RhinoCFD

Powered by PHOENICS

RhinoCFD User Guide - TR400

CHAM Ref: CHAM TR/400- RhinoCFD User Document

Document release date: April 2020

Software version: PHOENICS 2019 v1.1

Responsible contributors: A Carmichael, S Ahmed, R Dyer, D Glynn & G Michel

Published by: Concentration Heat and Momentum Limited (CHAM)

Confidentiality: Free Access

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

1 Introduction

RhinoCFD is a general purpose Computational Fluid Dynamics (CFD) software plugin, built directly into the Rhino environment. It allows users to investigate how a fluid, gaseous or liquid, flows in or around the geometry being modelled. The fluid may convect or conduct heat, and may carry species such as pollutants. RhinoCFD permits rapid optimization and testing without leaving the familiarity of Rhino.

RhinoCFD is powered by PHOENICS, the first commercially available, general-purpose CFD code, which has a proven track record simulating scenarios involving fluid flow, heat and mass transfer, combustion and/or chemical reaction for a wide range of industrial and environmental applications. PHOENICS distinguishes itself from other CFD software through its ease of use and inclusion of innovative features designed to help the user achieve the best simulation possible.

Section 2 of this document covers how to install RhinoCFD; Section 3 describes how to start the software and Section 4 describes the RhinoCFD interface, with specific reference to the toolbar. At this point the user is ready to start setting up a model. An overview of how this is done is given in Section 5, with further detail being given in subsequent sections.

This document is intended to allow users rapidly to gain familiarity with RhinoCFD. More comprehensive reference documents are provided as needed; many are available from the PHOENICS Online Information System (POLIS). NOTE: general PHOENICS documentation, such as that within POLIS, may reference the "VR Interface". This is the main graphical user interface to PHOENICS and is not used by RhinoCFD, which is based on the Rhino environment.

2 Installation of the RhinoCFD Plugin

2.1 Standard Installation

Users will be supplied with a RhinoCFD installer file ("RhinoCFD.rhi") for use with the 64 bit version of Rhinoceros 5 and Rhinoceros 6. Double-click the RhinoCFD icon to install the plugin. By default McNeel sets plug-ins to installs to the directory %AppData%\McNeel\Rhinoceros\5.0\Plug-ins on your computer.

To uninstall RhinoCFD follow the procedure detailed at: Uninstalling Rhino 5 RHI Plug-ins.

RhinoCFD no longer requires two installers (RhinoSatexe and VTKInRhino); these are now packaged into one.

2.2 Parallel Installation

Users who have purchased a licence allowing them to run RhinoCFD on multiple processors will need to have MS-MPI installed. RhinoCFD will work with version 6 of Microsoft MPI or later. Please note that parallel RhinoCFD is not yet set up for cluster computing (currently this enables users to run simulations on up to 16 processors on a single computer). To install MS-MPI double-click on:

MSMPIsetup.exe: installs additional features required to run the parallel enabled version of RhinoCFD.

MS-MPI will not need to be installed if MS-MPI version 6 or later is already installed on your computer.

2.3 Tool Bar

On the first activation of RhinoCFD, the tool bar will appear on the screen.

Figure 1: RhinoCFD Undocked Toolbar

This can be dragged to the top of the screen to become a tab. It will then be presented as:

Figure 2: RhinoCFD Docked Toolbar

2.4 Replacing Legacy Toolbars

If the toolbar does not appear, then a clean install is recommended. To complete a clean install go to the folder: AppData/Roaming/McNeel/Rhinoceros/5.0/Plug-ins, and, if present, delete the VTKInRhino, RhinoSatexe and/or RhinoCFD folders. Then complete the install again. The next time you restart Rhino the tool bar should appear on your screen.

3 Starting RhinoCFD

3.1 How to Start RhinoCFD

To start RhinoCFD go to the RhinoCFD Toolbar and click on the first icon  any time after Rhino has been launched. A file location dialog will appear where the working directory is to be selected. The working directory is a folder where all files relating to your geometry and CFD simulation will be saved. Next an options dialog will appear where the user should select one of the available versions of RhinoCFD.

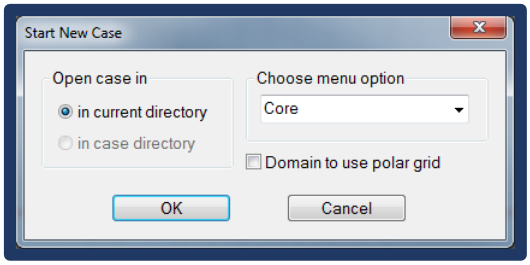

Figure 3: Start new case dialog

There are currently three versions of RhinoCFD:

Core: The general multi-purpose PHOENICS CFD solver, designed for a wide variety of uses.

FLAIR: FLAIR is a specialised version of PHOENICS used by architects and building services engineers. FLAIR provides designers with a powerful and easy-to-use tool for the prediction of airflow patterns, temperature distributions, and smoke movement in buildings and other enclosed spaces, as well as wind flows around buildings.

MARINE: This version is designed to help naval architects test hull designs with greater ease.

This guide is concerned solely with “Core”.

On exiting this dialog, a box representing the “domain” should appear on the screen and fit around all objects created in Rhino (see Figure 4). The domain will be the region of space in which the CFD solution is performed; its extent may be adjusted by the user as will be described later. RhinoCFD is now ready to use. Further objects can be added later if required e.g. to add flow and thermal boundary conditions.

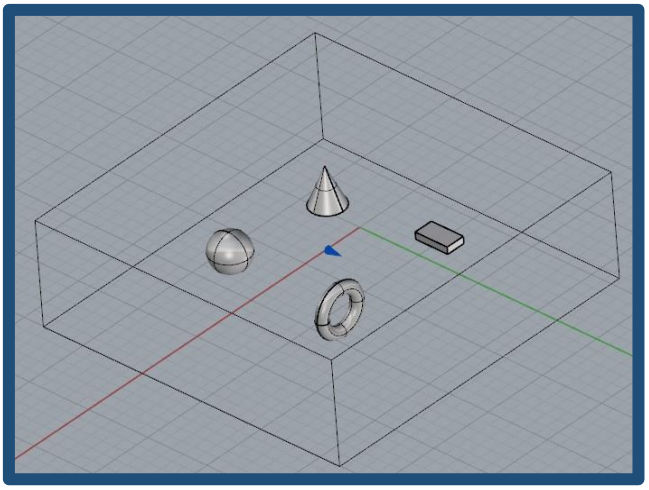

Figure 4: Domain

3.2 Deactivating RhinoCFD

To stop using RhinoCFD, right click on the first toolbar icon  . The domain will disappear along with the all CFD attributes applied to Rhino objects. However, it is generally recommended to leave the plugin data saved – otherwise all the CFD settings will be deleted.

3.3 Geometry Requirement

3.3.1 Correct 3D CFD Geometry

RhinoCFD is designed for use with ‘Closed elements’ such as polysurfaces or meshes. Some open elements will be recognized and work correctly provided holes in them are small in comparison with the CFD mesh. As this is not guaranteed, this element type should be avoided wherever possible.

3.3.2 2D Geometry

RhinoCFD does not recognize surfaces which are 2D objects. If these are present in your model, flow will pass right through them. Real solids have thickness, and so modelled solids must have thickness also. Note that for a solid object to be properly detected, its thickness must be similar to or greater than the mesh size in that direction.

4 RhinoCFD Interface

4.1 The RhinoCFD Toolbar

Figure 5: RhinoCFD Docked Toolbar

The icons in the toolbar have been placed in such a way as to make RhinoCFD easier to use:

1. The first eight buttons are separated from the remainder, as they are concerned with setting up and running simulations.
2. Buttons 9 to 15 are to visualize the results

3. Button 16 displays some of the internal files used by the PHOENICS CFD Engine (mostly used for checking the solution properties are correct).
4. The penultimate button is for saving all case files.
5. The last button is an 'About' button which displays minimal information about the creators. Table 1 gives a description of all buttons.

Icon	Left Button	Right Button	Comments
	Name	Name	
	Create Domain To Fit Object	Deactivate RhinoCFD	LC- Starts RhinoCFD, creates a domain, RC- Removes domain, CFD properties
	Edit Solution Parameters	Edit Domain Edge Properties	LC – Opens main CFD setting window RC – Edits domain boundary conditions
	Edit CFD Properties	Show Table Of Objects	LC - Edits currently selected object RC – Show list of all objects
	Show Grid Dialog		
	Show Grid	Hide Grid	
	Show Probe	Hide Probe	
	Add User Defined Functions		
	Run Solver	Display Convergence Plot	
	Load Results	Remove Visualisation	
	Edit Display Parameters		
	Show Scalar Key	Hide Scalar Key	
	Add Probes From Dropdown	Produces drop down (see next row for details)	
	 Add Cut-plane		
	 Add Isosurface Probe		

Drop downs	Add Stream Probe		
	Add Point Probe		
	Add surface contour		
	Add line plotter		
	Hide Current Object (selection)	Reveal Hidden probes	
Icon	Left Button	Right Button	Comments
	Next time step	Previous time step	
Drop downs	Back 10 steps	Back to first step	
	Play time series	Pause time series	
	Forward 10 steps	Forward to final step	

*Table 1: Tool bar Icon Descriptions. *LC-left click, RC = right click*

4.2 Main Menu

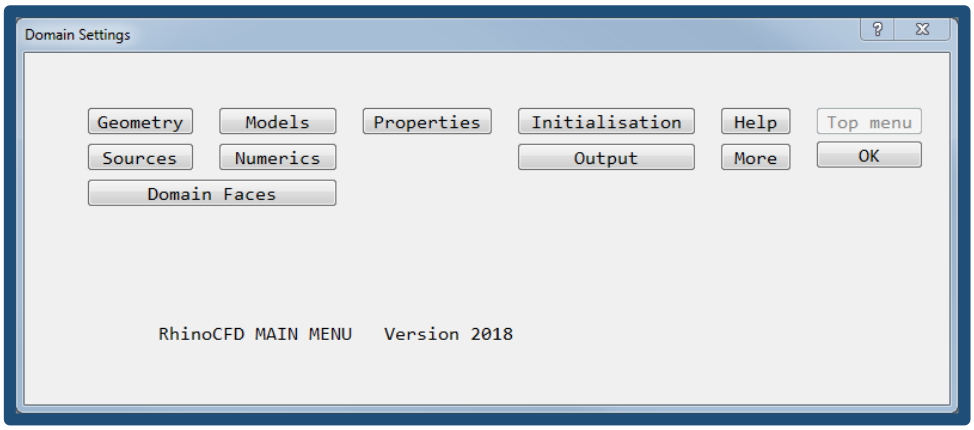

Figure 6: RhinoCFD Main Menu Window

Button	Function	Settings of interest
Geometry	Settings for grid, coordinate system and time discretization	
Models	Specify physical models used for simulation	Turbulence, energy, radiation, etc.

Properties	Specify the fluid properties that will be modelled in your simulation	Density, viscosity, ...
Initialisation	Specify the initial values applied to your simulations and activate restarts	
Help	Launch help	
Sources	Whole domain sources	Gravity, buoyancy and moving/rotating coordinate systems
Numerics	Numerical settings for solution control	Number of iterations and relaxation
Output	Result file sources printout and 3D solution dumping controls	Probe location, print out of variables, file dumps and extra derived variables
'?'	Launches a help window within the menu. Activate through clicking on '?' then on the any button	

Table 2: Main Menu Description

5 Setting Up a CFD Model

5.1 Introduction

Having discussed how to install and start RhinoCFD, it is now time to set up a CFD model. This section discusses the principal aspects of setting up a model:

5.2 Geometry

The geometry of the model is set up in Rhino, and therefore appears automatically in the RhinoCFD plugin, as shown in this example:

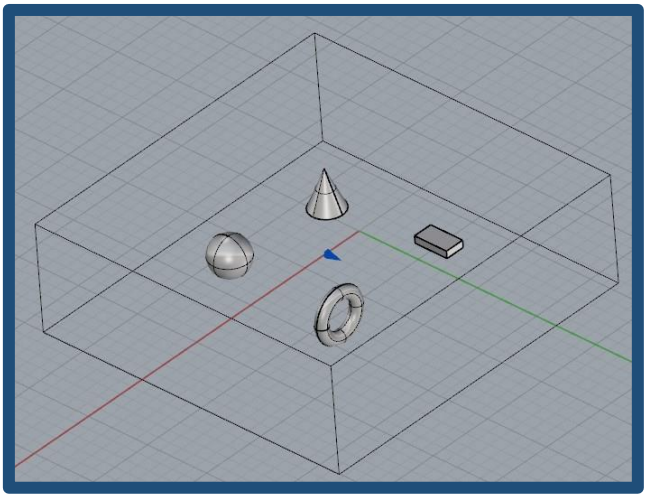

Figure 7: Solution domain around Rhino generated object

5.3 Domain

The region of space in which the solution is to be performed is termed the "Solution Domain". It is a rectangular Cartesian region as shown for example by the black lines in Figure 7. A default domain is defined when exiting the Start New Case dialog, as described in Section 3.1. The domain automatically will be created large enough

to encompass all the geometry set up in Rhino. It can be resized easily if a larger domain is required, e.g. for an external simulation of wind flow around buildings.

5.4 Mesh

A full discussion of both automatic and manual meshing is given in Section 6.

5.5 Boundary Conditions (Flow and Thermal)

The flow and the thermal conditions that characterise the model must be specified. For example the following boundary conditions need to be quantified, and their locations specified:

1. Inflows or outflows of fluid (see Section 9.2).
2. Fixed-pressure boundaries (see Section 9.3).
3. Fixed-temperature boundaries (see Section 8).
4. Heat sources (see Section 8).
5. Pollution sources.

Boundary conditions and sources are set up using CFD objects; see Section 7 for further information.

5.6 Solution Parameters

The above aspects of setting up a CFD model are all visual, in the sense that they are all concerned with geometry. Other, non-visual, parameters are also required to specify the model, e.g. what equations are to be solved, what is the fluid, what are the solids, what are the solution controls, etc. These aspects are all set in the "Main Menu" of RhinoCFD. For a full description of the Menu the reader is referred to the PHOENICS VR User Reference Guide. A list of the most important parameters which need to be considered when setting up a model is given in Section 5.7 below.

5.7 Other Important Aspects of CFD Models

Additional aspects of CFD modelling with Rhino are discussed in subsequent sections as follows:

1. Use of the Probe (Section 11)
2. Running simulations (Section 12)
3. Convergence and relaxation (Section 13)
4. Running transient models (Section 14)
5. Viewing the results (Section 15)
6. Use of the Scalar Key in contour and vector plots (Section 16)
7. Files associated with RhinoCFD (Section 17)

5.8 Steady and Transient Simulations

RhinoCFD can handle both steady and transient simulations. Steady simulations are independent of time and assume that boundary conditions are constant. Examples are a steady ventilation pattern in a building, or steady flow over an aerofoil.

In a transient simulation, the velocity pattern and temperature distribution can change with time. An example of a transient simulation is calculating the development of a fire in a building, to predict the build-up of the smoke layer, so as to determine the time available for escape. Transient simulations are computationally more expensive than steady ones.

RhinoCFD default simulation is set to be steady. Section 14 discusses how to run a transient simulation.

6 Meshing

6.1 Purpose of Mesh

The principle by which CFD works is to subdivide a domain into many small "control volumes" known as cells, and then solve the Navier-Stokes equations to obtain values for pressure, momentum, temperature and other

variables at each of these cells. The equations represent conservation of mass, momentum, energy etc. The more cells there are, the more accurate the representation of a scenario will be. However, the simulation will take a longer time to complete and converge, and incur greater computational expense.

6.2 Available Mesh Types

RhinoCFD uses Cartesian or cylindrical-polar structured meshes which are very easy to set up, and have lower discretisation errors than non-orthogonal, or general unstructured, meshes.

6.3 Geometry Detection

RhinoCFD uses a cut-cell method called PARSOL that detects automatically what part of a cell is solid and what part is fluid, and applies the correct boundary conditions to the relevant parts of the cell. This means that you don't need to spend hours defining the mesh so that it perfectly fits the geometry; just ensure you have a fine mesh and let RhinoCFD do the hard work.

6.4 Displaying the Grid

Left clicking on the 5th tool bar icon  reveals the grid within the current view. Rotate the perspective view, or click in a different view, to change the grid view. The grid location is determined by the 'probe' (Section 11); move the probe to move the shown grid around the domain.

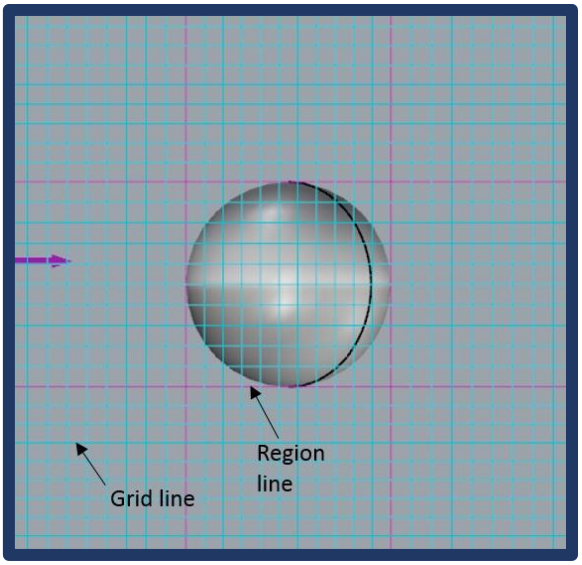

Figure 8: Grid Lines

6.5 Grid Regions

The standard Cartesian mesh is formed of regular hexahedral cells stacked in rows and columns, and is represented in RhinoCFD by means of blue lines which indicate cell faces.

In each of the x-, y- and z-directions the mesh is divided into a number of 'regions' in order to provide detailed grid control. The size and distribution of the cells within each region can be modified via the grid dialog menu. The grid lines representing region boundaries are indicated by orange lines (purple if an object affecting the grid is currently selected); so, for example, in Figure 8 above it can be seen that there are three regions in each of the vertical and horizontal coordinates.

Regions can be created by setting particular objects to 'affect the grid' (see Section 7.3), in which case region boundaries will be inserted at the extremities of the object (Purple lines in Figure 8). Alternatively, regions can be created by defining a Null object (see Section 7.4), which affects the gird, but does not affect the simulation. The minimum size of each region can be set by modifying the tolerance in the grid dialog.

6.6 Manual and Automatic Meshing

In each coordinate direction (x,y,z) the mesh may be constructed automatically, or with manual intervention if preferred. This choice may be made separately for each of x, y and z. The automatic meshing process ensures that the resulting mesh is "well-conditioned" in the sense that there are no abrupt changes in cell width; on the other hand, the cells may not be distributed optimally for the specific model. Manual meshing allows the user to distribute cells more efficiently. This can be a time-consuming process, but one that pays dividends in terms of the quality of the results. The default meshing method is automatic. This will therefore be described first.

6.7 Automatic Grid Generation

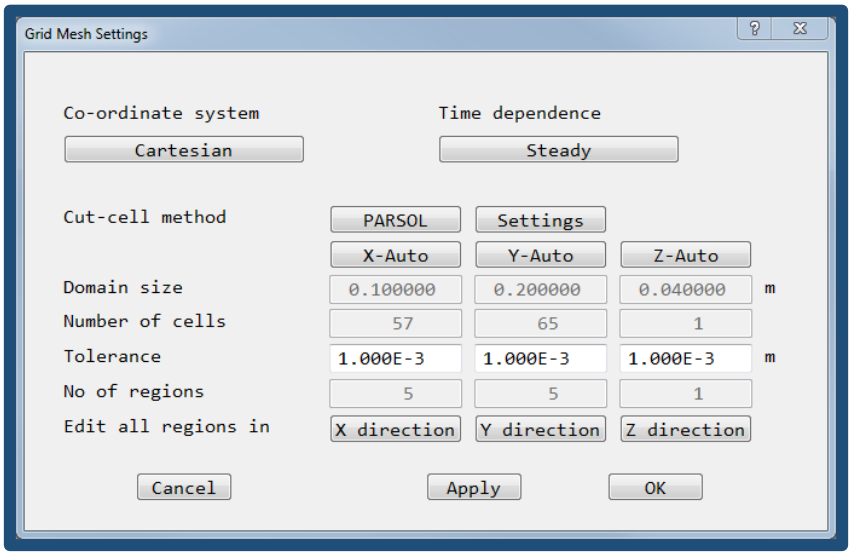

Figure 9: Grid Mesh Settings Dialog - Automatic

RhinoCFD uses an automatic mesher as the default option. The auto-mesher will always produce a well-conditioned grid for the geometry being analysed, though the user should check that the mesh distribution has adequate resolution. Clicking on X, Y or Z direction buttons brings up the settings dialog for that direction, with the options explained following Figure 10:

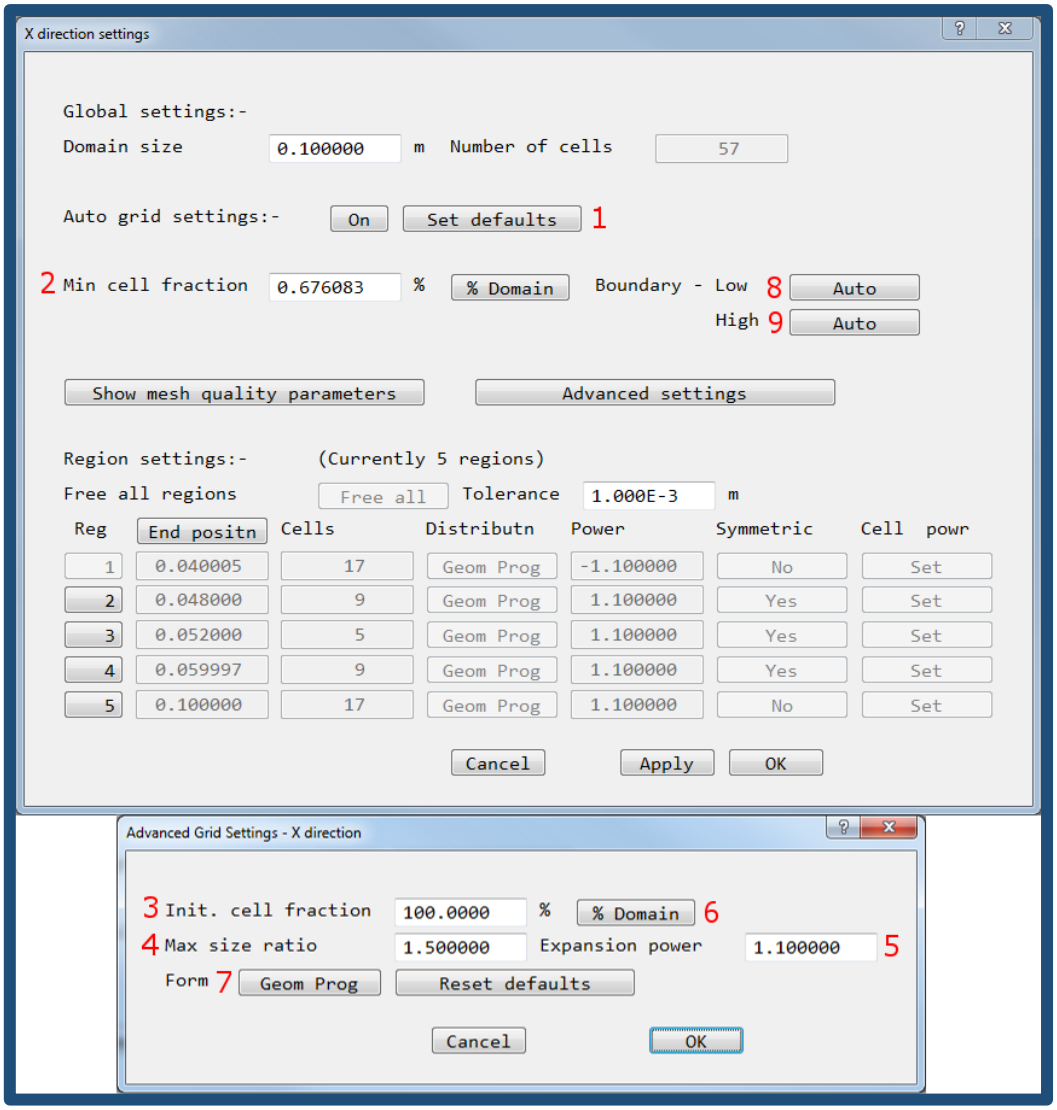

Figure 10: Auto-Mesh Dialog

1. Resets all values to their defaults.
2. Sets the smallest cell size allowed as a fraction of domain size in that direction. Default is 0.005.
3. The automatic mesher uses an iteration method to generate the mesh. This sets initial cell size for the iteration as a fraction of the domain size in that direction. Default is 0.05, which gives roughly 20 cells if the regions are fairly equal in size.
4. Sets maximum size ratio for adjoining cells in neighbouring regions; e.g. with a ratio of 1.5 the last cell in region 1 cannot be more than 50% wider than the first in region 2 (or vice versa).
5. Sets the expansion/power (see (7)) used to adjust the grid within a region to satisfy cell-size ratio requirements at region ends. Default value is 1.2. A value of 1 will enforce a uniform grid in the region. Increasing the value increases the degree of cell crushing towards one end of the region.
6. By default, the minimum and initial cell factors (2 and 3) are set as fractions of the domain size. When 'Fractions' is clicked, the setting changes to 'Size'. The minimum and initial cell sizes are now set as physical sizes in metres.
7. The grid can expand using either geometrical progression, or a power-law expansion. Geometrical progression gives a uniform expansion in grid size across the region, whereas using a power law concentrates the expansion towards one end of the region.
8. This allows the user to control whether a one-sided expansion towards the boundary, or a symmetrical expansion, is applied in the first region in the relevant coordinate direction. The former corresponds to OFF (default).

9. Similar to (8), but for the last region in the relevant coordinate direction.

Tips for using the Automatic mesher:

1. The initial cell factor is used as starting point for the number of cells. If you want a large number of cells, then a use a small fraction or size.
2. The minimum size of the cell is a limit rather than a determining factor for the grid. This can be used to ensure that the grid is fine enough at the walls of an object, or to ensure that all cells are the same size by making it equal to initial cell factor.
3. Using the fraction option may be helpful to set up a reasonable grid quickly when the domain is much larger in one coordinate than in another. Using sizes options can be easier for refining complex regions as input can be directly related to geometry dimensions.
4. The Max size ratio should not be greater than 2; ideally it should not exceed 1.5.
5. Low (8) and High (9) options should be used only if necessary to resolve flow details near domain boundaries. They should be left "Off" for external flows such as wind flow around buildings.
6. Users should examine grids critically before running simulations and assess whether the meshing is adequate to resolve details of complex geometry and flow features such as recirculation regions.
7. The Auto-mesher is often useful to create, quickly, a mesh for a complex model with many objects. The mesh generated will always be well conditioned. However, the user should check whether mesh resolution is adequate.

6.8 Manual Grid Generation

Figure 11 shows the grid mesh settings dialog layout. Each feature is described briefly here. Further help can be found by clicking on '?' and then on a button. This dialog is accessed by clicking on 'Geometry' in the main menu.

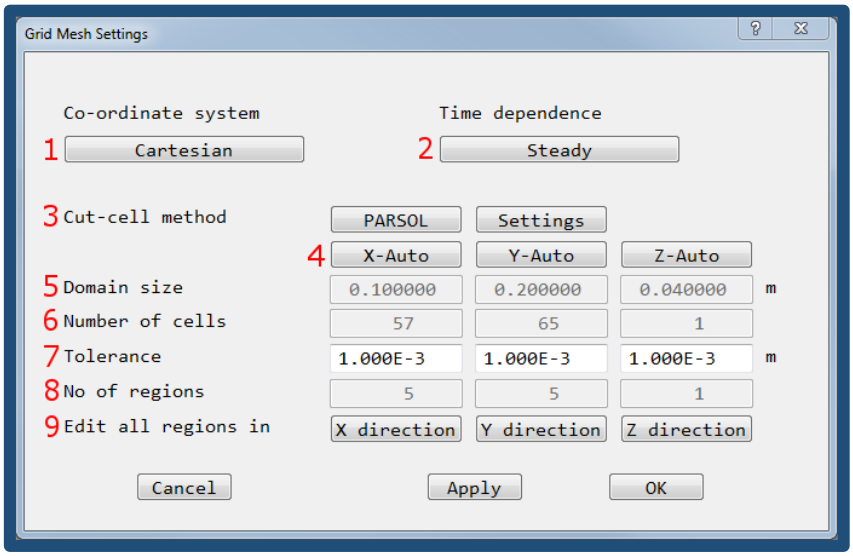

Figure 11: Grid Mesh Settings Dialog - Manual

1. Select either Cartesian or Polar coordinate system.
2. Choose either steady or transient simulation.
3. Change the cut cell method (Only PARSOL available currently).
4. Change grid specification to automatic or manual.
5. Display domain size (edit using 9).
6. Total number of cells in each coordinate.
7. Region tolerance - i.e. minimum region width.

8. Total number of regions in each coordinate.
9. Button to edit size, cells, regions and distribution in each coordinate.

Clicking on "X direction", "Y direction" or "Z direction" (9) brings up the following menu.

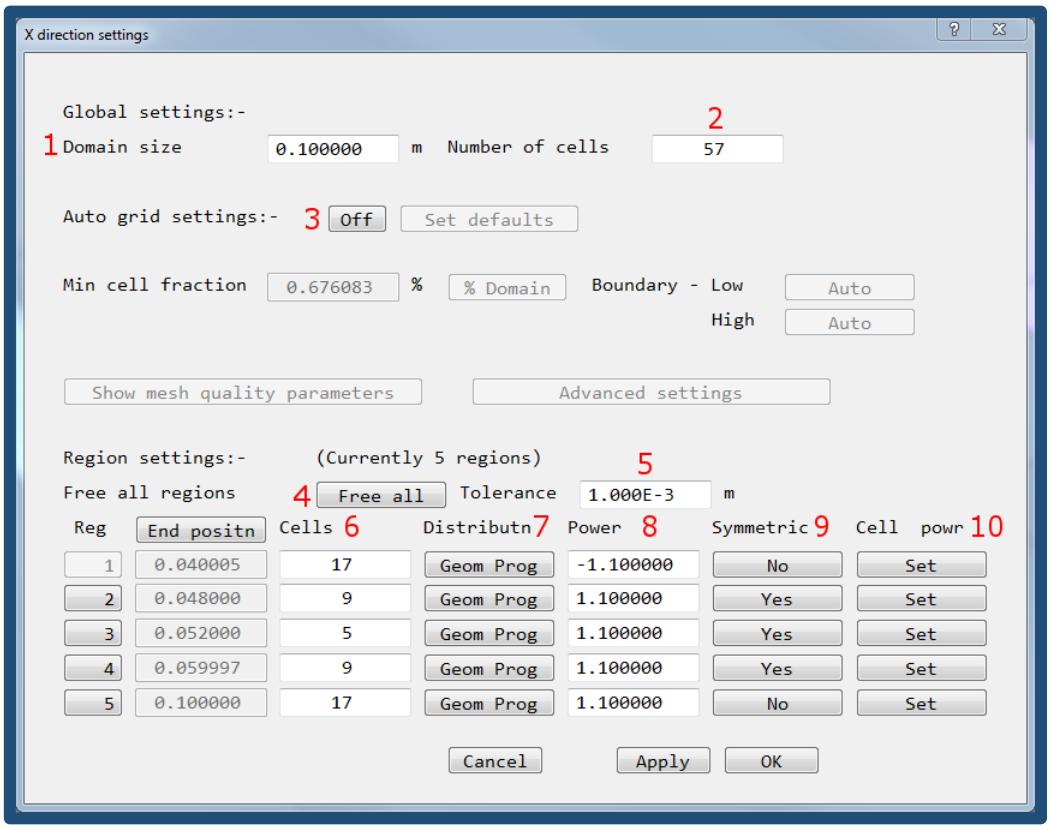

Figure 12: X Grid Dialog

1. Edit the size of the domain in the current coordinate.
2. Edit total cell number in current coordinate when auto grid is off (Click Apply to activate).
3. Turn auto grid on or off.
4. Frees all regions – where not specifically set by the user, the number of cells and the power/factor in a region cannot be changed unless the region is 'free'.
5. Edit region tolerance.
6. Edit number of cells in region.
7. Change distribution law – power law or geometric progression.
8. Edit power for power law or factor for geometric.
9. 'Symmetric' on/off - i.e. crush both ways.
10. Free individual regions.

6.9 Editing Regions

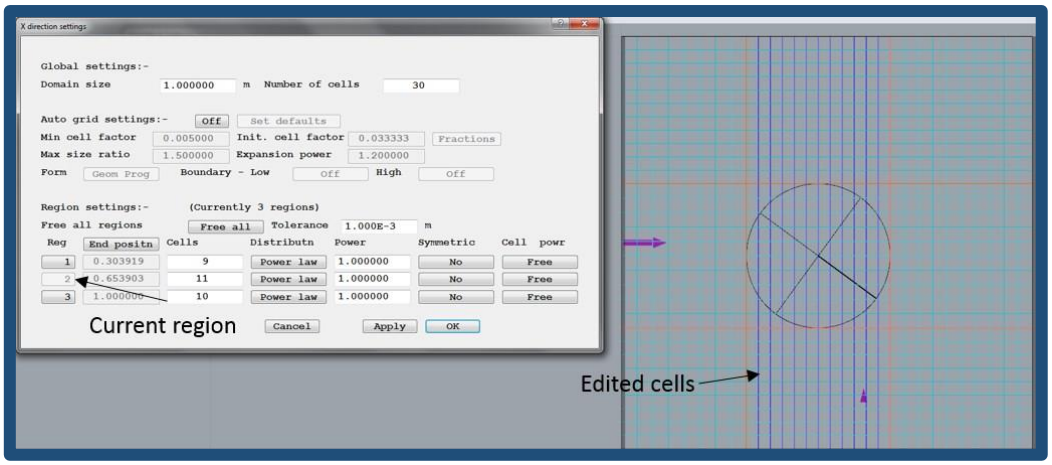

Figure 13: Example of Editing Regions

With complex geometry it is possible have many regions, which can make it hard to know which one is being edited. To help with this, RhinoCFD highlights the cells of the region you are editing after you click 'Apply'.

6.10 Best Practices

To help those new to CFD get to grips with creating and refining grids, here are a few best practices:

1. Grid expansion coefficients should not be greater than 1.3 if power distribution is used, or 1.1 for geometric distribution, and should not be less than 1.0. Use a negative coefficient to crush the cells in the opposite direction.
2. Neighbouring cells should have a width ratio no greater than 2:1, preferably less than 1.5:1.
3. Inlets and outlets should always 'affect the grid' (boxes ticked in Object Properties).
4. Objects do not always have to affect the grid, but areas of interest should.
5. Regions of complex geometry should contain enough cells to ensure that all geometry is properly detected.
6. RhinoCFD uses the 'PARSOL' feature to handle cut cells where the geometry is angled, curved or not aligned with cell faces. Cells should not be double cut in regions of interest - no more than a single cut should be seen in any cell.
7. In the object specification menu, 'Null' type objects can be created which have no effect but to create additional regions for the purpose of grid control.

6.11 Restarting with a Different Grid

It is possible to run a simulation with a small number of cells, and then refine the grid and restart from where the previous simulation finished as the solver interpolates from the old grid to the new. To activate this feature, run a simulation, change the grid and activate restart in the initialisation tab in the main menu (see Section 4.3). When 'run solver' is selected the following warning will appear:

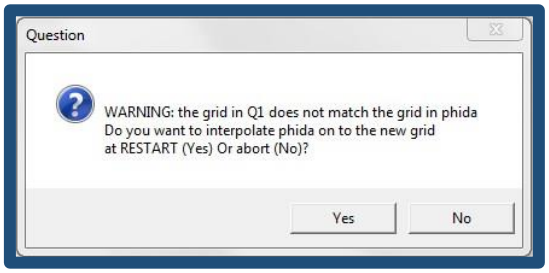

Figure 14: Grid Interpolation Warning

Click "Yes" to continue, and the simulation will begin. It is recommended that large grid changes are avoided and refinement is incremental as interpolation works best when cell difference is not large.

6.12 Coordinate Systems

Coordinate System	Description
Cartesian	The default grid coordinate system. This is a structured mesh in which cells stack in rows and columns in the x, y and z coordinate directions. Cells are 6 sided blocks (hexahedrals).
Cylindrical Polar	For use where the geometry has significant cylindrical features with a common axis. Here x must be the circumferential, y the radial and z the axial coordinate.

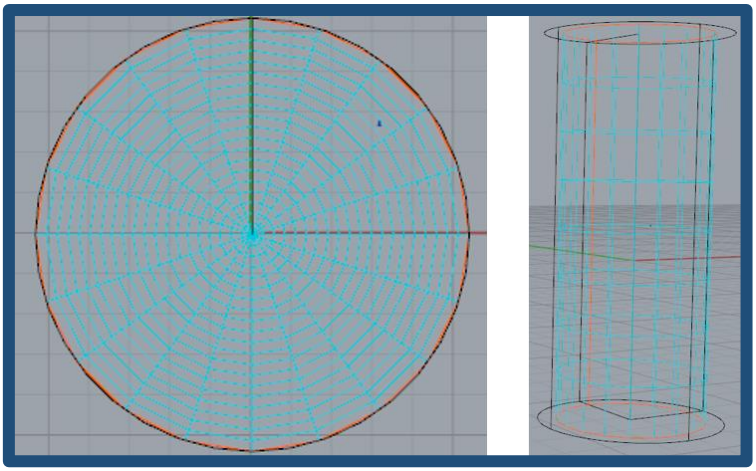

Figure 15: Polar Grid

6.13 Cylindrical Polar Simulations

A cylindrical polar grid can be selected by ticking the box next to 'Domain to use polar grid'.

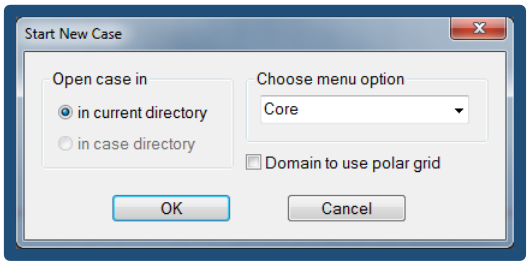

Figure 16: Start RhinoCFD with a Polar Domain

The grid can also be changed any time after starting RhinoCFD, by left clicking on 'Show Grid Dialog', and then clicking on 'Co-ordinate system - Cartesian' which brings up the following menu:

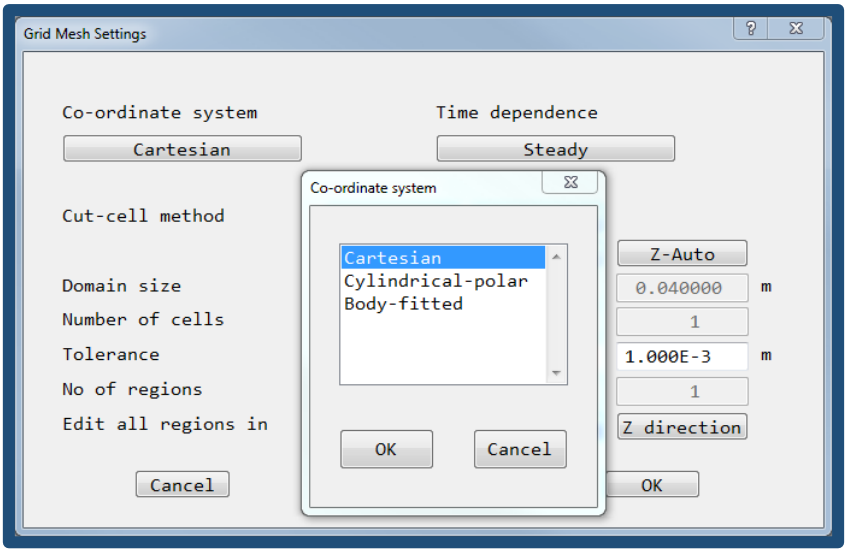

Figure 17: Coordinate Selection Menu

Note: the domain may have to be moved to fit geometry perfectly if the coordinate system is changed after RhinoCFD.

7 CFD Objects

7.1 The Use of Objects

RhinoCFD uses Rhino-generated geometries, known as CFD objects, which are three-dimensional geometrical entities. These CFD objects perform specific functions for CFD simulations, such as defining the geometry, describing fluid inflows and outflows, heat and momentum sources, etc. CFD objects may be given additional attributes relevant for specifying the CFD simulation. One set of attributes controls whether the geometrical extremities of the object are to be taken into account when the mesh is generated; this is referred to as "affecting the grid". Another set of attributes attach features such as material properties, flow rates, heat sources etc. to the object; these can then be used to define sources and boundary conditions for the CFD simulation.

Objects are created in standard Rhino environment and given CFD attributes using the RhinoCFD object dialog. Special CFD objects, such as Wind, can be created automatically via the RhinoCFD menu.

7.2 Creating CFD Objects

Objects are created in the standard Rhino environment and then given CFD settings using the object dialog. Special CFD objects, such as Inlets, can be created automatically using the RhinoCFD menu.

7.3 Object Specification Dialog

Selecting an object and clicking on the third tool bar icon  brings up the object specification dialog.

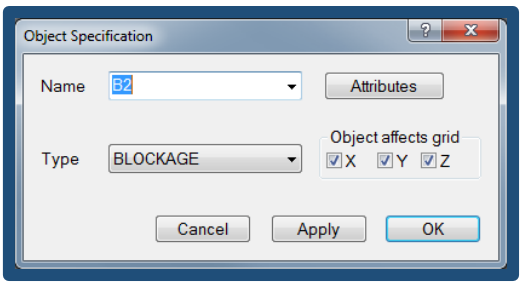

Figure 18: Object Specification Menu

This dialog specifies:

1. Object name.
2. Attributes, i.e. CFD properties - temperature, pressure, velocity, etc.
3. Object type: Blockage, Inlet, Outlet etc.
4. Whether or not the object creates grid regions ("affects the grid"):

It is important to note the following:

1. Different object types and options have entirely different attribute options.
2. Appropriate naming of objects will make them much easier to identify both in the Rhino environment and in the Q1 file (CFD input file - see Section 17.2).

7.4 Common Object Types

Objects created within the Rhino environment will automatically be classed as 'blockages' once a domain has been created. This can be changed, via the RhinoCFD tool bar, to a wide range of object types using the Object specification menu. The most common object types are explained in Table 3. Many more object types are available - please see either Appendix A or Object types in PHOENICS for more information.

Object Type	Description	Key Settings
Blockage	Volume object, used either to specify a solid region or a fluid region where for example a heat source or momentum source is located. Default material is "198 Solid", a non-conducting solid.	Material type, heat sources, porosity, momentum sources
Inlet	Used on the domain boundaries to specify a mass flow through it, either as an inflow or an outflow.	Mass flow rate, volume flow rate, velocity
Outlet	Used on the domain boundaries to specify a value for external pressure. Mass flow through it is calculated automatically based on the difference in pressure near the outlet and the prescribed value.	External pressure
Angled-in	Similar to Inlet, used to prescribe a fixed mass source or sink within the domain. It must intersect a solid blockage, lying partially inside it. The area specified as the Inlet is the plane of intersection between the blockage and the angled-in.	Mass flow rate, volume flow rate, velocity
Angled-Out	Similar to outlet, used to prescribe a fixed pressure region within the domain. It must intersect a solid blockage, lying partially inside it. The area specified as outlet is the plane of intersection between the blockage and the angled-out.	External pressure
Wind	Covers the entire domain and automatically sets boundary conditions for all domain faces based on the prescribed wind conditions. It also automatically applies wind profiles and initializes the solution to the prescribed profile.	Reference velocity height, access to EnergyPlus database, type of wind profile, upstream conditions

Null	Used exclusively to set up new grid regions for mesh control. Does not affect the flow in any other way.

Table 3: Common Object Types

8 Blockage Objects – Heat and Momentum Sources

8.1 Introduction

Blockage objects can be used to represent:

10. Blocked regions which do not conduct heat, and so for which no material properties are defined (these are known as “198” objects).
11. Blocked regions which do conduct heat, and are made of a particular material with specified physical properties.
12. Regions of fluid where something particular is specified - e.g. heat or momentum sources.
13. The attributes dialog varies, depending on various settings in the RhinoCFD menus, such as energy models, co-ordinate systems, object types etc. Some important and commonly used attribute settings are explained here and in the following subsections.

8.2 Non-thermally-conducting ‘198’ Blockage

The default object type for any object drawn in RhinoCFD is the non-thermally-conducting “198” Blockage; clicking on Attributes brings up the dialog shown in Figure 19.

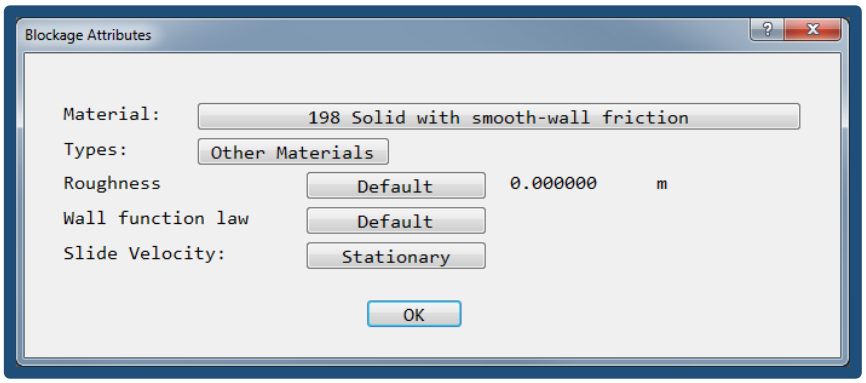

Figure 19: Default Attributes Menu

The Material setting for this form of Blockage is “198 Solid with smooth-wall friction”; this is the default setting. Click the relevant buttons if you want to:

1. Set a Wall roughness.
2. Select a non-default wall-function.
3. Specify a ‘slide velocity’ (i.e. if the wall is acting like a conveyor belt).

8.3 Thermally-conducting Blockage

Click the Types button on the Attributes dialog (Figure 20) to bring up a list of material types:

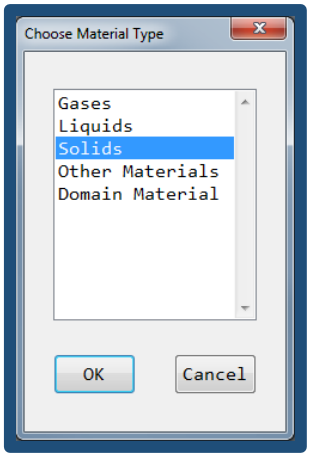

Figure 20: Material Types

Click Solids and select the appropriate material from the list. The purpose of this is to specify the properties of the material (density, specific heat, thermal conductivity etc.). The numerical values for the various materials are listed in Appendix C to this Guide. If you need a material with properties different from the materials in the list, you can easily specify these using the InForm feature.

When a solid material is selected, the Blockage Attributes panel will look like this:

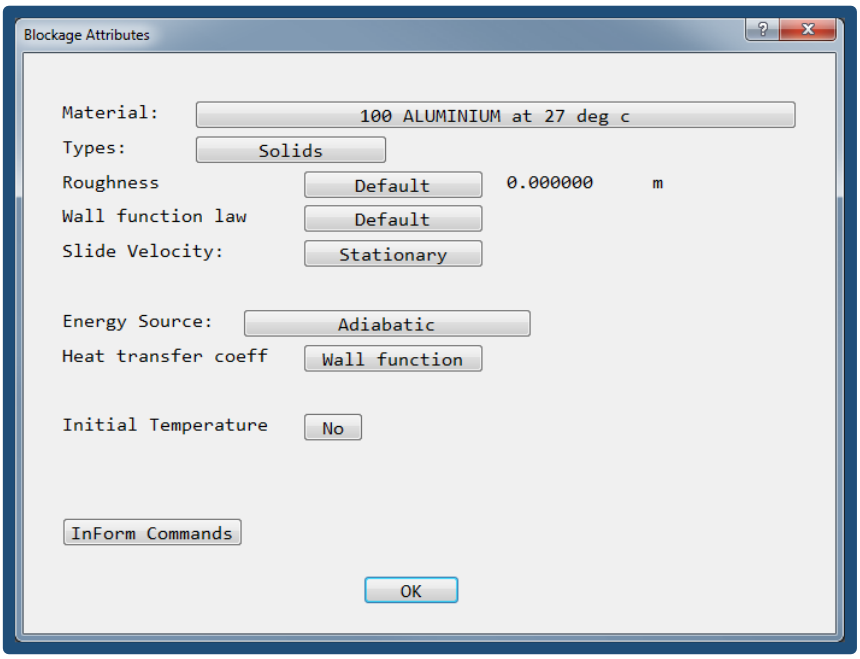

Figure 21: Blockage Attributes

In this example aluminium has been selected. '100' is the index number of aluminium in the materials list. Click the relevant buttons in the Attributes panel if you want to:

1. Set a wall roughness
2. Select a non-default wall-function law
3. Specify a "Slide velocity" (i.e. if the wall is acting like a conveyor belt)

It is possible to apply a heat source to a thermally-conducting Blockage, once the solution of the energy equation has been switched on in the Models section of the Main Menu. Clicking on Energy Source in the Attributes dialog will produce the following panel, where the type of energy source may be selected (these options are described in Section 8.4 below):

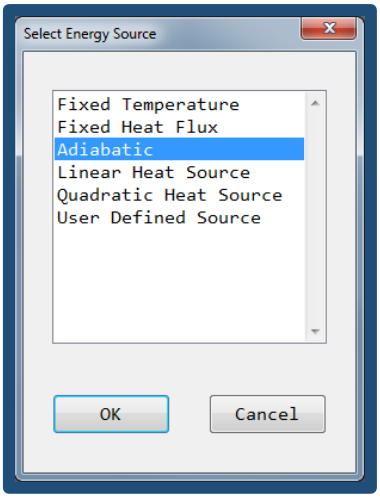

Figure 22: Energy Source Menu

8.4 Source Region in the Fluid

If the Blockage is intended to demarcate a region of the gas or liquid where a heat source or momentum source (for example) is to be applied, you should click the Types button on the Attributes panel to bring up the list shown in Figure 20 and then select Domain Material from the list. The term “Domain Material” describes the type of fluid flowing in the domain, as explained in Section 10.4.1.

The Attributes dialog will now be as shown in Figure 23.

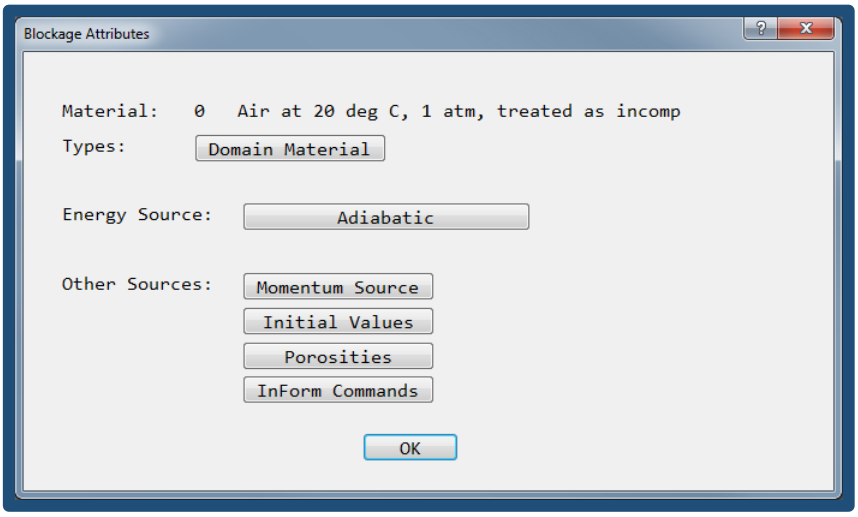

Figure 23: Domain Material

It is possible to apply a heat source to a domain-material Blockage, once solution of the energy equation has been switched on in the Models section of the Main Menu. Clicking on Energy Source in the Attributes dialog will give the following options explained in the table below:

Energy Source	Description
Fixed temperature	The temperature throughout the volume of the object is fixed to the specific value

Fixed Heat Flux	A heat flux throughout the object is specified, with the set value. The value can be specified as a total flux for the object, or as a flux per unit volume (units W or Wm3).
Adiabatic	There is no heat source. This is the default setting for a new object
Linear Heat Source	The heat source in any cell within the object is calculated from the expression $Q = Vol \times C(V - T_p)$ where V is an external temperature with units K or °C, C is a heat transfer coefficient with units W/m^3K, T_p is the local cell-centre temperature and Vol is the cell volume.

Table 4: Commonly used energy source options

Users can specify, within the region occupied by the Blockage, the velocity (m/s) or a force on the fluid (N), specified as three Cartesian components. Clicking on Momentum Source will produce the following panel, where the type of energy source may be selected:

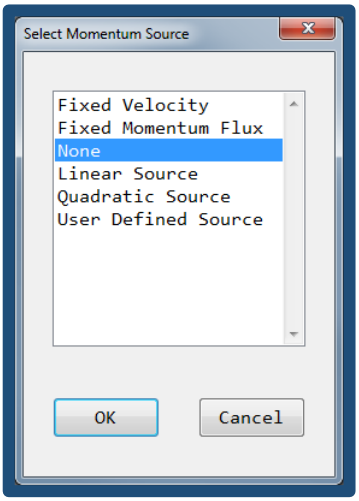

Figure 24: Momentum Source Dialog

These options are defined in the following table:

Momentum Source	**Description**
Fixed Velocity	The velocity throughout the volume of the object is fixed to the specified value
Fixed Momentum Flux	The momentum flux (force) throughout the volume of the object is fixed to the set value. The value is specified as a total force in Newtons for the object
None	There is no driving force. This is the default setting for a new object
Linear Source	The force in Newtons is calculated from the expression F =mass-in-cell $\times C$ ($V - Vel_p$). Where C and V are user-defined constants with units of $1/s$ and m/s respectively and Vel_p is the velocity in any cell within the object

Table 5: Commonly used momentum source options

9 Specifying Inflow and Outflow

9.1 Introduction

Inflows or outflows can be specified in two ways; either as a specified flow rate or as a fixed external pressure. With the latter, the flow rate is adjusted automatically to match the specified external pressure, and to balance the other flows in the system.

Sections 9.2 and 9.3 describe how such inflows and outflows may be specified in RhinoCFD. Section 9.4 discusses when you should specify a fixed flow rate and when a fixed pressure.

9.2 Fixed Flow Rate Sources

9.2.1 Fixed Flow Rate Sources on the Domain Boundary (Inlet)

To specify a fixed flow rate (i.e. inflow or outflow) at the domain boundary, an "Inlet" object should be used. The flow rate may be specified in one of the following ways:

1. Cartesian velocity components (m/s)
2. Volume flow rate (m^3/s)
3. Mass flow rate (kg/s)

If there is flow through an entire face of the domain, this is specified easily in RhinoCFD using the Domain Faces menu, accessed by right-clicking on the second toolbar icon  .

The following panel shows the attribute settings for Inlet objects.

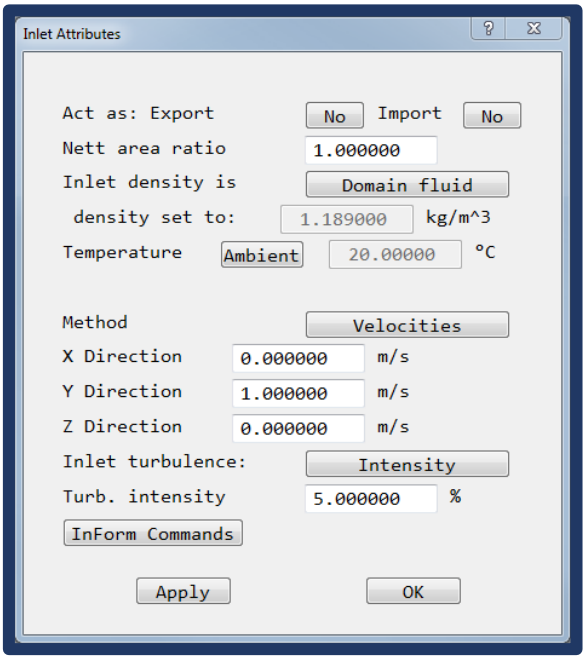

Figure 25: Inlet Attribute Settings

Note the following:

1. Flow definition method: velocity components, mass or volume flow rate, – can be selected.
2. Positive values for mass and volume flow rate indicate flow into the domain, negative values extract fluid from the system.
3. Positive and negative values for velocity represent the direction of the flow in relation to the Cartesian coordinates.

4. The temperature at Inlet is either user-set or “ambient” (default) - note that the ambient temperature value is set in the Properties panel of the Main Menu.
5. If scalars are solved (e.g. pollutant concentrations), appropriate values must be set at every Inlet.

The turbulent intensity at the Inlet, the density of the inflowing fluid, and the net area ratio for inflow through a grille may also be set. Details of these options may be found at TR326-Inlet.

9.2.2 Fixed Flow-rate Sources within the Domain (Angled-in)

To specify fluid entering or leaving within the domain, use an ‘Angled-in’ object. An Angled-in is used to define a region of fixed flow rate, either in or out. The region of influence is the part of the surface of any blockage object(s) enclosed by the Angled-in object. An Angled-in is used to delineate the part(s) of the surface of an internal blockage where flow is entering the system from the blockage, or being extracted from the system into the blockage. It should be placed such that it overlaps the blockage, to create an inter-sectional region. The amount the object protrudes from the blockage is not important, but it is recommended that the Angled-in extends at least one cell in thickness either side of the blockage surface. An example of an Angled-in is shown here.

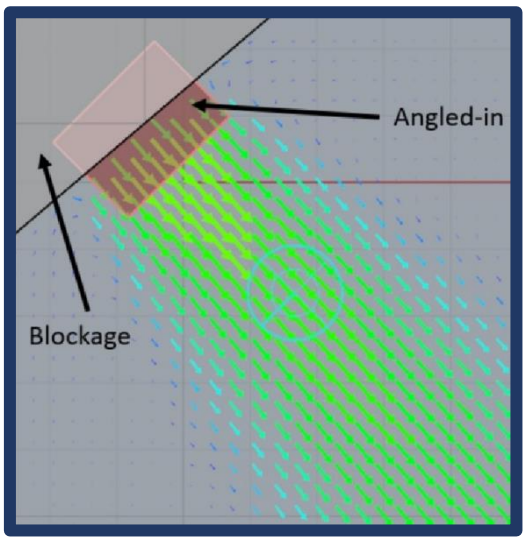

Figure 26: Example of use of Angled-in

The flow-rate through an Angled-in may be specified in one of the following ways:

1. Cartesian velocity components (m/s)
2. Normal velocity (m/s)
3. Volume flow rate (m^3/s)
4. Mass flow rate (kg/s)

“Normal velocity” allows users to specify velocity normal to the surface of the underlying blockage. If mass or volume inflow is specified, the direction of the inflow will be normal to the blockage surface.

The following panel shows the Attributes settings for Angled-in objects. They are very similar to the attribute settings for Inlets, described previously.

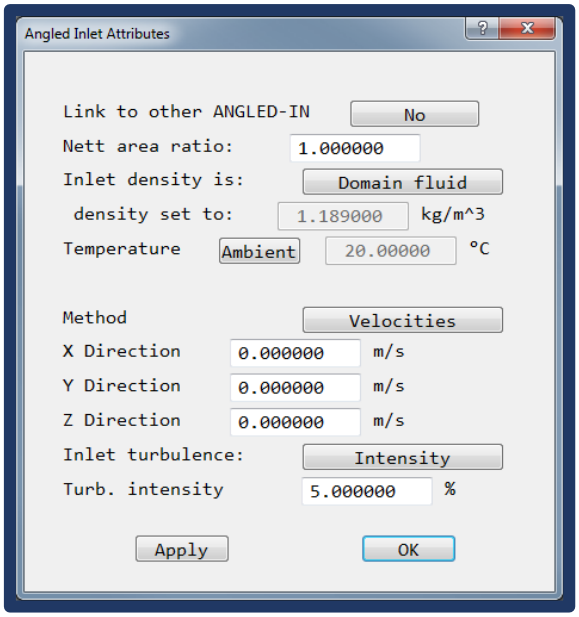

Figure 27: Angled-In Attribute Settings

9.2.3 Linked Angled-ins

Specific to Angled-in objects is the linking feature displayed at the top of the attributes panel. This can be used to represent the flow through a piece of equipment within the domain which is not modelled in detail, but is represented by a blockage. To do this we use a pair of linked Angled-in objects.

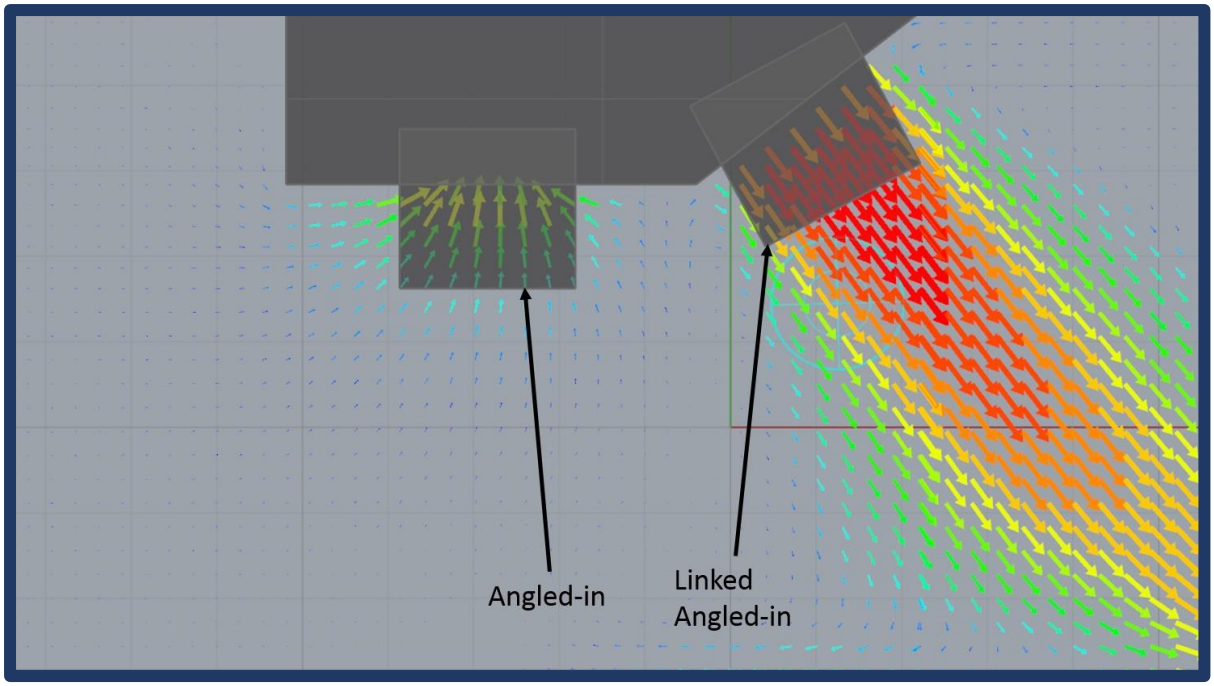

Figure 28: Linked Angled-ins

For example consider an induction fan, where air is sucked in on the underside of the fan and blown out sideways at an angle. An Angled-in can be used to represent the suction into the fan; here the suction rate is specified. The nozzle of the fan is represented as a second Angled-in, linked to the first; the linking ensures that the mass flow rates for the outflow nozzle and for the suction balance, and the temperatures (and any other scalars such as smoke concentration) balance. It is linking of temperatures at the suction side and at the nozzle for which the feature is required. An additional heat source can be specified if desired, representing any extra heating or

cooling taking place. Alternatively, the exit temperature can be specified, or the temperature rise (or fall) specified.

The mass flow is set at the suction Angled-in, and the temperature and the other scalars are based on the average values of the air sucked in. The 'nozzle' Angled-in should be linked to the suction Angled-in. Velocity at the nozzle will be deduced from the mass flow rate and the area. If the density is set to use the Ideal Gas Law (in the Properties section of the Main Menu), the density at the nozzle will be evaluated at the mean temperature of the air passing through the nozzle.

More information is available on Angled-ins at TR326 - Angled-in.

9.3 Fixed Pressure Sources

9.3.1 Fixed Pressure Sources on the Domain Boundary (Outlet)

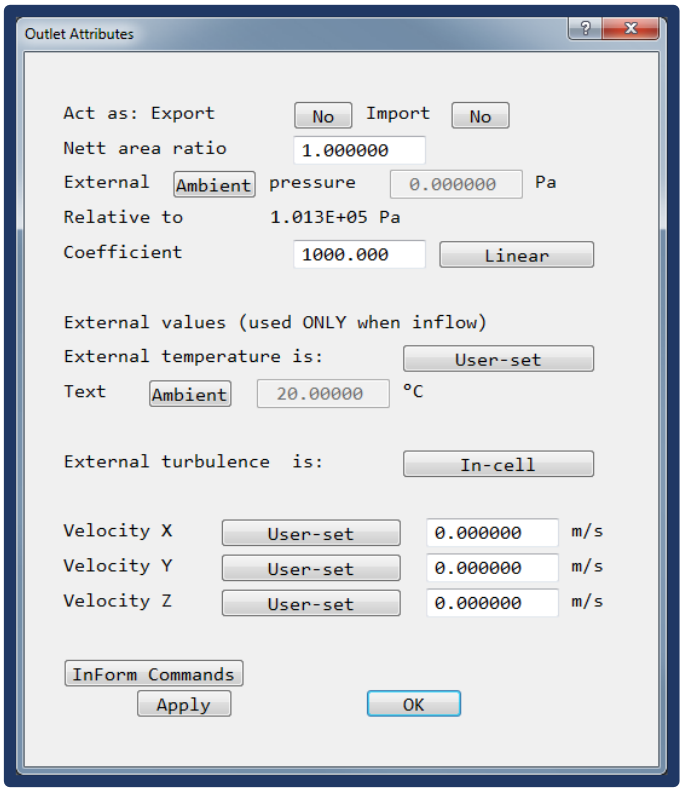

Figure 29: Outlet Attribute Settings

To specify a flow boundary with fixed pressure, located at the domain boundary, you should use an 'Outlet' object (known as an 'Opening' in FLAIR). The user specifies the external pressure (Pa), and the flow rate (kg/s) through the boundary is automatically determined as the pressure difference between this specified value and the internal pressure in the adjoining cell, multiplied by the user-specified 'coefficient', which therefore controls how closely the internal pressure matches the specified external pressure. The panel shows the attribute settings for Outlet objects.

In many cases there is no inflow at outlet objects. In this situation all the attribute settings may be left at their default. The discussion that follows relates to settings that the user may require on occasion, but in general may be left alone.

The Attributes menu provides options to:

14. Set the external pressure - this is defaulted to zero, and should usually be left at zero. This zero represents pressure relative to a reference pressure, usually taken to be atmospheric pressure. This reference pressure may be set in the Properties section of the Main Menu.

15. Set the temperature of the inflowing fluid - either User-set or Ambient (default). Note that the 'ambient' temperature may be set in the Properties panel of the Main Menu.
16. Set the value of any scalars (e.g. pollutant concentrations) of inflowing fluid.
17. There may be inflow, outflow, or both, at an Outlet/Opening. Temperature, scalar and velocity settings are relevant only for inflow. These are required only by advanced users; guidance may be found in TR336-Outlet.

If pressure is fixed over an entire domain face, this is specified easily in RhinoCFD using the Domain Faces menu accessed by right clicking on the second toolbar icon  .

9.3.2 Fixed-Pressure Sources within the Domain (Angled-out)

'Angled-out' objects are exactly analogous to Angled-in objects, as described in Section 9.3.1, except that, instead of fixing the flow rate, the external pressure is fixed. Available options are the same as for Outlet objects described in Section 9.3.1 and, as for Outlets, in many cases all the attribute settings may be left at their default values.

Angled-out objects **cannot** be linked in the same way as Angled-ins.

9.4 When to Use Fixed-Flow / Pressure Conditions

This section discusses when to use fixed-flow conditions (i.e. Inlets or Angled-ins) and when to user fixed-pressure conditions (i.e. Outlets or Angled-outs, or Openings in FLAIR). The references here will be to Inlets and Outlets, but the same considerations apply to Angled-ins and Angled-Outs.

This is best discussed by giving some examples:

1. If possible there must be at least one Outlet (or Angled-out) object, to provide a pressure reference for the solution domain. At this Outlet, pressure should generally be set to zero.
2. If there are no inflow or outflow boundaries, there must be a Pressure relief object (See Appendix A) to provide the pressure reference.
3. For a through-flow situation, i.e. with flow entering at one end of the domain and leaving at the other, we would generally recommend fixing the flow rate (using an Inlet) at the inflow end and fixing then pressure (using an outlet) at the outflow end. The solution will ensure mass continuity, i.e. it will force the mass outflow rate to be equal to the mass inflow rate.
4. Suppose that there are a number of air supply vents and a number of air extracts, all with known mass flow rate. As stated above, one of these should be represented as an Outlet/Opening. The others can be Inlets with the appropriate mass flow rates specified. The solution will ensure that the mass flow rate for the Outlet/Opening balances mass continuity
5. You may have a situation where you know the pressures, but the flow rates are unknown. For example in a naturally-ventilated building, the air flow will be driven by the pressures at the open windows and doors; and the objective of the study may be to determine the ventilation flow rate. In this situation, all the open windows and doors should be represented as Outlets/Openings. The pressure values at these Outlets/Openings will need to be specified. They might typically be taken from the results of an external CFD simulation.

10 Solution Parameters

10.1 Introduction

The aspects of setting up a CFD model which have been discussed above are all visual, in the sense that they are all concerned with geometry. Other non-visual parameters are also required to specify the model, e.g. what equations are to be solved, what is the fluid, what are the solids, what are the solution controls, etc. These aspects are all set in the "Main Menu" of RhinoCFD. For a full description of the Menu the reader is referred to

Section 9 of the PHOENICS VR User Reference Guide. A brief discussion of how to set the most important parameters is given in here.

To access the Main Menu, click the  icon in RhinoCFD. The Menu contains the following sections:

1. Geometry
2. Models
3. Properties
4. Initialisation
5. Sources
6. Numerics
7. Output
8. Domain Faces

The basic aspects of these are discussed in turn in the subsections below.

10.2 Geometry

This takes one to the "Grid Mesh Settings" panel which, alternatively, can be accessed by clicking  .

10.3 Models

In this panel the user can specify which transport equations are to be solved. A full description may be found here. In this section the most commonly used options are described.

10.3.1 Solution for Temperature

Solution for temperature can be switched on and off.

10.3.2 Turbulence Models

A large number of turbulence models are available in RhinoCFD. The complete set of models are described in the POLIS Encyclopaedia under Turbulence, where each has its own descriptive article. The turbulence models most likely to be of interest to the RhinoCFD user are the following.

Turbulence model	Description
LAMINAR	Flow is laminar and there is no turbulence model.
KEMODL	Classic two-equation k-epsilon model.
KECHEN	Default model. Chen-Kim improvement to the two-equation k-epsilon model. Gives better prediction of separation and vortices than the classic k-epsilon.
KEREAL	Realisable k-epsilon. Gives better prediction of separation and vortices than the standard k-epsilon.
KERNG	RNG-derived two-equation k-epsilon model. It gives improved prediction of separation and vortices Like Chen-Kim. However, the user is advised that the model results in substantial deterioration in the prediction of plane and round free jets in stagnant surroundings.

LVEL	A zero-equation algebraic model based on distance from the wall. Good for channel flows and "cluttered" spaces where there is not enough mesh to resolve boundary layers properly.

Table 6: Turbulence Models in RhinoCFD

10.3.3 Radiation

The "Immersol" radiation model can be switched on and off. This is an economical but approximate diffusion-based model. Theoretical details may be found here.

10.3.4 Variables

The "Solution control/extra variables" button enables quick review of which variables are stored, and which solved, and by what method. Here "variable" means a quantity which has a value stored at every cell in the domain; a variable being "solved" means that the variable is a conserved quantity subject to a transport equation.

The Y/Ns are toggles, Y indicating Yes and No; clicking on one changes Y to N or vice versa.

The top line "SOLUTN 1 STOR" indicates whether or not each variable is stored. They are all Y, because if you click one to change it to N, that variable is no longer stored, and it disappears from the table.

The second line "SOLUTN 2 SOLV" indicates whether or not each variable is solved (i.e. has a transport equation which is solved).

The third line "SOLUTN 3 WHOF" indicates whether the linear solver for the variable is the 3D whole-field solver (Y), or the 2D slab-by-slab solver (N). Pressure, temperature and scalar variables such as pollutants or smoke should always be solved whole-field.

To solve an additional transport equation, e.g. for a pollutant such as CO, type CO in the "SOLVE" box and click Apply. Be sure to set CO to be solved whole-field.

10.3.5 AGE

This enables solution for a variable AGE which represents the mean age of air since entry to the domain. Regions with high values of AGE can be useful to indicate poorly-ventilated regions.

10.4 Properties

10.4.1 Domain Material

The term "Domain Material" describes the type of fluid flowing in the domain. It may be set to any of the gases or liquids listed by clicking "Gases" or "Liquids", but almost always one of the following will be appropriate.

Gases

0 air modelled as incompressible

2 air modelled as compressible using the ideal-gas law

Liquids

67 water

For gases numbers 0 and 2 listed above, note that these numbers not only indicate that the fluid is air, but also what density formulation is to be employed.

10.4.2 Reference Pressure

When using RhinoCFD the pressure at one or more of the fixed-pressure boundaries ("Outlets") should generally be set to zero. This represents the pressure relative to atmospheric; the absolute value of this pressure reference should be set here (default value 101325 Pa).

10.4.3 Reference Temperature

This should be 0 if working in degrees C, 273 for degrees K.

10.4.4 Ambient Temperature

The 'Ambient temperature' setting is be used for two purposes:

1. To initialise the temperature at the start of a simulation (only if "Initialise from ambient" is set),
2. To specify the temperature at which the reference density is set, the latter being used in specification of the buoyancy force (only if "Set buoyancy from ambient" is set). See Section 10.6 "Sources".

The "Initialise from ambient" temperature setting enables easy specification of the reference density, and so is generally advisable when there are temperature variations. Note that the ambient temperature does not necessarily represent the temperature of external air. For flow within a contained space, it should generally be a typical value of the temperature within the space.

10.5 Initialisation

Here the initial value may be specified for each variable. The variable is set to this value over the whole domain at the start of the iteration.

Generally, it is not necessary to select initial values in a steady (i.e. time-invariant) simulation. An exception is temperature, but this is initialised automatically if "Initialise from ambient" is selected in the "Properties" menu (see 10.4.4).

In transient solutions it is very important to set the initial conditions.

In general, the principal use of the Initialisation menu will be to:

1. Select "Activate restart for all variables", to specify a restart run, i.e. to continue the solution from a previous run - "Name of restart file" specifies the file to be used for restart data; or to
2. "Reset initial values to default" to undo this and start afresh.

10.6 Sources

This panel allows the creation of whole-domain sources, which are not attached to any specific object. All sources or boundary conditions which do not apply to the whole domain must be attached to an object, and set through the appropriate object attribute dialog box. A full description may be found here. In this section the most commonly used options are described.

10.6.1 Gravitational Forces

Here you may switch these on or off.

10.6.2 Buoyancy Models

The most frequently used models are the following:

1. Constant - Applies the full gravitational force.
2. Density Difference – Bases the buoyancy force on a density which has a "reference density" subtracted. This has the effect of removing the hydrostatic pressure gradient.
3. Boussinesq Approximation – Bases the buoyancy force on temperature, rather than density, difference. Can be used with constant-density setting. Only use for small temperature variations.

All are described in the Encyclopaedia under Gravitational body forces. If 'Set buoyancy from ambient' on the Properties panel is set to ON, the reference density will be computed from the ambient temperature set in the Properties panel.

10.6.3 Gravitational Acceleration

Allows Cartesian components of the gravitational force vector to be set. The Cartesian z-axis is commonly taken as the vertically upwards direction therefore the default setting for the gravitational force vector is (0, 0, -9.81).

10.6.4 Reference Density

Can only be set if "Set buoyancy from ambient' on the Properties panel (Section 10.4.4) is set to OFF.

10.6.5 Buoyancy Effect on Turbulence

In turbulent flows, gravity interacts with density gradients to affect the turbulence. In stably-stratified flows (dense below light), turbulence is damped; in unstably-stratified flows (light below dense), turbulence is augmented. Clicking this ON introduces terms which take this effect into account. These terms are not included by default, as they can have an adverse effect on the stability of the simulation. For situations with temperature stratification it can be advisable to switch on this option.

10.7 Numerics

This is where the total number of iterations ("sweeps") for the simulation are set. Typically, thousands of iterations may be necessary for full convergence to be achieved.

It is sometimes recommended to reduce the "Global convergence criterion" from 0.01 to 0.001, as this can prevent premature cut-off of the solution.

If changes to the relaxation parameters are required (see Section 13), these may be set in the "Relaxation Settings" panel accessed from the "Relaxation control" button.

10.8 Output

Full details of the extensive output settings and controls may be found here. This section describes a few of the most commonly useful settings.

1. At the top of Field Printout Settings panel is a check-box labelled "Suppress all field printout". This should always be checked. It eliminates non-essential output to the Result file, thereby enabling attention to be focussed easier on the convergence information.
2. NPRINT (in the Field Printout submenu) controls sweep frequency of Nett Source printout to the Result file. While not essential it is often useful. Setting NPRINT to 250 may be convenient.
3. It can be useful to specify intermediate field dumps at regular intervals. This is achieved from the Field Dumping submenu, by setting Intermediate field dumps to ON, and then specifying the frequency for the dumps.

10.9 Domain Faces

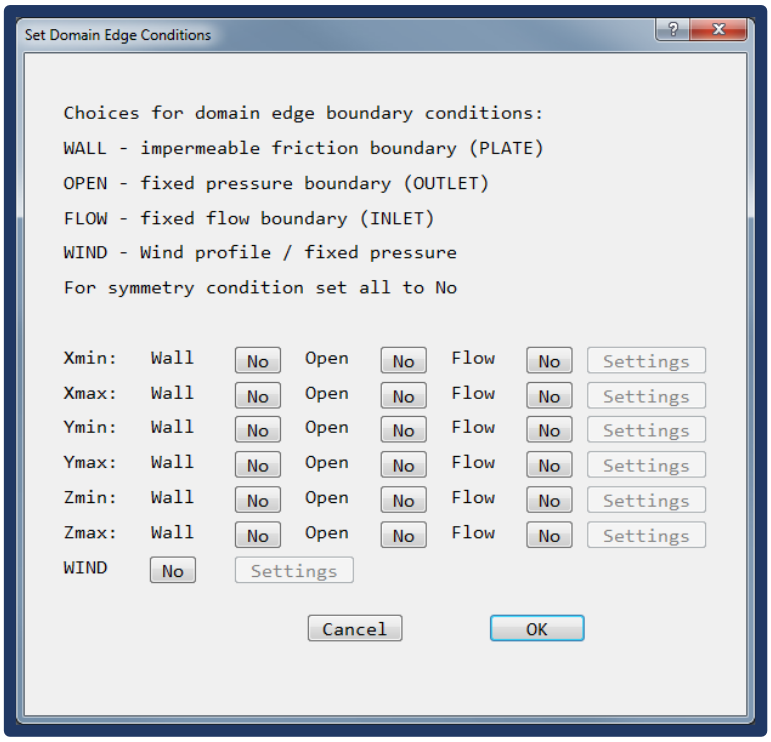

Figure 30: Domain Face Setting

This panel allows boundary conditions applying to entire faces of the domain to be set up rapidly. Xmin, Xmax, Ymin, Ymax, Zmin, Zmax denote the faces of the domain at the "low" and the "high" end-faces of the domain, in the X, Y and Z directions respectively. The available options are:

Domain face	Function
Wall	Provide wall friction and heat transfer appropriate for a wall boundary
Open	Create an Outlet object at domain boundary with fixed external pressure
Flow	Create an Inlet object at domain boundary to specify mass inflow or extraction rate
Wind	Creates Wind object for easy specification of wind profile and direction. Boundary conditions on the six faces will be created automatically.

Table 7: Domain face Dialog Options

Yes/No toggles may be set quickly to implement required conditions. Once OK is selected and menu is closed, the relevant objects will appear in the domain.

11 The Probe

The Probe is shown as  in RhinoCFD, which can be positioned by the user anywhere in the domain.

When setting up a model the probe represents the location used to monitor solution convergence. It will appear at the centre of the domain initially and may be repositioned by the user as desired.

When post-processing the results of a simulation, the probe may be used to extract values of variables at a particular point.

Useful tips about using the probe:

1. The probe should be located, preferably, out of the mainstream flow, perhaps in the wake of an obstacle; downstream rather than upstream, and not adjacent to any boundary.
2. The probe should not be located inside a solid.
3. Viewing the probe – the location can be found easily by left-clicking on toolbar icon  .
4. The probe can be hidden by right-clicking on the same icon.
5. While viewing the grid and the results, the current plotting plane passes through the probe position; therefore, the viewing plane can be changed by moving the probe.

It is important to note that the position of the probe will not affect the simulation in any way. It is a tool for investigating and monitoring simulations.

12 Running Simulations

12.1 Starting a Simulation

To run a simulation, left click on the 'run solver' icon in the RhinoCFD toolbar:

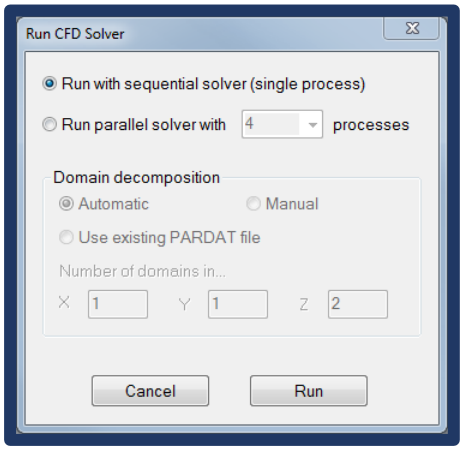

Figure 31: Run CFD Solver Settings

If you have a sequential-only licence, this will begin the simulation. If you have a parallel licence, the following screen will appear asking for the number of processors on which to run the simulation; automatic domain decomposition is recommended.

Clicking on 'Run' will start the simulation and the convergence monitor plot will appear. Monitor plots can be used to track the convergence of the simulation, as explained in the next section. When the simulation is completed the monitor plot will close and the results can be viewed. For information on convergence see Section 13.

12.2 Monitor Plots and Interactive Options

The monitor plot is launched when a simulation is running. It gives real time graphical and numerical data indicative of the convergence of the simulation. The monitor plot screen can be used to edit settings during a simulation without changing the Q1 file. Pressing any character key pauses the run, and displays the in-run options menu as shown in Figure 32.

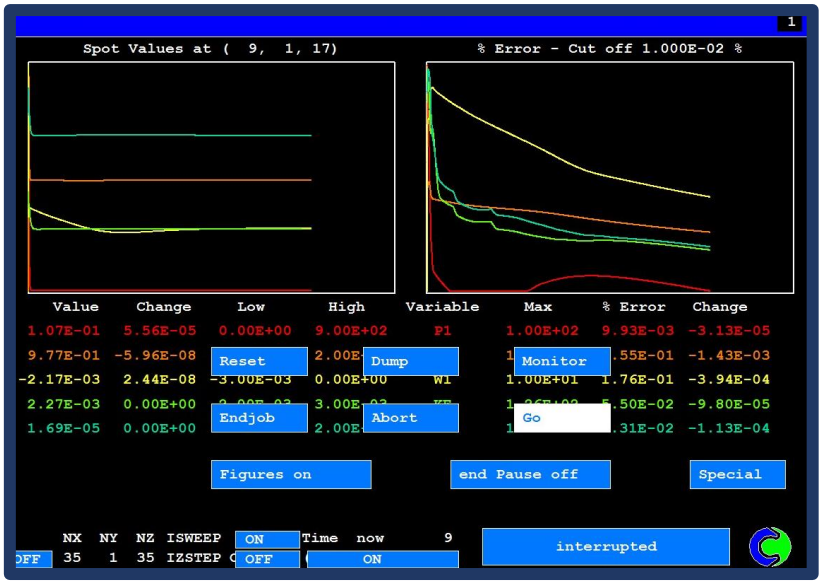

Figure 32: Monitor Plot Menu

The action of the buttons is as follows:

1. Reset – change relaxation settings / number of sweeps.
2. Dump – dump data from current iteration; these results can then be viewed on the screen.
3. Monitor – change probe position of the probe / change window width / toggle monitor plot type.
4. Endjob – End simulation and dump all data from current run.
5. Abort – End the run without saving any data from simulation.
6. Go – End pause, continue with simulation.
7. Figures on – Toggle figure on or off.
8. End pause off – Setting to leave the convergence monitor on screen after run is completed.

Toggling the monitor plot type allows access to a plot of Maximum Absolute Corrections, which is another useful indicator of convergence (see Section 13.1).

13 Convergence and Relaxation

13.1 Introduction

A CFD simulation involves solving conservation equations for variables such as mass, momentum and energy. These equations are highly nonlinear and, as a result, have to be solved iteratively. It is essential that, during the iterative process, the solution will "converge", meaning that variables tend towards asymptotic values, and errors in equations reduce in size and (hopefully) become very small. The idea is that after a certain number of iterations (dependent on the complexity of the simulation) the process will converge to a solution in which all equations are satisfied with minimal error.

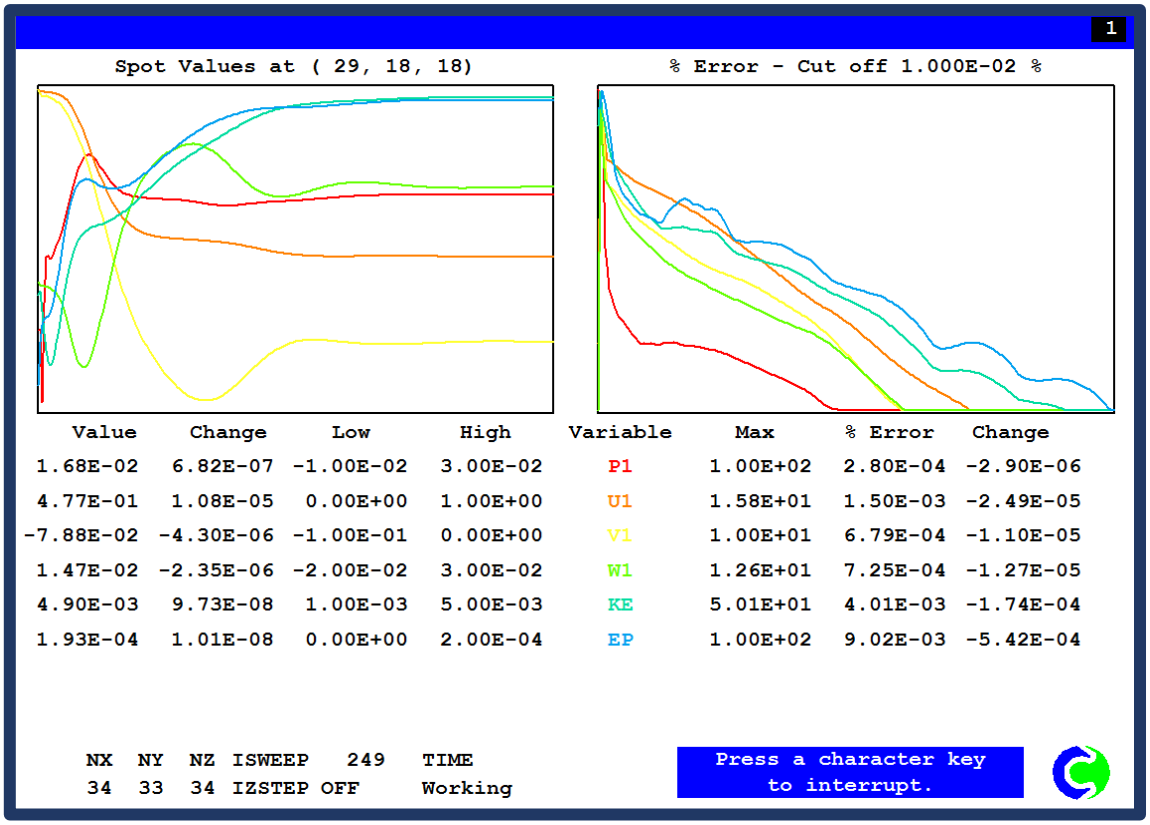

Figure 33: Spot Value and Residuals Plot (Steady Simulation)

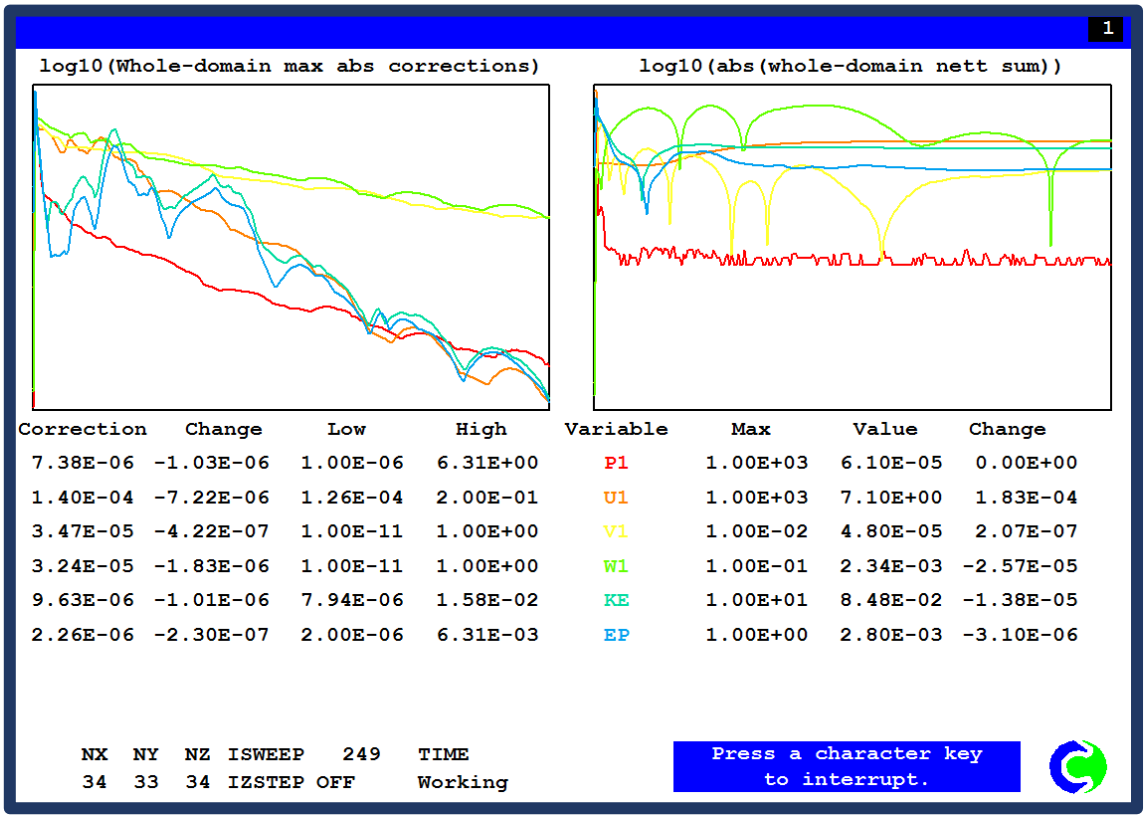

Figure 34: Maximum Absolute Correction Plot (Steady Simulation)

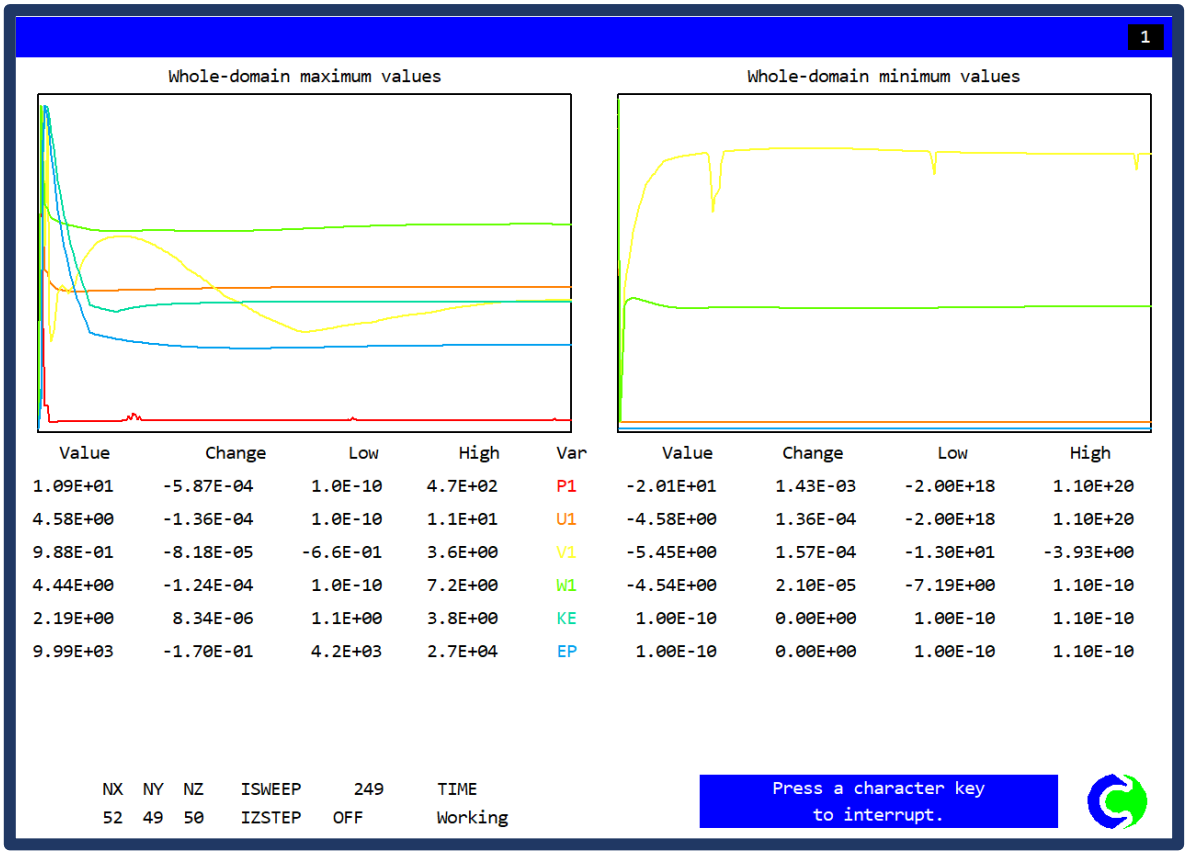

Figure 35 Maximum and minimum values in the domain of each variable (Steady Simulation)

If relaxation controls are set inappropriately, the solution may diverge which is what the CFD modeller does not want to see. This section discusses how to recognise convergence and avoid divergence, and how to assess when a simulation is adequately converged.

Figure 33 shows a typical example of the convergence process. The left-hand curves show values of variables at a given location (the probe) converging asymptotically to a solution; the right-hand curves show the errors diminishing. In these plots each coloured curve represents one of the equations.

This section explains how to assess whether your run is converging, and how to use a technique known as “Relaxation” to ensure that it does converge.

13.2 Checking Convergence

13.2.1 Method 1 - Use of Graphical Data during Simulation

Locate the probe somewhere ‘interesting’ – downstream from the inlet and away from solids (See Section 11.) When running simulations, check out the graph for ‘spot values’ on the left hand side. When all variable lines become horizontal, it means that the run is converged. On the right is another plot labelled ‘% Error’; this is a logarithmic plot showing sums over the whole field of the imbalances in variable flows on a cellwise basis. The curves in the right-hand plot may show irregularities, but should show a steady fall. This fall in the errors, coupled with stabilising of the values in the left-hand plot, are indicative of convergence. The simulation will terminate if the lines on the right-hand plot all fall below the horizontal axis, the height of which is set by a variable RESFAC in the “Numerics” menu.

If the % Error values fall below 1% for all variables, convergence may generally be deemed acceptable.

Two examples of monitor plots which show good convergence characteristics can be seen in Figures 33 and 34; these are typical for steady and for transient runs respectively.

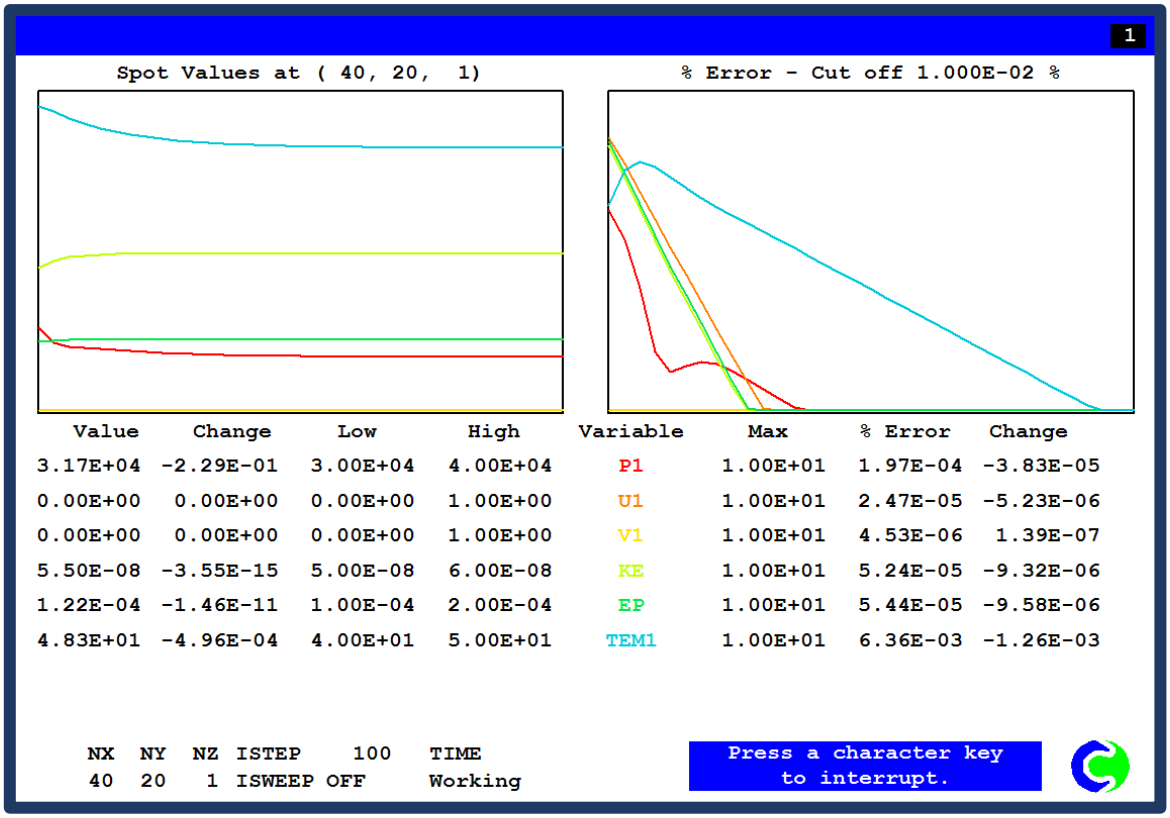

Figure 36: Spot Value and Residuals Plot (Transient Simulation)

Alternative convergence plots can be selected, namely the maximum absolute correction of each variable in the domain (which ideally should tend to zero) and the maximum and minimum values of each variable in the domain, which should become stable values (see Figures Figure 34 and Figure 35)

13.2.2 Method 2 – Reviewing the Nett Sources

Further and more detailed information about the level of convergence can be found in the "Result" file, accessible from the toolbar or from your working directory. In it you will find quantitative information regarding the residual errors - and, most importantly, "Nett Source" information, which lists the following information for each variable:

1. Inflow rate for each fixed-inflow object (kg/s for mass R1, W for heat flux TEM1)
2. Outflow rate for each fixed-outflow object (kg/s for mass R1, W for heat flux TEM1)
3. For each fixed-pressure object, mass inflow and outflow rates plus average temperatures
4. Total inflow rate for all objects ("pos. sum")
5. Total outflow rate for all objects ("neg. sum")
6. "Nett sum", indicating total in/outflow for the variable for all objects

For good convergence, the nett sums should be several orders of magnitude smaller than the individually itemised flow rates.

It is good practice to check the nett sources for mass (R1) and temperature (TEM1), as well as for any variables representing pollutants or smoke, both to check convergence, and to check that all the specified mass, heat and pollutant sources are being recognised.

```
Sources and sinks
!! Zero nett sources are not printed !!!

   Nett Sources have units of mass_per_unit_time * variable
   Average values have units of the variable

   Typically the units of the sources are:
       U1,V1,W1 - Force  - Newtons
       R1       - Mass   - kg/s
       TEM1     - Energy - Watts

Nett source of U1   at patch named: OB1      (B1          ) =-5.034943E-03
Nett source of U1   at patch named: OB2      (DOM_XMIN_I  ) = 1.042156E+02
Nett source of U1   at patch named: OB3      (DOM_XMAX_O  ) =-1.087048E+02
pos. sum=104.215569 neg. sum=-108.709862
nett sum=-4.494293

Nett source of W1   at patch named: OB1      (B1          ) = 1.180570E-08
Nett source of W1   at patch named: OB3      (DOM_XMAX_O  ) = 3.541401E-05
pos. sum=3.542581E-05 neg. sum=0.
nett sum=3.542581E-05

Nett source of R1   at patch named: OB2      (DOM_XMIN_I  ) = 1.042156E+02
Nett source of R1   at patch named: OB3      (DOM_XMAX_O  ) =-1.042156E+02 (Mass Out -1.042156E+02
pos. sum=104.215569 neg. sum=-104.215553
nett sum=1.525879E-05
```

Figure 37: Sources and Sinks in Result File

Note that if the Nett sums of R1, TEM1, pollutants etc. are not small, then the run is not converged. But these sums being small is not a sufficient condition for convergence – you have to check the graphical plots as well (see above).

13.3 Relaxation

The most widely-known technique for achieving convergence is the use of “Relaxation”. This has an effect akin to damping, and slows down (relaxes) the changes made to the variables from one iteration to the next. It is important to note that relaxation does not alter the final solution of the equations, only the way that that solution is achieved.

Two types of relaxation are available - linear (“LINEAR”) and false-time-step (“FALSDT”). Here we will not look at the theory behind these, just at their practical application. Relaxation is a big subject, and only an outline guide will be given here.

When the user sets up a model in RhinoCFD, an automatic convergence tool known as “Conwiz” is employed. This sets up relaxation parameters based on a user-set reference velocity and length scale, and applies linear relaxation to each of the variables. It also limits the maximum increment of a variable between sweeps, and applies further internal relaxation to the pressure-correction equation. However this automatic tool is sometimes over-conservative, leading to run times which are longer than they need be. Some experimenting with manually set relaxation parameters can therefore pay dividends. The default length and velocity scales are 1 metre and 10 m/s. If these are inappropriate for the model in question, they should be reset to more representative values before trying manual relaxation.

To activate manual relaxation, enter the main menu, click on Numerics and then on Relaxation control. “Automatic Convergence control” should be switched off. You will then see that each variable has a “RELAX” method (LINEAR or FALSDT) and a “VALUE”. Describing this in detail is a complex topic, so here we will discuss only the basics.

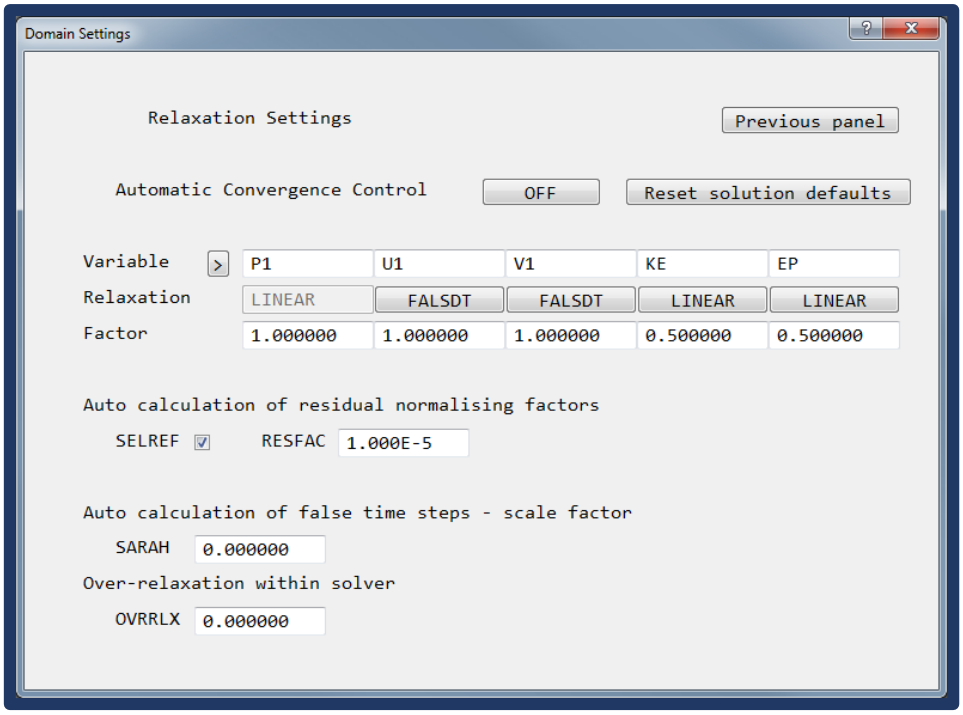

Figure 38: Relaxation Settings

Relaxation for the pressure P1 is set to LINEAR with a value of 1 - do not change this. The velocities U1, V1 and W1 should be set to FALSDT, and the value (i.e. the relaxation parameter) should be set to a typical characteristic time for the flow, in seconds. Often a suitable value may be found by estimating how long the flow might take to flow through a single cell in a region where the mesh is fairly fine. In other words, take a typical flow velocity in m/s, and divide it by the width of one of the smaller cells (in metres). Use the same number for all three velocity settings. For air flow in and around buildings, we often find that a FALSDT value of 0.1 seconds is satisfactory.

For temperature, if there are thermally conducting solids present, use LINEAR with a value of perhaps 0.3. If there are no conducting solids you can do the same, or use FALSDT with the same numerical value as for the velocities.

The relaxation for the turbulence variables (i.e. the turbulence kinetic energy KE and the dissipation rate EP) should be left unchanged.

More information on convergence can be found here.

14 Transients

14.1 Introduction

RhinoCFD can handle both steady and transient simulations.

Steady simulations are independent of time and assume that the boundary conditions are constant. Examples are a steady ventilation pattern in a building, or steady flow over an aerofoil. It is important to be aware that in a steady simulation, the iterative solution process does not represent the actual physical time-development of the flow.

In a transient simulation, the velocity pattern and the temperature distribution can change with time. An example of a transient simulation is the development of a fire in a building, to predict the build-up of the smoke layer, so as to determine the time available for escape. Transient simulations are computationally more expensive than steady ones.

When you enter RhinoCFD the simulation is set to be steady by default.

14.2 About Transient Runs

It is important to note the following.

1. A transient run is solved as set of time steps.
2. Each time step is iterative and requires sweeps.
3. Convergence curves are displayed for every time step.
4. The time steps form a “grid” in time.
5. There must be enough time steps to adequately resolve process being modelled.
6. A transient run might typically require one or several hundred steps.
7. It is generally helpful if the time steps are uniform, although this is not necessary.
8. The solution uses “implicit” differencing, meaning there is no theoretical limit on step size.
9. However, larger time steps may compromise accuracy.

14.3 Setting up a Transient Model

14.3.1 Changing From Steady to Transient

In order to switch from steady to transient, enter the Main Menu by clicking  . Click on Geometry, and then on the button that says ‘Steady’. It will change to “Transient”, and an extra button to edit time step settings will appear.

14.3.2 Setting Time Step Duration

In the Time Step Settings panel, to set the real time length of your simulation, input your desired value in seconds into the box labelled “Time at end of last step”.

Enter the number of time steps required for the simulation into the box labelled “Last step number”, and then click “Apply”. This divides the total time for your simulation by the total number of steps and thus defines the size of your time step.

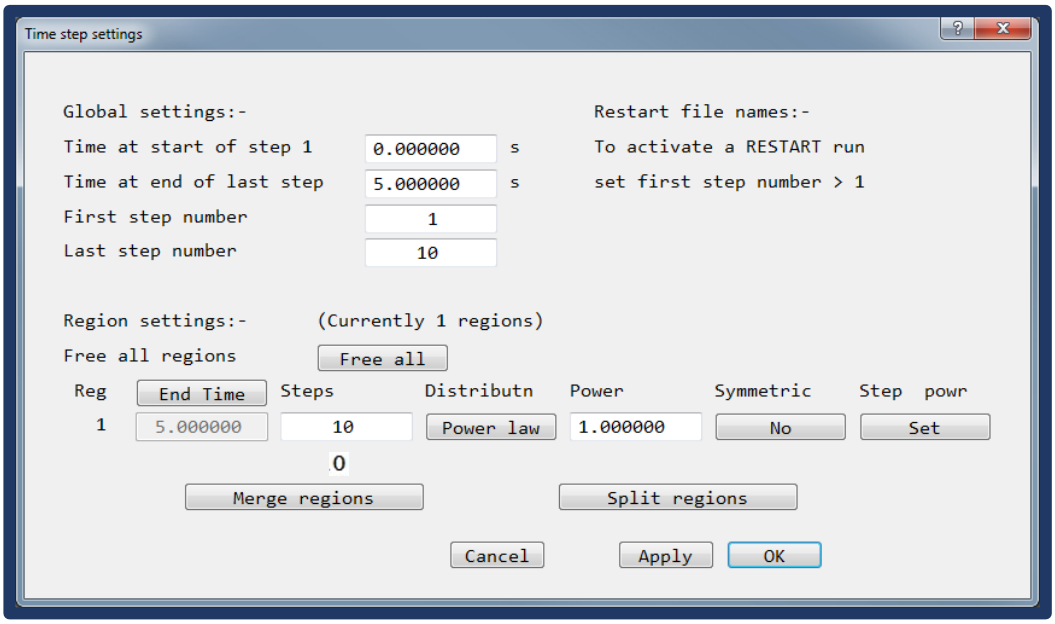

Figure 39: Time Step Settings

14.4 Setting Initial Values

In steady-state simulations the initial values serve only to give the solution an initial guess to start the iteration. In a transient, however, the initial values represent the initial state of the system, which is of vital importance in determining how the system evolves in time thereafter.

If the initial state can be specified using uniform values for velocity, temperature and all the variables, then the appropriate values can be entered in the “FIINIT” boxes of the “Initialisation” panel of the Main Menu. See Appendix variables for an explanation of the individual

Variables.

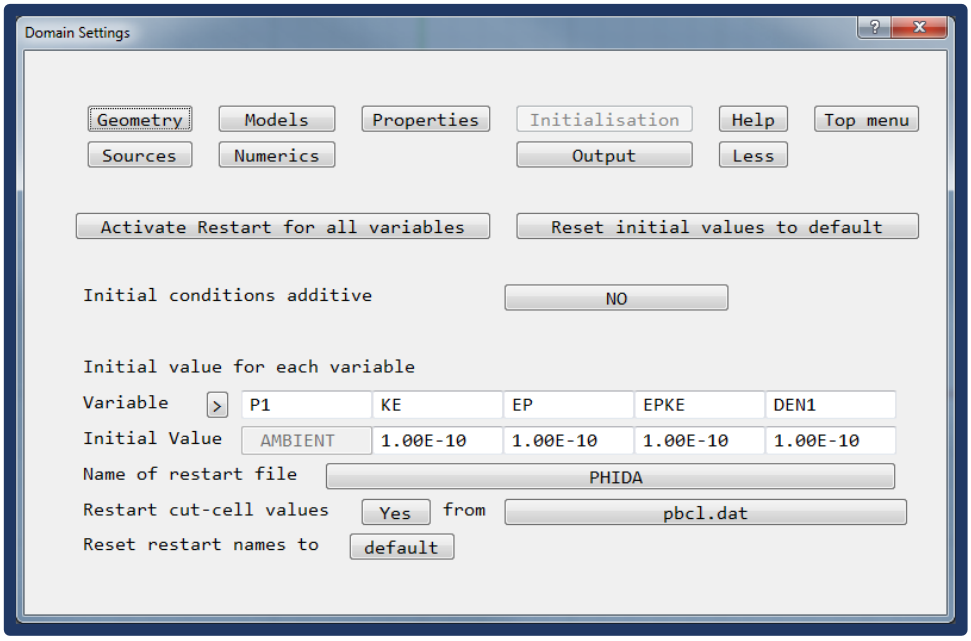

Figure 40: Initialisation Menu

Sometimes, the initial state might have non-uniform velocities or temperatures. For example, to simulate flow over a cylinder sheds vortices the user would first need to perform a steady simulation of flow over the cylinder to obtain a velocity field from which to start the transient simulation.

To do this, one runs the steady simulation, and saves the files by left-clicking  and providing a file name; this will apply to all the files created, with different extensions. The solution file from which the transient is to be restarted will be named <filename>.pda (where <filename> is the name provided when saving), and this will need to be entered in the “Name of restart file” box in the “Initialisation” panel of the Main Menu for the transient run. Note that “Restart cut-cell values” should be set to NO.

14.5 Number of Sweeps

A typical steady solution might require several thousand sweeps (iterations). However, a typical transient run might need only 20 to 50 sweeps per time step because in a short time (such as a single time step) nothing can change very much. If more than 100 sweeps are required to converge the individual time steps, use of shorter time steps should be considered. The total computational expense of the solution is proportional to the number of time steps multiplied by the number of sweeps. So if you increase the number of time steps, but thereby reduce the number of sweeps per step, this may not have a significant effect on the overall run time.

The number of sweeps per step is set in the Main Menu “Numerics” panel (obtained by clicking ).

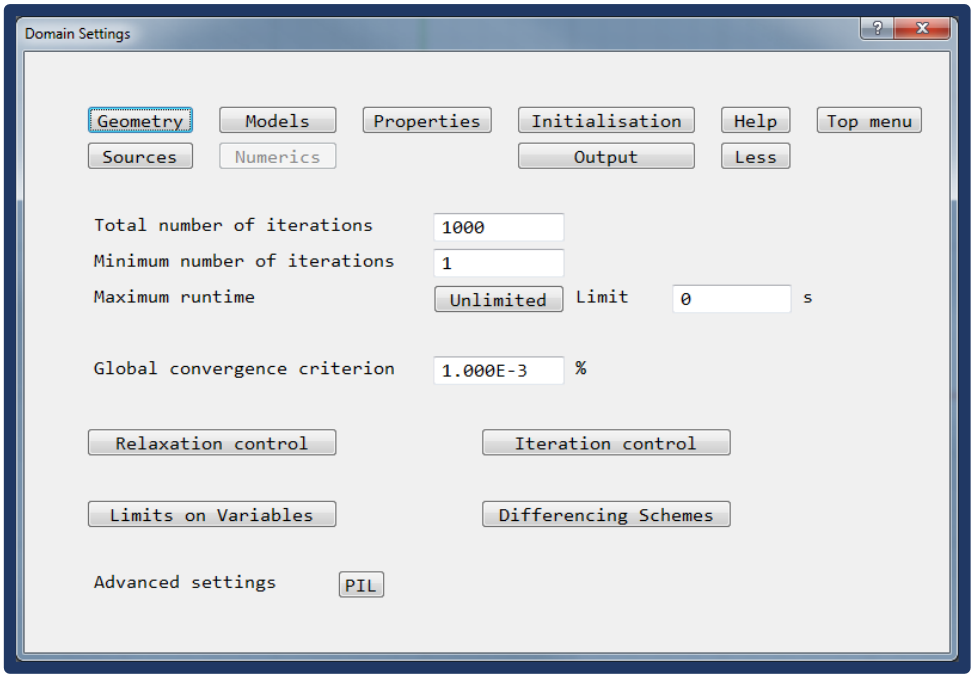

Figure 41: Numerics Menu (Transient Option Enabled)

14.6 Saving Time Step Results

In a steady solution, the results of interest are produced at the end of the solution, after the iteration has converged. Intermediate results have no physical meaning, and may only be interesting insofar as they may be helpful in indicating convergence.

In a transient simulation, the development of the flow with time is likely to be of interest, and to obtain this from the solution it is necessary to dump intermediate results, i.e. to save them into your working directory.

To do this, enter the main menu (left-click ) and click on 'Output' and then 'Field dumping'. In "Step frequency" enter the interval (in time steps) at which you wish dumps to be produced. Bear in mind that the files produced may be large, and a large number of them can require a very large amount of disc space. 'Start letter for solution file name' specifies a prefix letter for the intermediate solution files. This can help keep track of the files produced from different runs.

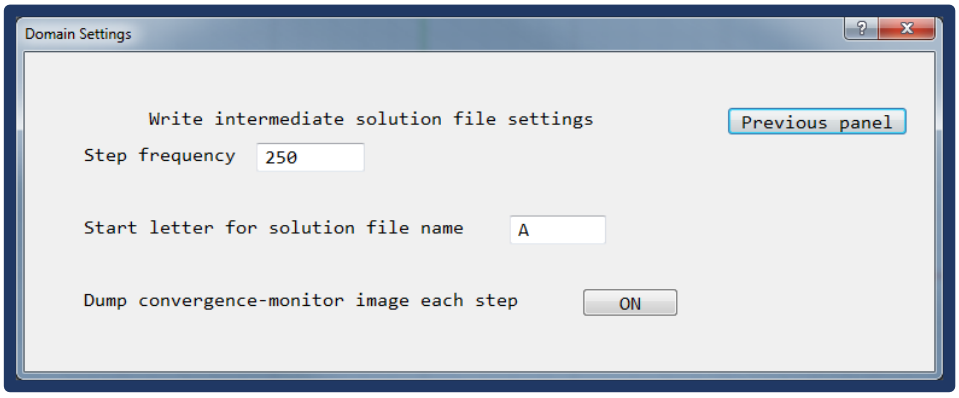

Figure 42: Dump Settings

In order to organise the transient results, it is recommended that after each simulation the results are saved to a separate folder. This is done by left-clicking on the second-to-last toolbar icon  . These results can then be loaded into Rhino later, and viewed by right-clicking on the same icon and navigating to the desired folder.

14.7 Convergence and Relaxation for Transient Runs

The transient term in the equations has a stabilising effect akin to relaxation. This may be understood intuitively by realising that in a short time step, nothing much can happen, and so there is not much work for the solver to do, and convergence should be straightforward. Provided that the time steps are fairly short, no additional relaxation may be necessary. If you want to provide some relaxation, we would suggest setting FALSDT for the solved variables (but not the pressure P1) to a value similar to the time step.

It is important that each step be adequately converged, otherwise errors will propagate through the run. Sufficiently small time steps should generally give good convergence; often it is easier to converge transient runs than steady runs. Reducing the time step size will generally require less sweeps to converge.

14.8 Monitoring the Time Variation at a Point

"Point_History" objects may be used to mark locations at which time-histories are required for any of the variables.

15 Viewing Results

15.1 Loading Visual Results

After completing a simulation results can be viewed by clicking on the 'load results' icon  on the toolbar. The latest results will load. While in post-processing mode all geometry is locked, but if necessary can be unlocked with Rhino's Unlock command.

15.2 Visualisation Probes

There is a range of visualisation probes to choose from which can be selected from the 'add probes' drop down icon. All probes can be moved and rotated using the gumball like normal Rhino3D objects.

A probe specifies a location from where results data will be filtered to produce a visual representation of parts of the solution which are of interest. The selected probe can be moved around and placed at different points to investigate further.

15.2.1 Cutting Planes

A 2D plane that spans the length of the domain can be used to display scalars in the projected plane. The values can be exported to a .csv file by selecting 'Export Cutting Plane Values' on the Results Panel.

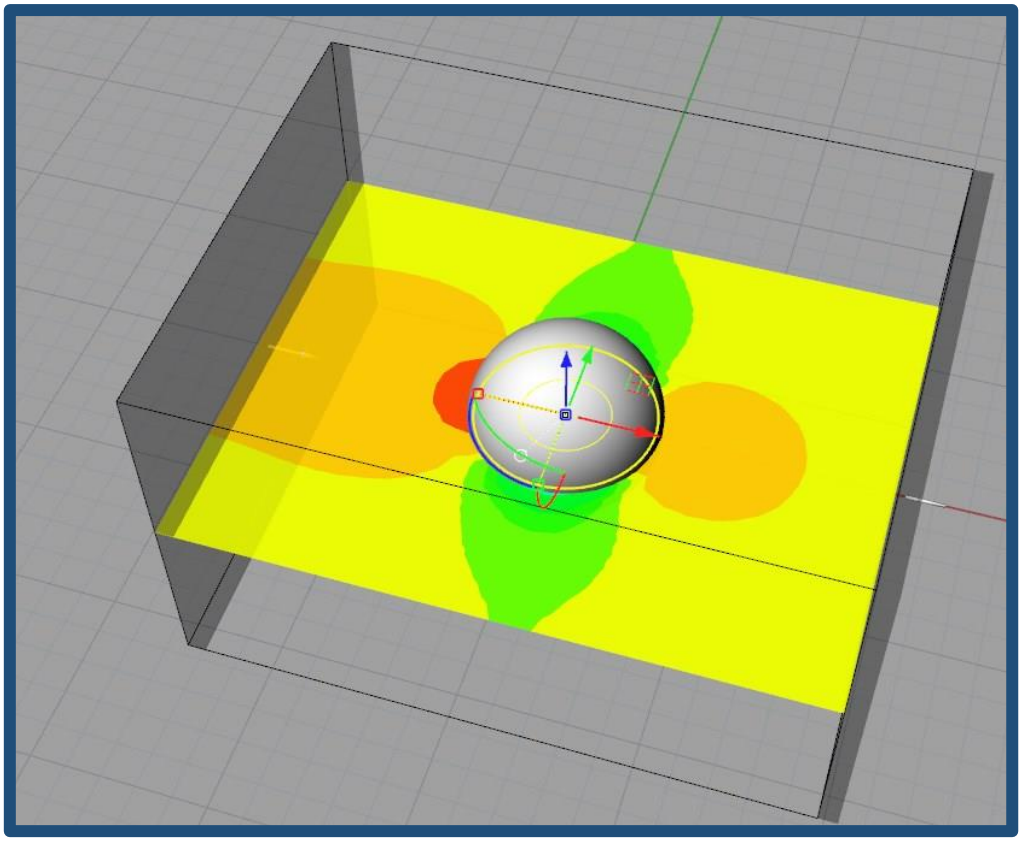

Figure 43: Cut Plane

15.2.2 Iso-Surface Probes

This probe creates a surface that interlinks all cells with the same proscribed value.

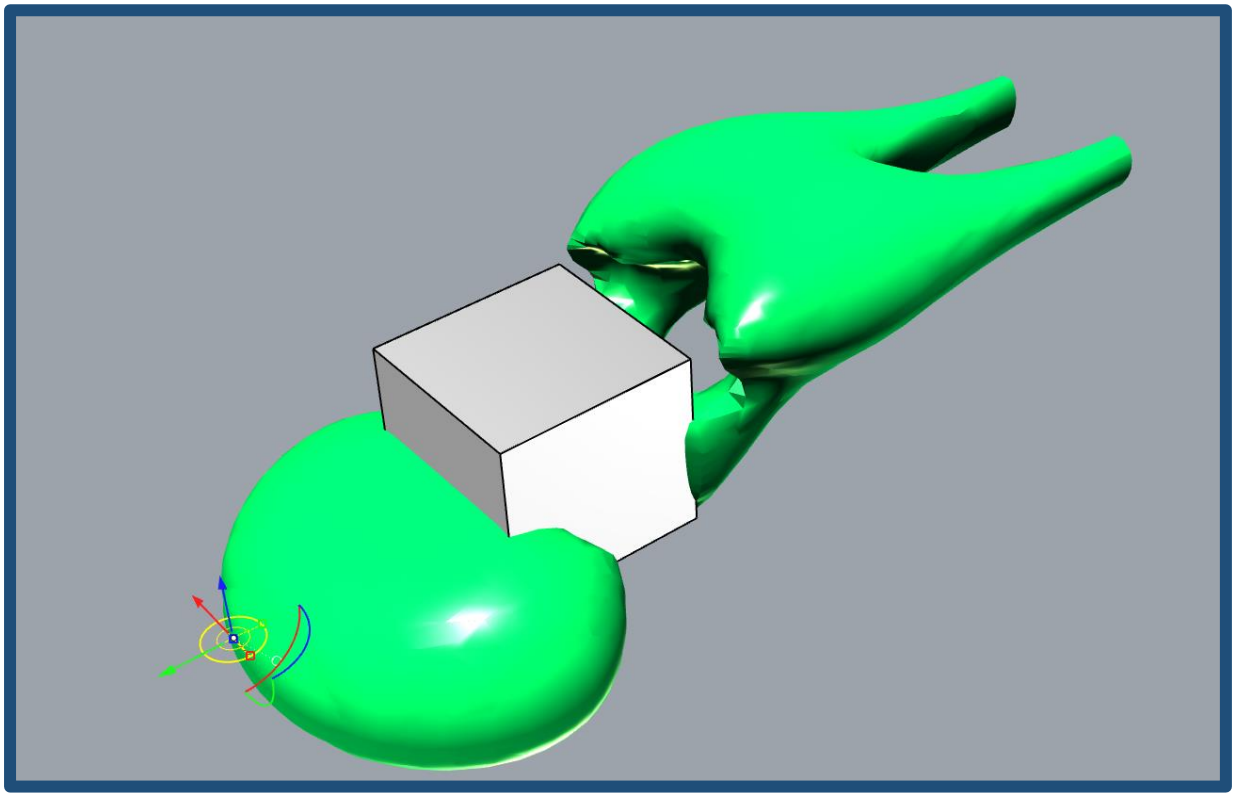

Figure 44: Iso-Surface

15.2.3 Streamline Probes

The streamline probe creates a series of lines simulating the flow path a given particle at that location will take.

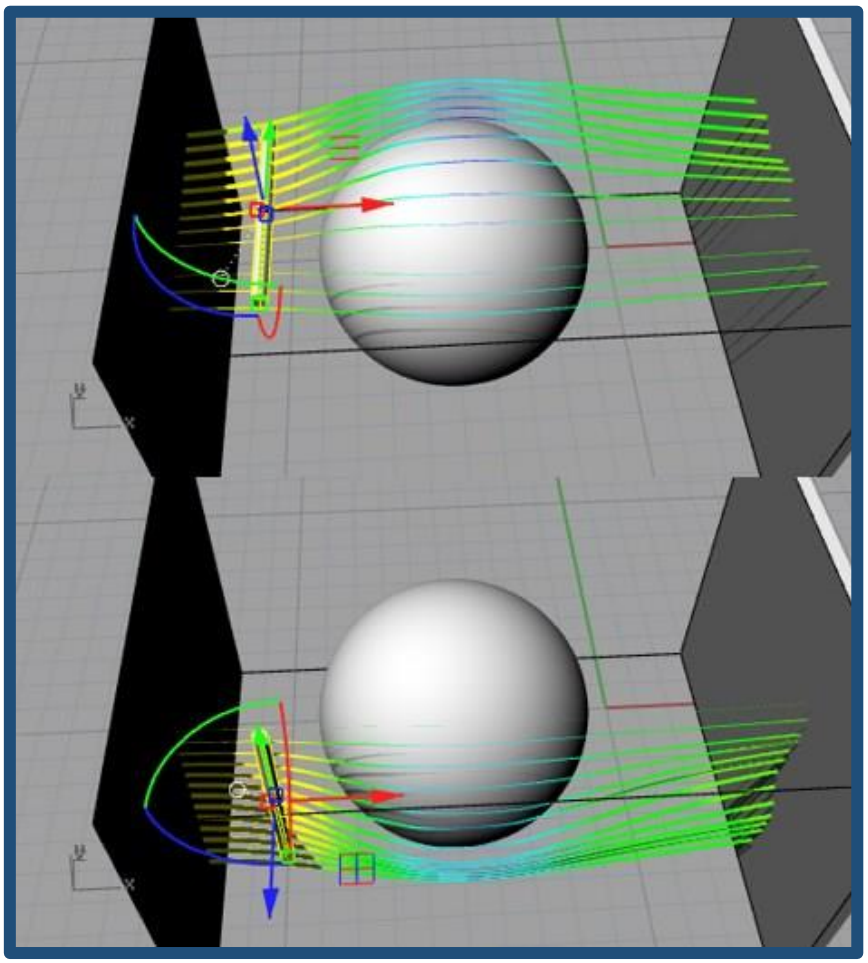

Figure 45: Stream lines

15.2.4 Point Probes

The point probe is used to display the value the selected variables at a given location inside the domain.

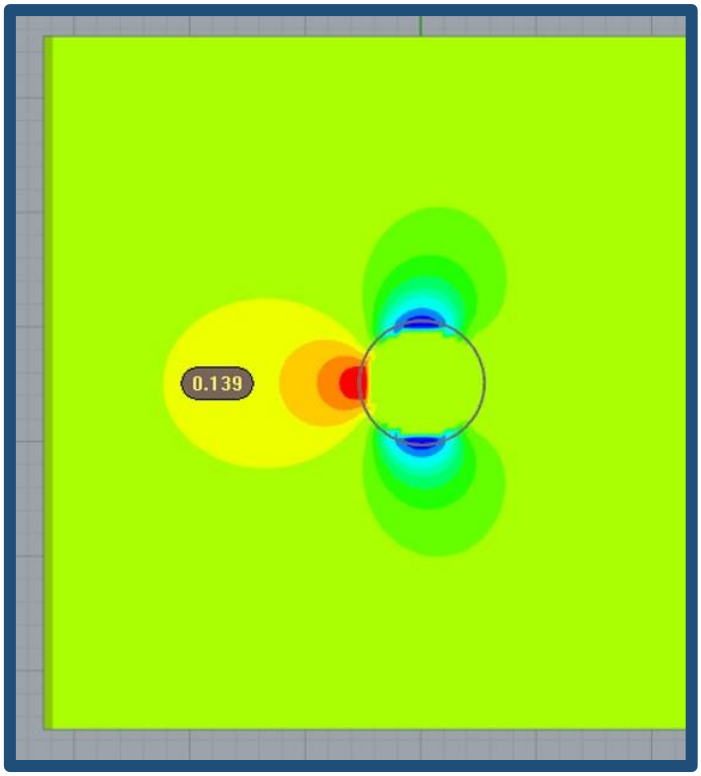

Figure 46: Point Probe

15.2.5 Surface Contour

Any calculated variable can be projected onto the surface of a blockage. To create a surface contour, click on the ‘surface contour’ drop down and then select any blockage. Please note that the values that are represented are those of the cells nearest to the object in question.

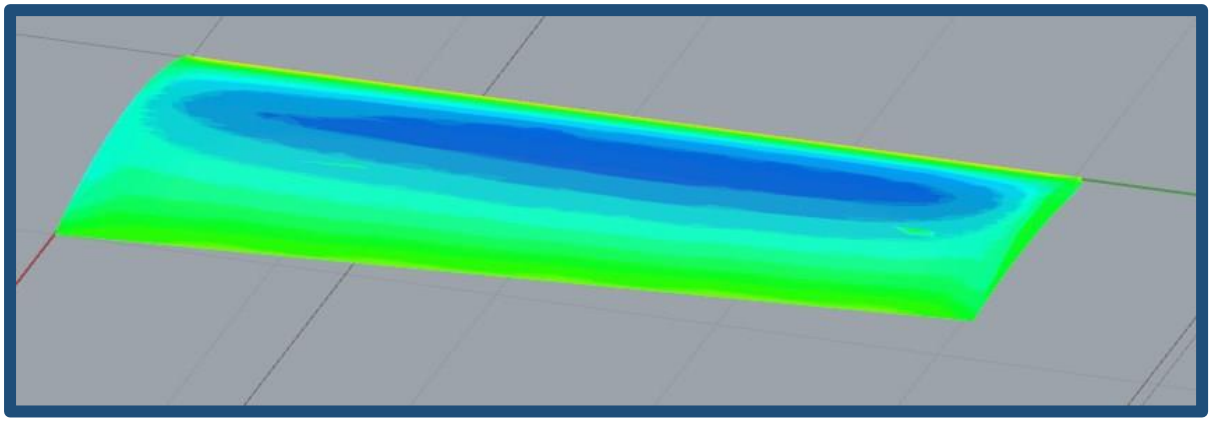

Figure 47: Surface Contour on an Aerofoil

15.2.6 Line Graph

A line probe that will provide the value of the selected scalar at multiple points. Placing this probe along an interesting point in the domain and then pressing ‘save graphics’ will produce a CSV file in the working directory.

15.3 Results Panel:

Probes are controlled by the results panel which can be hidden or shown via the  icon. The Figures 48, 49, 50, 51 and 52 explain the post processing features available for each type of probe.

15.3.1 Cut Planes

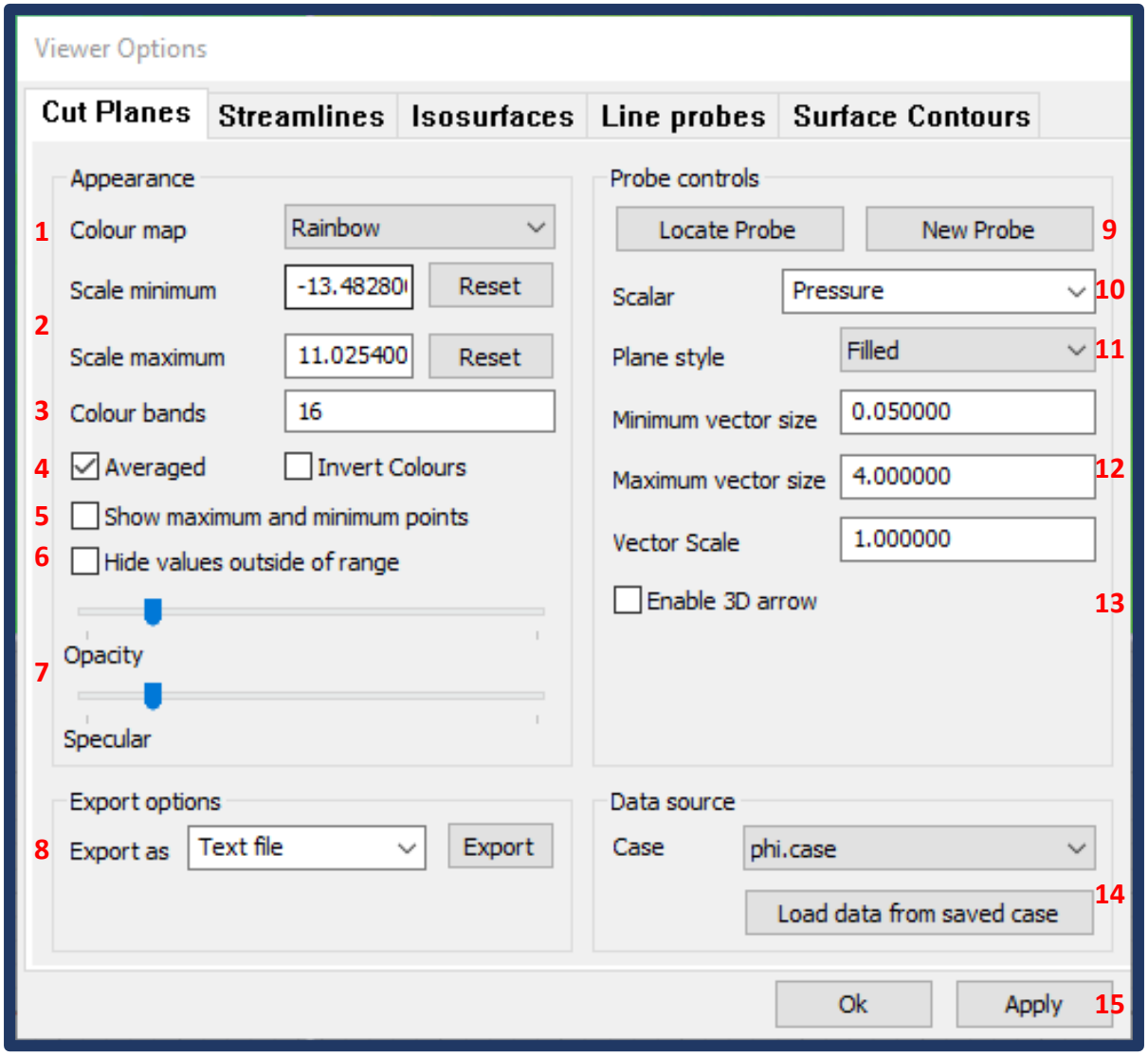

Figure 48: Cut Plane Results Panel

1. Allows selection of colour maps for cutplane
2. Sets the maximum and minimum ranges on the scale
3. Sets the number of colour bands
4. Displays averaged out results or exact value in each cell (default is averaged) and allows to invert/flip the colour map
5. Creates probes that display the points where maximum and minimum values are observed
6. Hides values outside of the range specified in **2**
7. Sliders to control the opacity and specular reflection of data display
8. Export options for data: can be exported as text (.csv) and as Rhino objects (.ply that can copy pasted into other rhino instances)
9. Highlights the location of the cut plane control probe; creates a new cutplane – multiple cut planes can be created
10. Drop down list to select what variable to display

11. Selection of how cutplanes are drawn: plane, lines, vectors or any combination
12. Options to adjust vector sizing and scale
13. Legacy option for larger vectors – rendering can be slow with these
14. Load data from previous runs – if saved as case
15. Accept and exit the Viewer Options or apply changes

15.3.2 Streamlines

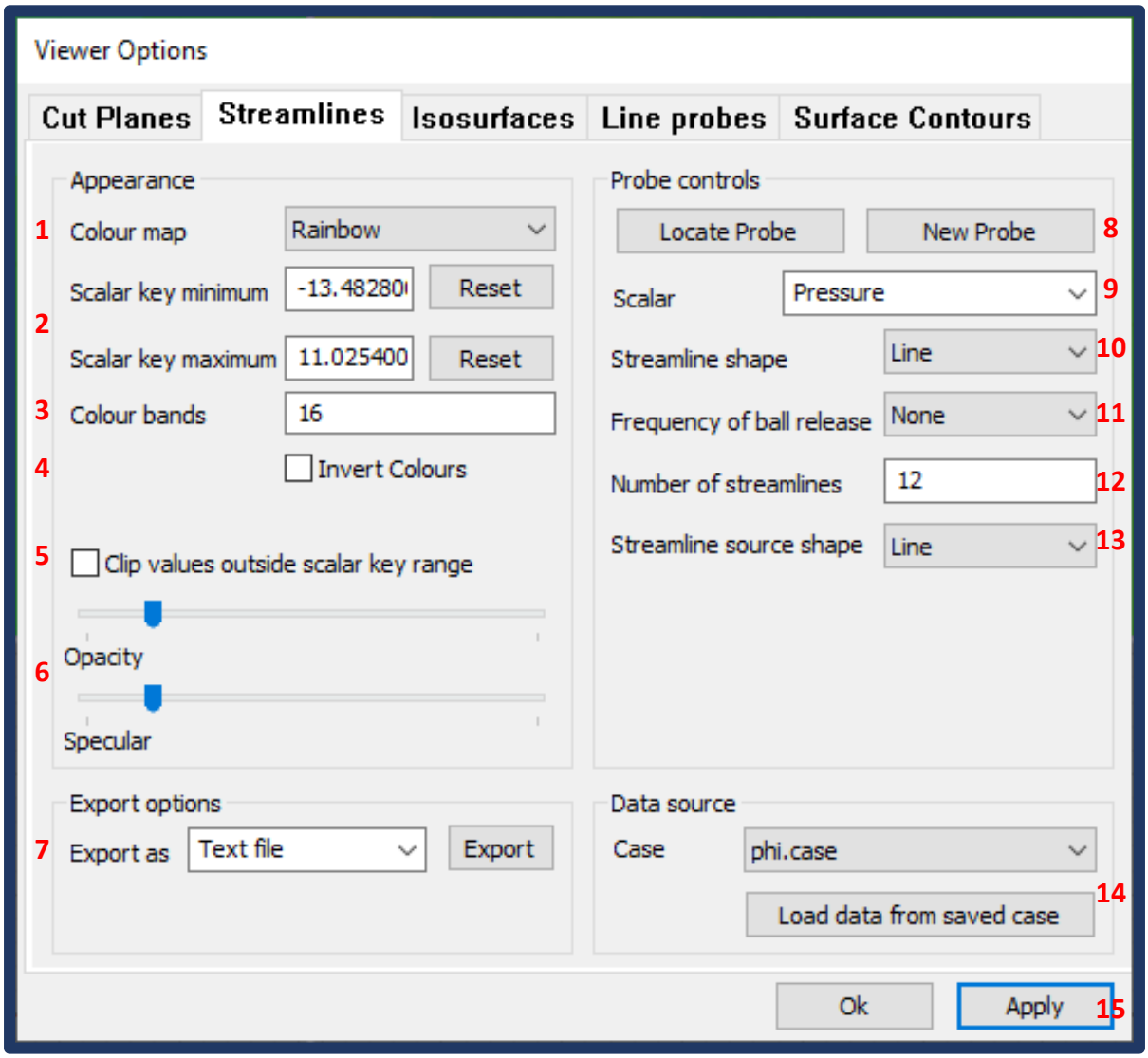

Figure 49: Streamline Results Panel

1. Allows selection of colour maps for streamlines
2. Sets the maximum and minimum ranges on the scale
3. Sets the number of colour bands
4. Allows to invert/flip the colour map
5. Hides values outside of the range specified in **2**
6. Sliders to control the opacity and specular reflection of data display
7. Export options for data: can be exported as text (.csv) and as Rhino objects (.ply that can copy pasted into other rhino instances)
8. Highlights the location of the streamline control probe; creates a new streamline – multiple streamlines can be created
9. Drop down list to select what variable to display
10. Drop down list to select what shape each streamline takes: line, ribbon and tube
11. Animates the streamline
12. Sets the number of streamlines drawn
13. Allows to swap the streamline source as line or a rectangular area
14. Load data from previous runs – if saved as case
15. Accept and exit the Viewer Options or apply changes

15.3.3 Isosurfaces

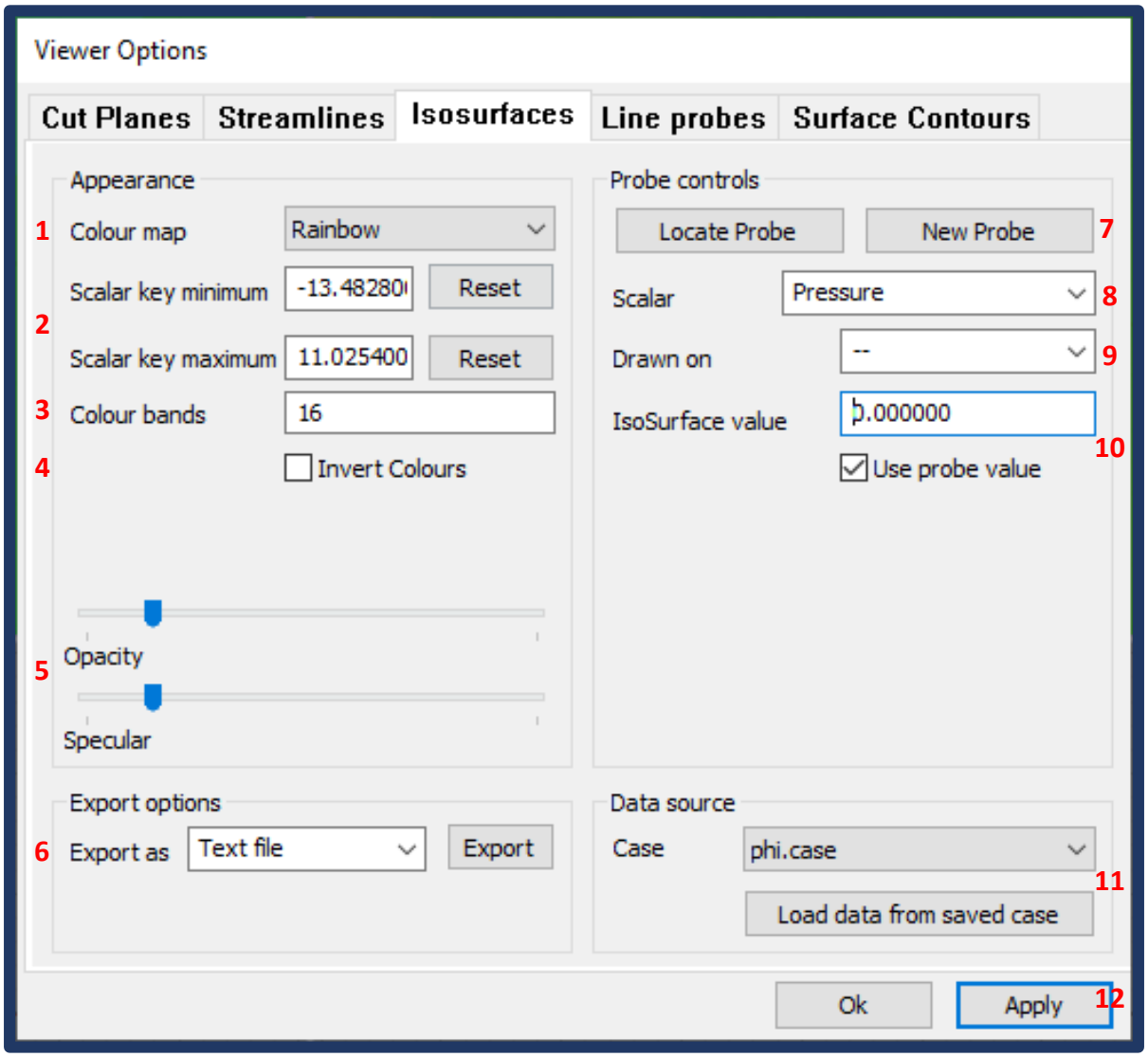

Figure 50: Iso-surface Results Panel

1. Allows selection of colour maps
2. Sets the maximum and minimum ranges on the scale
3. Sets the number of colour bands
4. Allows to invert/flip the colour map
5. Sliders to control the opacity and specular reflection of data display
6. Export options for data: can be exported as text (.csv) and as Rhino objects (.ply that can copy pasted into other rhino instances)
7. Highlights the location of the control probe; creates a new iso-surface control probe – multiple iso-surfaces can be created if using probe value
8. Drop down list to select what variable to display
9. By default it will draw the selected scalar on itself – but should the user chose to draw eg: velocity on water-air boundary they should set Scalar:Velocity and Drawn on:SURN with Iso-Surface value of 0.5 and untick Use probe value
10. Iso-surface value can be specified here or drawn based on the value at probe location
11. Load data from previous runs – if saved as case
12. Accept and exit the Viewer Options or apply changes

15.3.4 Line probes

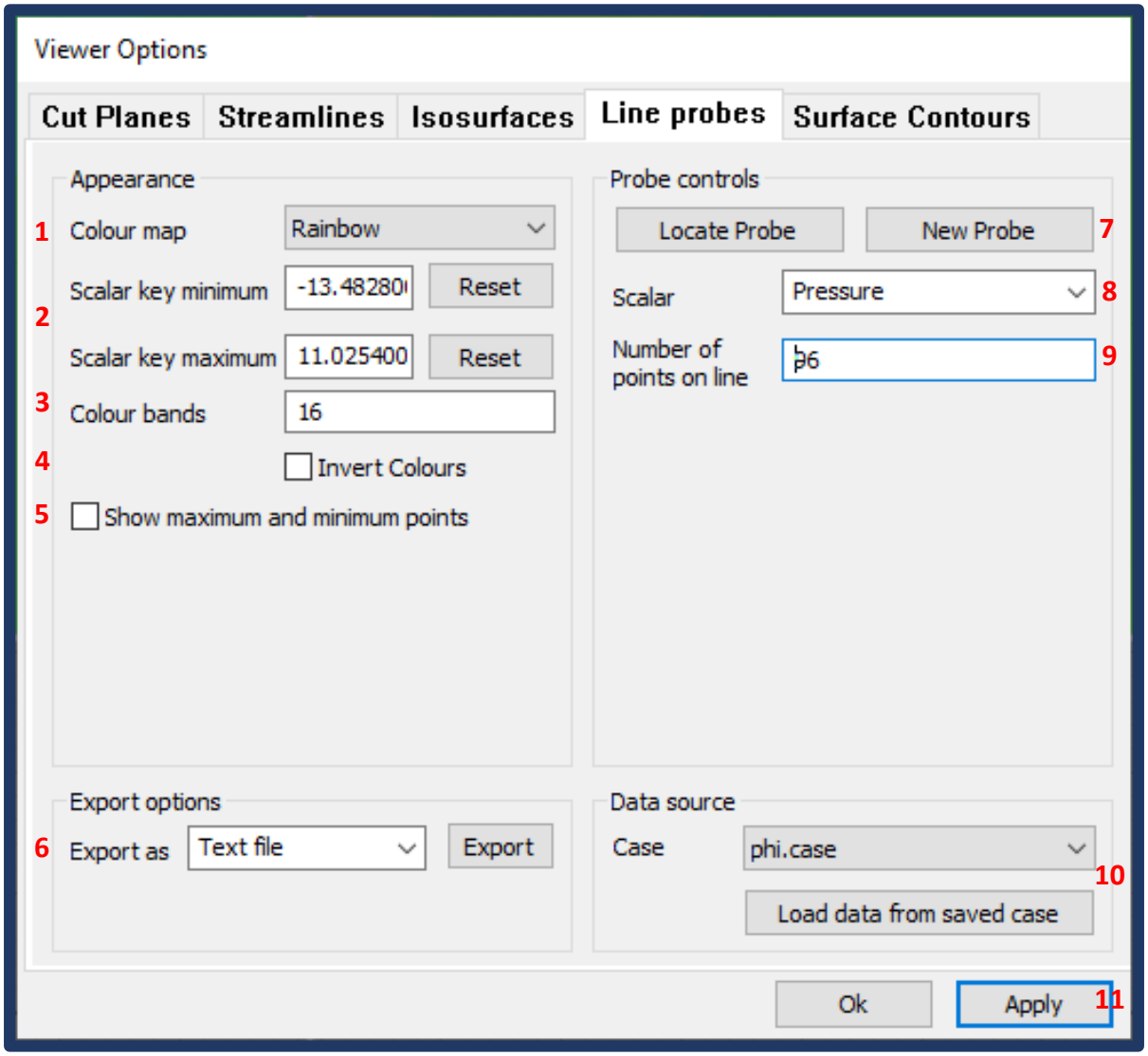

Figure 51: Line Probes Results Panel

1. Allows selection of colour maps
2. Sets the maximum and minimum ranges on the scale
3. Sets the number of colour bands
4. Allows to invert/flip the colour map
5. Creates probes that display the points where maximum and minimum values are observed
6. Export options for data: can be exported as text (.csv) and as Rhino objects (.ply that can copy pasted into other rhino instances)
7. Highlights the location of the line control probe; creates a new line – multiple lines can be created
8. Drop down list to select what variable to display
9. Sets the number of data points on the line – this number should be bigger then the number of cells the line spans
10. Load data from previous runs – if saved as case
11. Accept and exit the Viewer Options or apply changes

15.3.5 Surface Contours

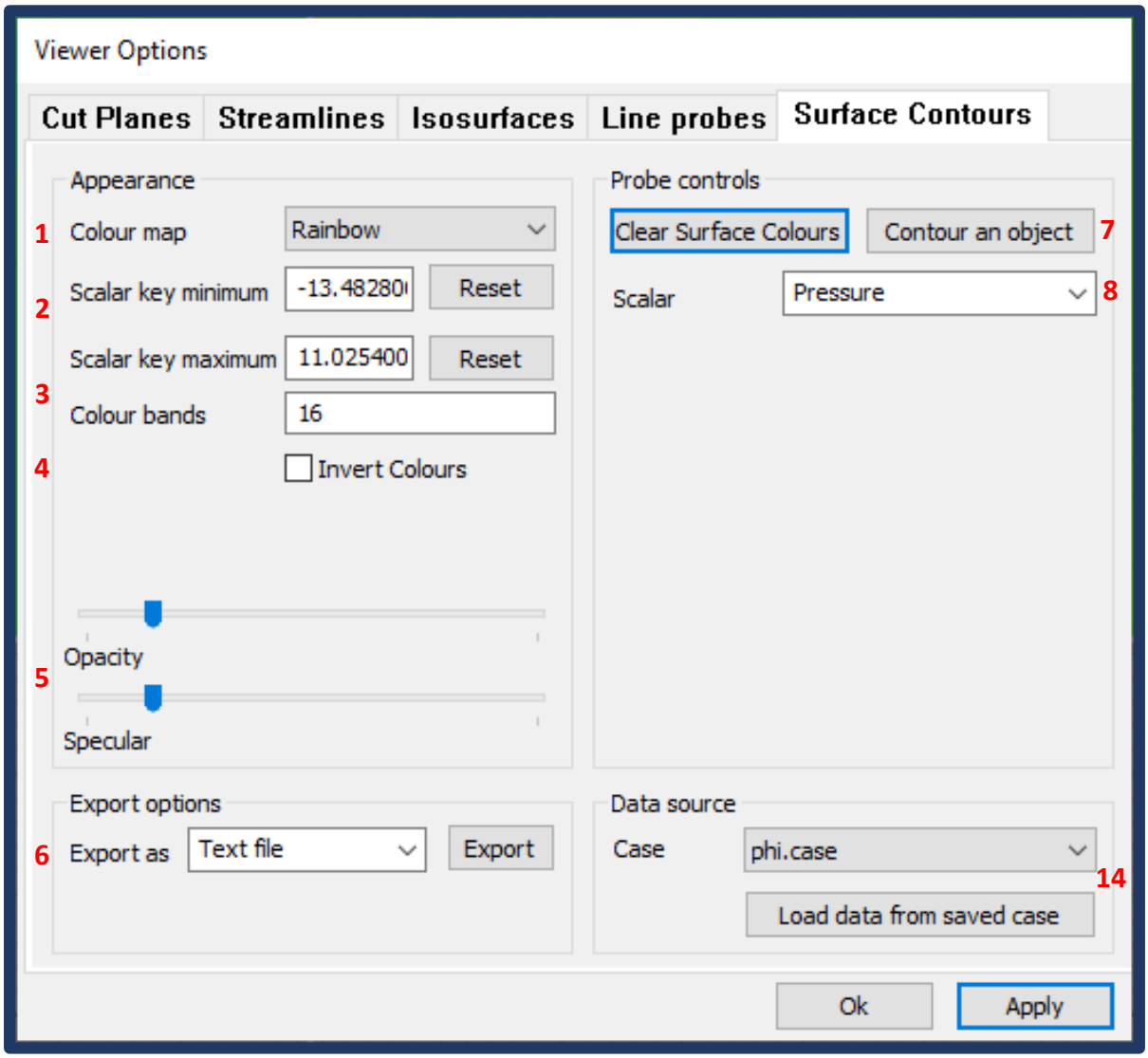

Figure 52: Surface Contour Results Panel

1. Allows selection of colour maps
2. Sets the maximum and minimum ranges on the scale
3. Sets the number of colour bands
4. Allows to invert/flip the colour map
5. Sliders to control the opacity and specular reflection of data display
6. Export options for data: can be exported as text (.csv) and as Rhino objects (.ply that can copy pasted into other rhino instances)
7. Highlights the location of the control probe; creates a new surface-contour – multiple surfaces can be coloured
8. Drop down list to select what variable to display
9. Load data from previous runs – if saved as case
10. Accept and exit the Viewer Options or apply changes

*Surface contours on objects with poor grid can produce erroneous outputs

15.4 Variables

The list of variables to be selected shows the full names of the variables. These are later replaced by internal variable codes, as shown in Appendix B.

15.5 Transient Results

Transient and steady simulations can be viewed in the same way. Each time step is contained in a 'CadRes' VTK file (see Section 15.3.1) the current time step (or CadRes file) is shown in the results panel and at the bottom of the scalar key. To view other time steps use the icon in the RhinoCFD toolbar.

Figure 53: Transient Toolbar Tools

Each step is contained in a separate CadRes VTK file. To scroll through the time steps, left click on the toolbar. To play through all results, click the 'play time series' icon in the tool bar or in the drop down.

16 The Scalar Key

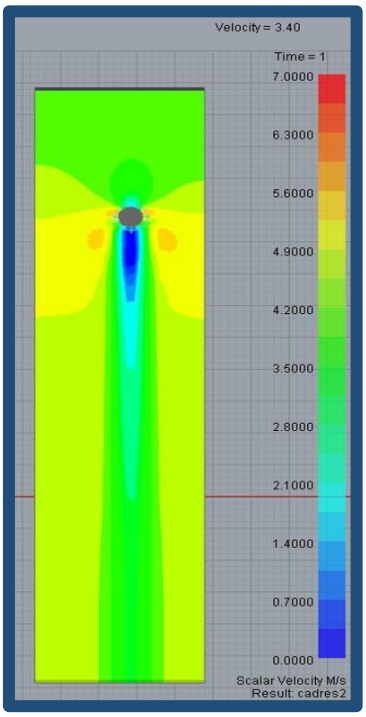

Figure 54: Velocity cut-plane with scalar key

Left-clicking the scalar key button  displays a numbered colour key for contours or vectors plotted in the Rhino environment. At the bottom of the key the variable, units and the name of the VTK solution file are shown; at the top, the variable value at current probe location, and the current time, are displayed. The scalar key button is a toggle, so clicking on it a second time will remove the key.

17 RhinoCFD Files

17.1 Rhino3D Files

RhinoCFD models are saved using the same file type as normal Rhino3D models: .3dm. The files contain both the usual Rhino model information and the CFD information. These files are used for opening Rhino3D and RhinoCFD cases. For more information see the Rhino3D user guide.

17.2 CFD Input File (Q1)

The Q1 file is a text file created by RhinoCFD which contains all the pertinent CFD information regarding both the geometry and the input parameters for the simulation. It is created automatically by RhinoCFD and can be viewed by left-clicking the  button. A detailed explanation of the commands in the Q1 file can be found here.

17.3 Output Files

17.3.1 Solution File

The solution file contains the 3D store of values for each variable at each cell. It is used to create the visual graphics (contour or vector plots) which are displayed when results are loaded in Rhino; or to provide the data for a restart run. Solution files have the suffix .vtk, and are created by RhinoCFD at the end of a simulation or if so specified at intermediate sweeps. The solution file phi.vtk is created at the end of a run.

The file phi.vtk is used by RhinoCFD to create a 'CadRes' file which saves the results for the current Rhino session when 'load results' button  is selected. CadRes files are created each time this button is selected, and are numbered from 0 onwards. If RhinoCFD is closed and re-opened in the same working directory, the CadRes numbering will then start from 0 again.

For more information on VTK files click here.

17.3.2 Log Result File

In addition to the solution file, RhinoCFD produces a text file called "Result" which serves as a log of the simulation that has been run, including complete information about settings used. It also contains the "Nett Source" report, which shows the source of every variable at every object. "Nett Source" data are important for two reasons: (1) to provide a check that all the specified sources are indeed present; (2) it is easy to check that the sums of sources and sinks balance, which is a fundamental way of checking convergence. See Section 13.1.2 for more details. To view this file right-click on  .

17.4 Transferring Files to New Locations

In order to view any RhinoCFD simulation in a different location or on a different computer, then the .3dm and phi.vtk (there may be several if transient) files must both be copied. This enables both the model and results to be viewed.

Appendices

A. Object Types

Object type	Brief Description
Blockage	Specifies solid blockages / Applies heat and momentum sources in solid or fluid.
Inlet	Fixed mass source
Outlet	Fixed pressure
Wind	(Whole domain) Applies wind profiles at domain boundaries
Fan	Fixed velocity
Angled-in	Fixed mass source on surface of underlying BLOCKAGE object
Angled-out	Fixed pressure on surface of underlying BLOCKAGE object
Pressure Relief	Fixes reference pressure when steady and no Outlets
Point History	Single cell monitoring point for transient
Sun	(Whole domain) Applies solar radiation heat load within domain
Foliage	Represents frictional effect of vegetation
User Defined	For setting user-defined sources (Advanced)
Null	Defines regions for mesh control, no effect on solution

B. Variables

Variable	Definition	Typical Initial Value
P1	Pressure	“Ambient”
U1	X Velocity	0
V1	Y Velocity	0
W1	Z Velocity	0
KE	Turbulent Kinetic Energy	0.001
EP	Turbulent Dissipation Rate	0.001
C1	Scalar Tracker	0
TEM1	Temperature	“Ambient”
T3	Radiation Temperature	“Ambient”
AGE	Age of Air	0
SMOK	Smoke Tracer	0
El1	Mixing Length-Scale	

ENUT	Laminar Kinematic Viscosity
EPKE	Ratio of Dissipation Rate to Turbulent Kinetic Energy
DEN1	Density
ENUT	Laminar Kinematic Viscosity
EMIS	Emissivity
WDIS	Distance from Wall
AEE	Air Exchange Efficiency
SPH1	Specific Heat

Note: The "Ambient" values used for initialising the pressure and temperature are set in the "Properties" section of the Main Menu (accessed by left-clicking ).

C. Properties Table for Solids

Index Ref	Material	Temperature ˚C	Density kg/m^3	Specific Heat Capacity J/kg˚C	Thermal Conductivity W/m˚C
100	100 ALUMINIUM	27	2700	896	204
101	101 BAKELITE	20	1270	1590	0.23
102	102 BRICK	20	1600	840	0.69
103	103 COPPER	27	8954	383	381
104	104 EPOXY	27	1900	789	0.4
105	105 FIBREGLASS	27	48	1100	0.038
106	106 GLASS	20	2700	840	0.78
107	107 GOLD	27	19290	129	305
108	108 NICKEL	27	8906	446	90
109	109 POLYSTYRENE	27	1057.2	1344	0.15
110	110 SILVER	27	10524	234	415
111	111 STEEL	27	7800	473	43
112	112 TIN	27	7304	226	66
113	113 Asbestos Cement Sheet		700	1050	0.36
114	114 Asbestos Cement Decking		1500	1050	0.36
115	115 Brickwork (Outer leaf)		1700	800	0.84

116	116 Brickwork (inner leaf)	1700	800	0.1
117	117 Cast concrete (dense)	2100	840	1.4
118	118 Cast concrete (medium)	2000	1000	1.13
119	119 Cast concrete (lightweight)	1200	1000	0.38
	Concrete block (heavyweight)	2300	1000	1.63
121	Concrete block (mediumweight)	1400	1000	0.51
122	Concrete block (lightweight)	600	1000	0.19
123	Aerated concrete slab	500	840	0.16
124	Fibreboard	300	1000	0.06
125	Plasterboard	950	840	0.16
126	Tile hanging/Roof tile	1900	800	0.84
127	External rendering / Plaster (dense)	1300	1000	0.5
128	Plaster (lightweight)	600	1000	0.16
129	Asphalt/Felt/Bitumen layers	1700	1000	0.5
130	Screed	1200	840	0.41
131	Stone chippings	1800	1000	0.96
132	Wood wool slab	500	1000	0.1
133	Timber flooring / Wood	650	1200	0.14
134	Expanded polystyrene slab	25	1400	0.035
135	Glass fibre slab	12	840	0.04
136	Glass fibre quilt	25	1000	0.035
137	Mineral fibre slab	30	1000	0.035
138	Phenolic foam	30	1400	0.04
139	Polyurethane Board	30	1400	0.025
140	Urea formaldehyde foam	10	1400	0.04