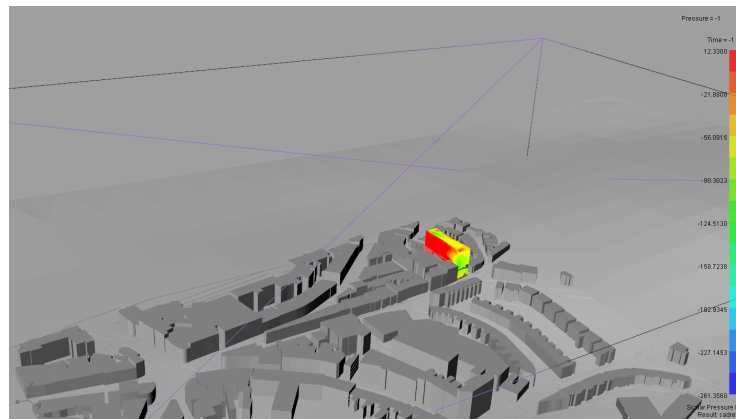


Figure 12: Pressure contours on the building of interest

Further work

Try investigating the results when the grid in the X and Y axis is increased to $100 \rightarrow 150$. This should show how the grid more thoroughly captures the complex geometry and the flow field behind the building changes accordingly.

Appendix - Using Wind data files

Data files can be used to set wind profiles across an inlet of a wind object to match specific conditions from experimental data. This can be done in two ways:

- Using a weather data file
- Using user defined velocity and turbulence files

Weather data files

Weather data files use data supplied by [EnergyPlus](#) (an open source weather data website) which allows users to select weather data from over 2100 locations world wide, and contain data for every day in a year.

The weather data files can be downloaded and accessed by clicking on the top button next to ‘Use weather data file’ in the wind attributes menu, then ‘configure file’ and the following dialog will appear:

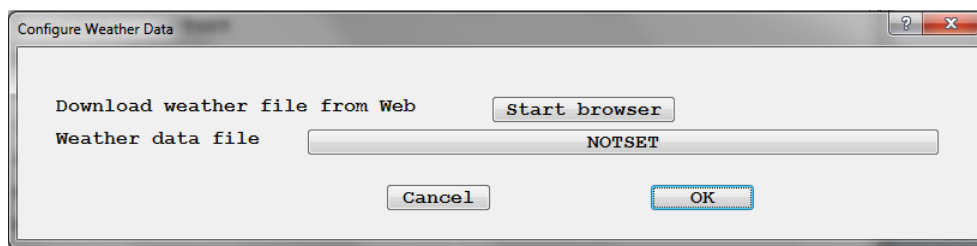


Figure 13: Weather data file menu

Clicking on ‘Start browser’ will launch the EnergyPlus website in a new browser tab. Download any .epw data file and save it somewhere easy to find. Then using the above dialog select the data file and click ‘load data file’. You can now choose month, day and time, click OK to confirm the settings.

User set files

User set data files should contain two columns: distance (usually z axis) in meters and velocity in m/s or turbulent kinetic energy in J/kg (m^2/s^2). The data must start with the lowest height and end with the highest. The usual format for these files is comma delimited (.csv).

These files can be used by first clicking on ‘logarithmic’ next to Profile type to bring up a Profile menu, select ‘Table file’ and click OK. Two new options will appear in the menu: Velocity profile file and Turbulent kinetic energy file. Clicking on ‘NOTSET’ next to these options launches a file select dialog which is used to select the data file. Ensure the data files are placed in the current working directory.



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