RhinoCFD Tutorial - Flow Past a Sphere

Document release date: April 2023

Software version: 2.1.5 January 2022

Solver version: PHOENICS 2021 v1.0

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

The tutorial describes a simple calculation of flow around a sphere and various methods of visualizing the results. It is assumed that the user is familiar with Rhino; if not, this useful introduction is helpful in getting you up and running.

2 Geometry

In a 'small objects – Meters' Rhino environment, create a sphere of any size using the sphere tool in Rhino.

3 CFD Analysis

3.1 Creating the Domain

Create the domain (fluid region) with the 'create domain to fit objects' button on the RhinoCFD toolbar:

Figure 1: Tool Bar

A file find dialog will then appear; use this to locate a directory where you wish to work preferably an empty directory to make the process clearer, as all the RhinoCFD intermediate and result les will be saved here.

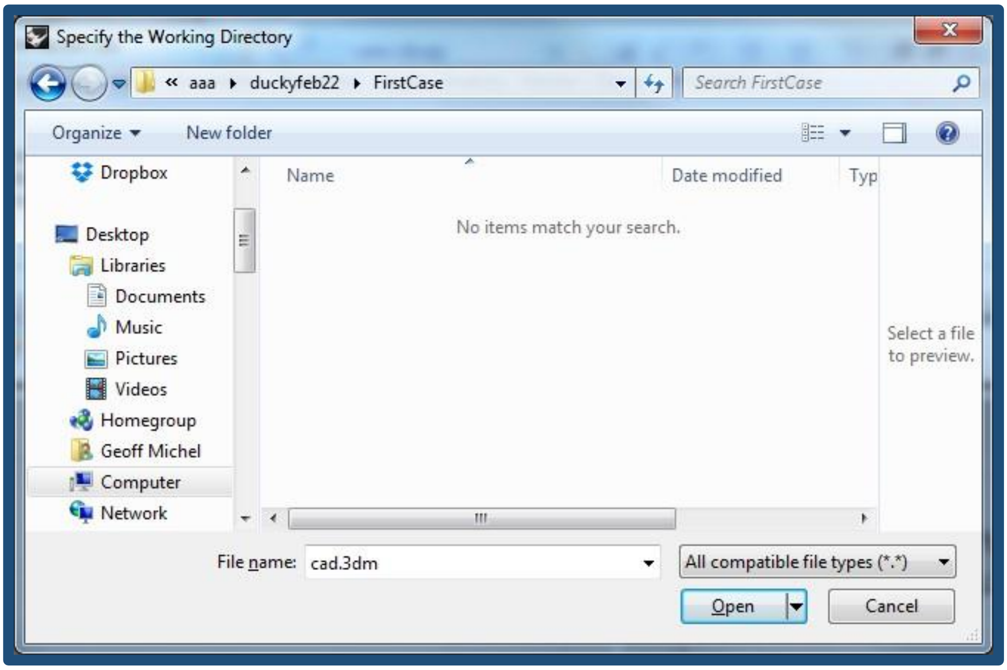

Figure 2: Working Directory Selection

The next choice is the type of problem to be solved; choose Core.

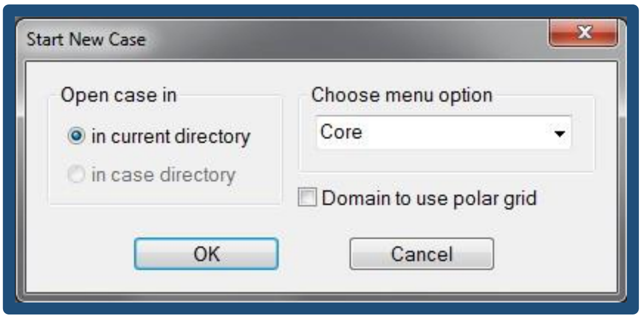

Figure 3: Type of Simulation Selection

Resize the domain by scaling or by using the Gumball tool - Type `Gumball On' if you cannot see the Gumball. Make the domain significantly larger than the sphere.

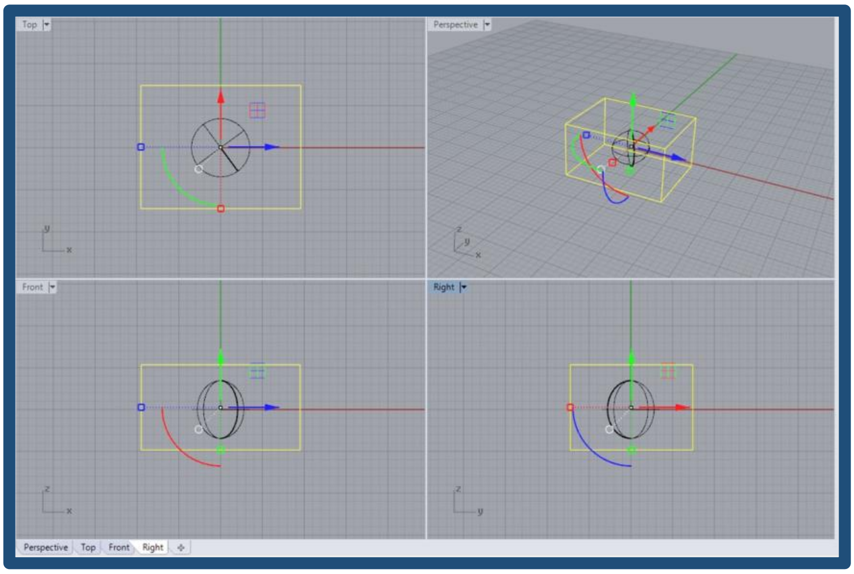

Figure 4: Domain and Sphere

3.2 Creating Fluid Boundaries

Now we need to set the inlet, outlet and other boundary conditions. In general, you must always add at least one inlet and outlet, or nothing interesting will happen. The easiest way to do this is via the Domain Settings. Right click the 'edit solution parameters' button in the RhinoCFD toolbar.

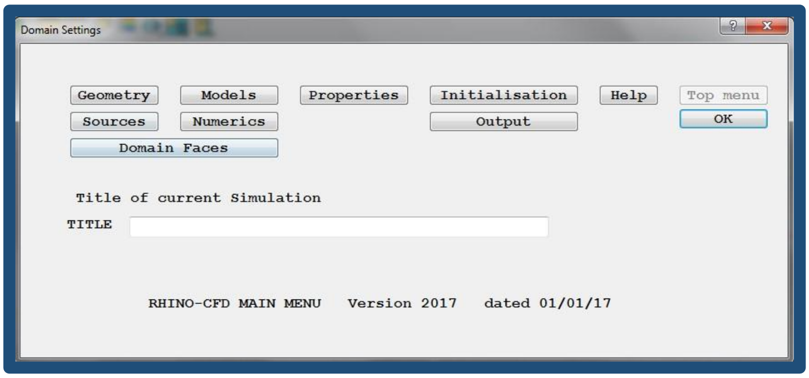

Figure 5: RhinoCFD Main Menu

Click on Domain Faces to add an inlet and outlet at low and high X locations of the domain. Click on settings and change the X velocity to 1m/s. Click OK and a reminder window will appear which you can also close by clicking OK.

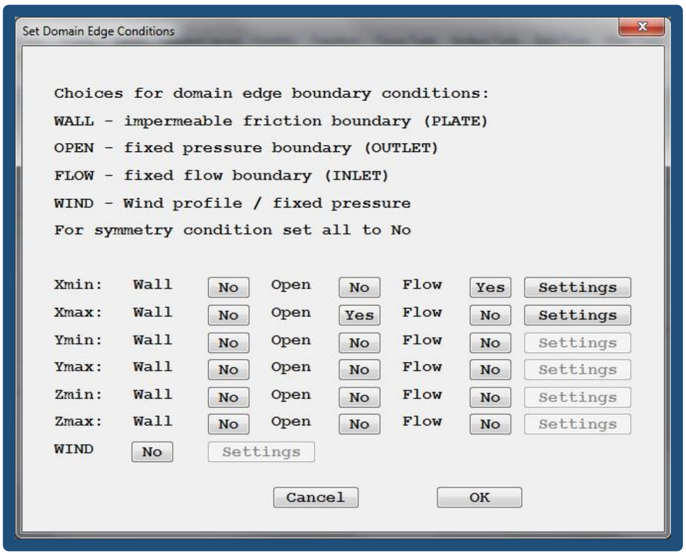

Figure 6: Domain Face Options

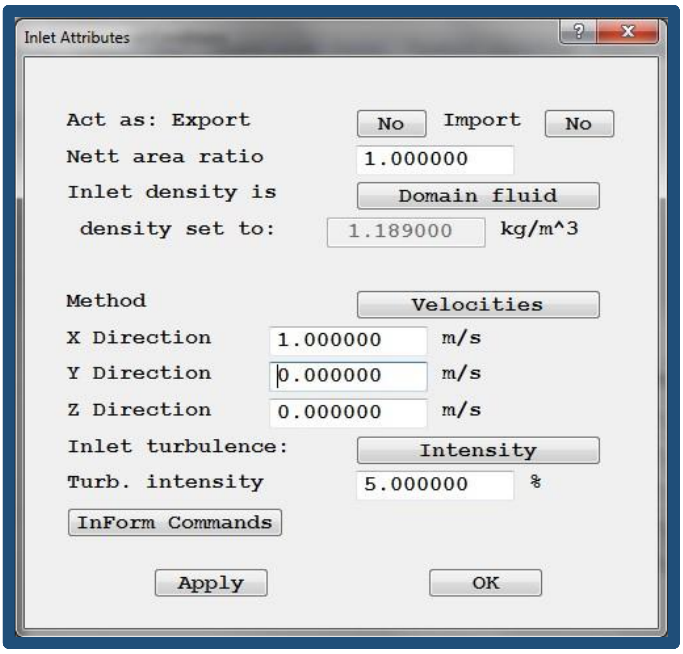

Figure 7: Edit Object Attributes

The CAD appearance will change to include two new objects (the inlet and outlet). The inlet and outlet properties can be modified ed by selecting the inlet or outlet and clicking the 'edit CFD properties' button in the RhinoCFD toolbar.

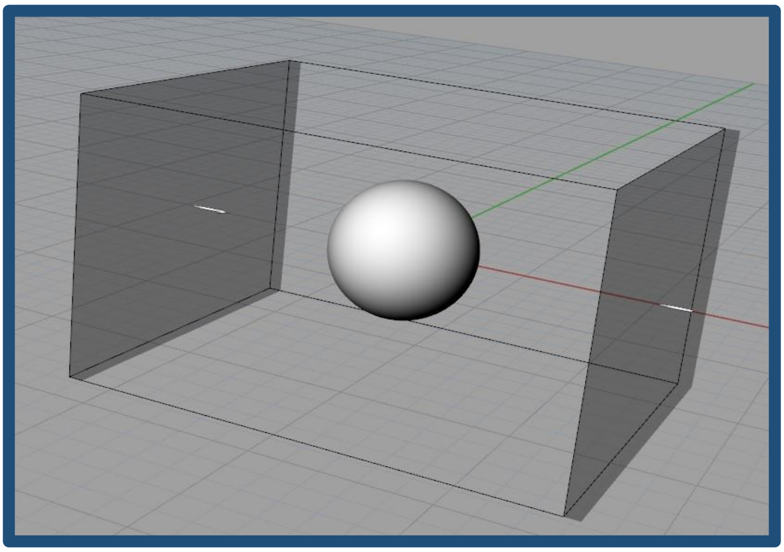

Figure 8: Sphere, Inlet and Outlet

3.3 Meshing

The next step is to ensure the mesh is adequate to resolve all the features in the simulation. Of prime concern is the sphere, which needs reasonable number of cells to de ne the curves. A secondary concern is to ensure the mesh "looks good", that is, that there aren't sharp differences in size between one cell to the next. More information on RhinoCFD grids can be found in our Meshing User Guide. To modify the mesh, click on the fourth button on the toolbar .

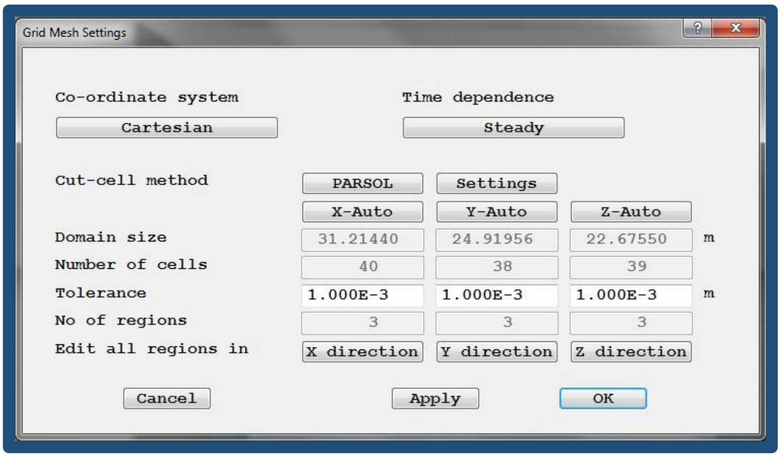

Figure 9: Geometry Dialog

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

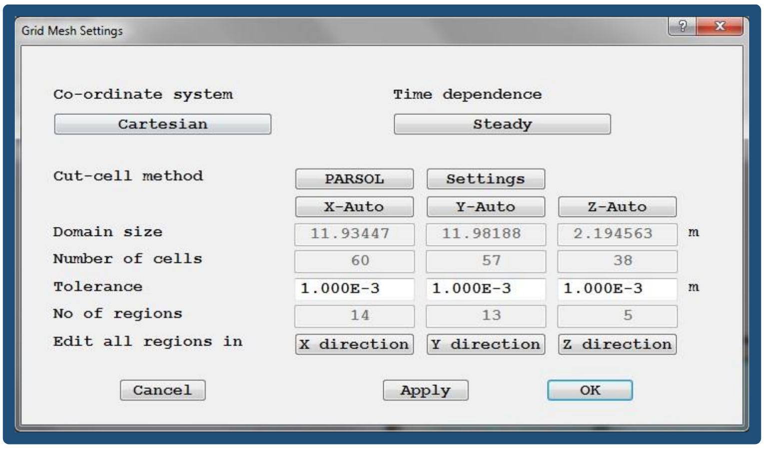

Figure 10: Mesh Around Sphere

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

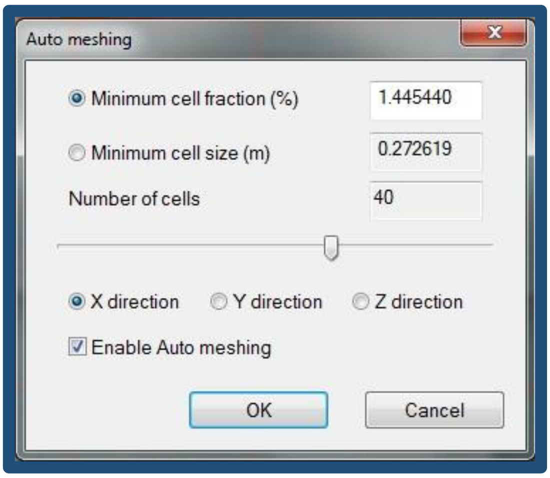

Figure 11: Automesh Dialog

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

3.4 Running the Simulation

Once the case set up is complete; run the solver with the 'run solver' icon in the RhinoCFD toolbar and select 'run' on the dialog that appears. The convergence monitor will then appear:

Figure 12: Convergence Monitor

Your simulation will have finished and achieved convergence when your monitor displays similar results to the following:

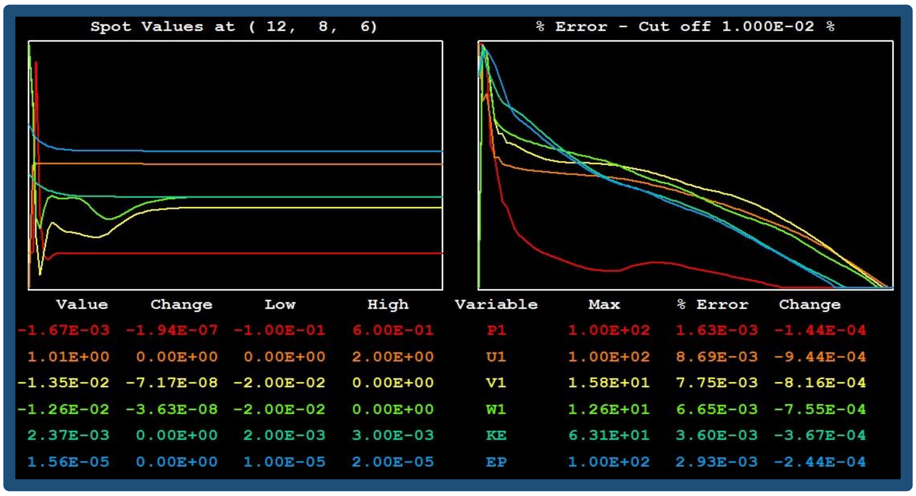

Figure 13: Converged Solution

Please see the User Manual for more information on convergence.

3.5 Results

Load the results using the 'load results' icon in the RhinoCFD toolbar. A dialog asking for the result le name will appear.

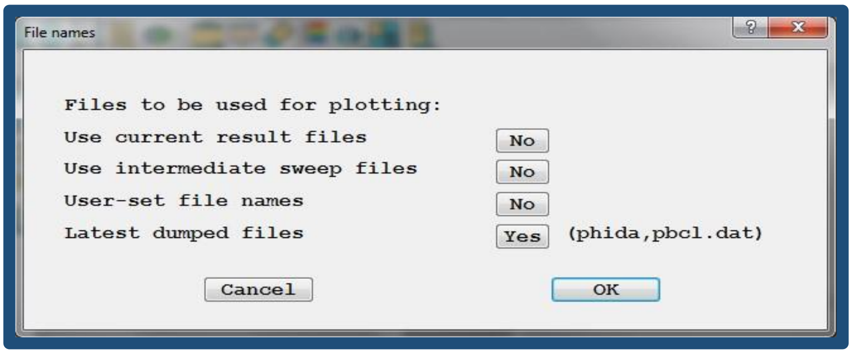

Figure 14: Load Results Options

Click OK to load the results from the latest dumped files. For a typical case you will create a range of contours to inspect the ow eld. The eld variable is initially set to P1 (Pressure). Figure 15 shows typical results:

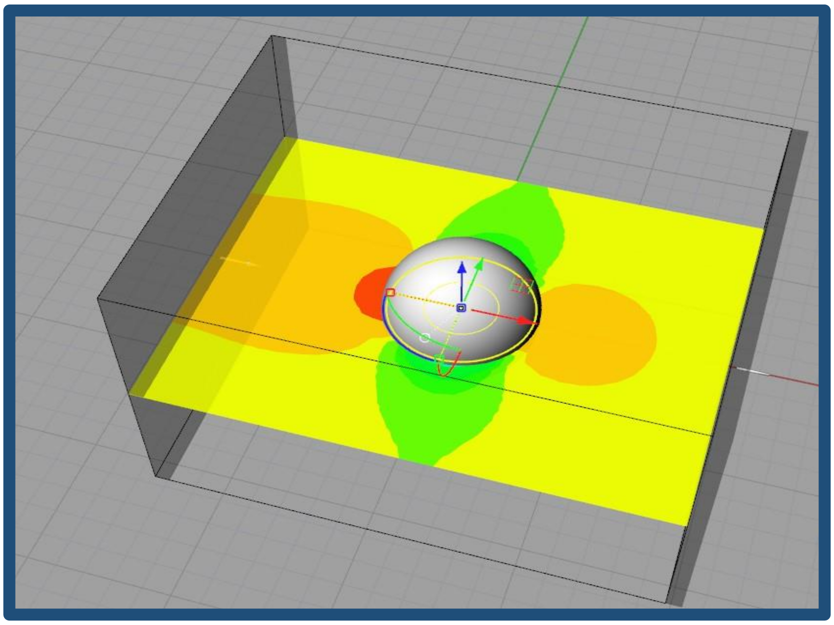

Figure 15: Pressure Contour in Z Plane

Try moving and rotating the cutting plane using the Gumball to inspect different in the domain.

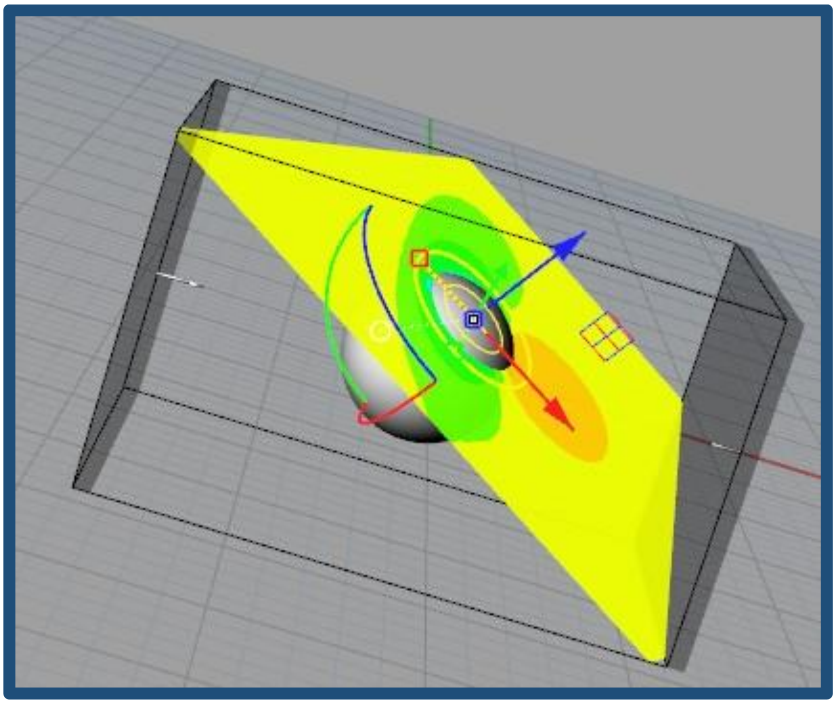

Figure 16: Pressure Contour and Shifted Cutting Plane

A selected cutting plane can be hidden by left clicking on 'hide current object (selected)' button in the RhinoCFD toolbar. Right clicking on the same button will bring all previously hidden objects back.

The viewer options menu/results panel can also be closed by clicking on 'OK'. It is then brought back by left clicking on 'edit display parameters'. The result properties menu contains all the settings to control selected cutting planes, streamlines, IsoSurface probes, line plots and contour plots. It is useful to note that if multiple probes are active in the Rhino environment, then any change to any tab of the results panel will change the settings of all probes, unless an individual plot probe has been selected first.

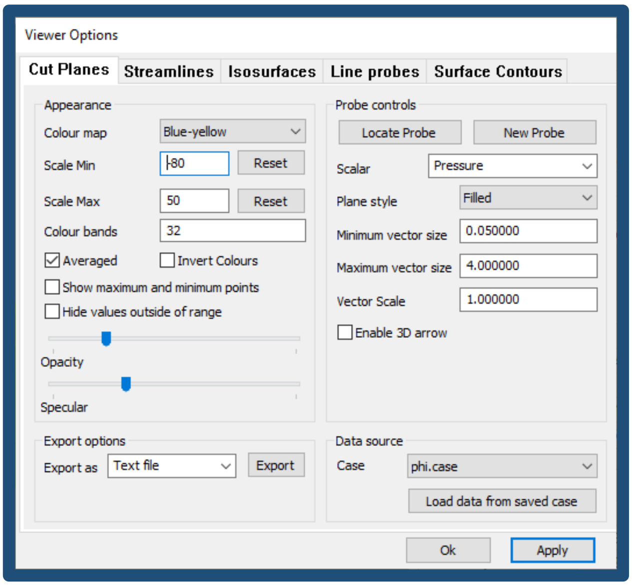

Figure 17: Cut Plane - Results Panel

The displayed variable can be changed from the drop down list at the top. There are also various styles of cutting planes that can be chosen, which can be used to display the results in a different way. Figure 18 shows the available options.

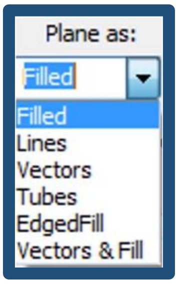

Figure 18: Plane Type Drop Down

Streamlines, IsoSurfaces, line probes and surface contours can be generated via the drop down 'Add probes from drop down' and choosing the desired probe. Click on the streamlines option to produce something similar to Figure 19. In this case the stream probe has been resized to stretch across the domain using the Gumball.

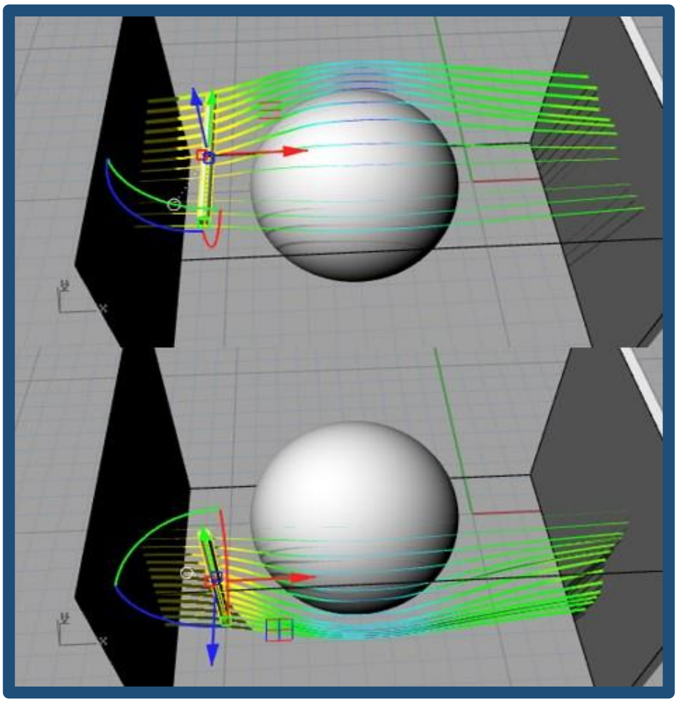

Figure 19: Streamlines

The streamline cylinder probe can be rotated to generate streams in other planes, as seen in the previous figure.

The streamline properties can be viewed and edited by clicking on the stream tab on the results panel:

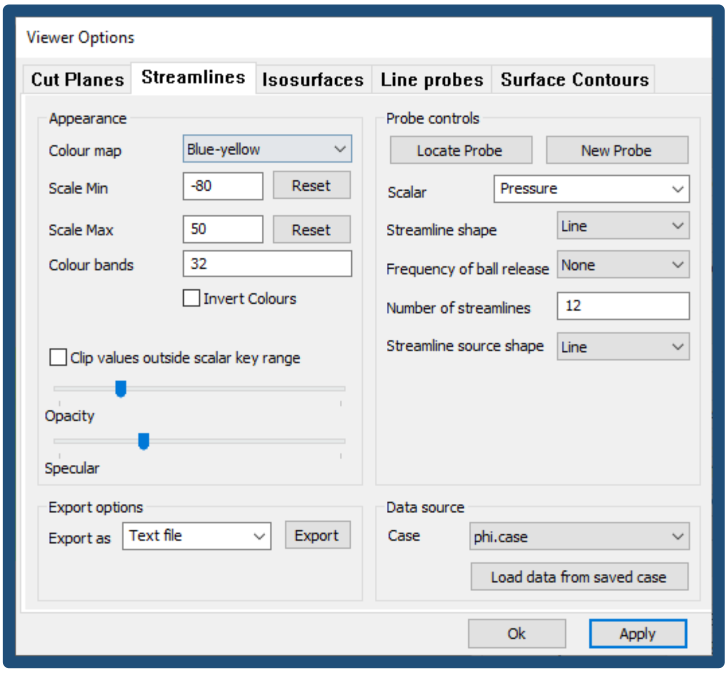

Figure 20: Streamlines Results Panel

This panel allows the number of streamlines, the type of stream line and the source type to be changed, as well as the other the same variable and range options seen in the cut plane tab.

Change the number of stream lines to 50 by editing the ‘No Streams’ input box and typing 50. This will produce a greater number of streams across the stream probe. Try changing the ‘Stream as’ to tubes and lines to see a difference in display.

Select ‘IsoSurface probe’ from the RhinoCFD toolbar and a probe will appear in the domain. To change the IsoSurface options, click on the IsoSurface tab on the results panel.

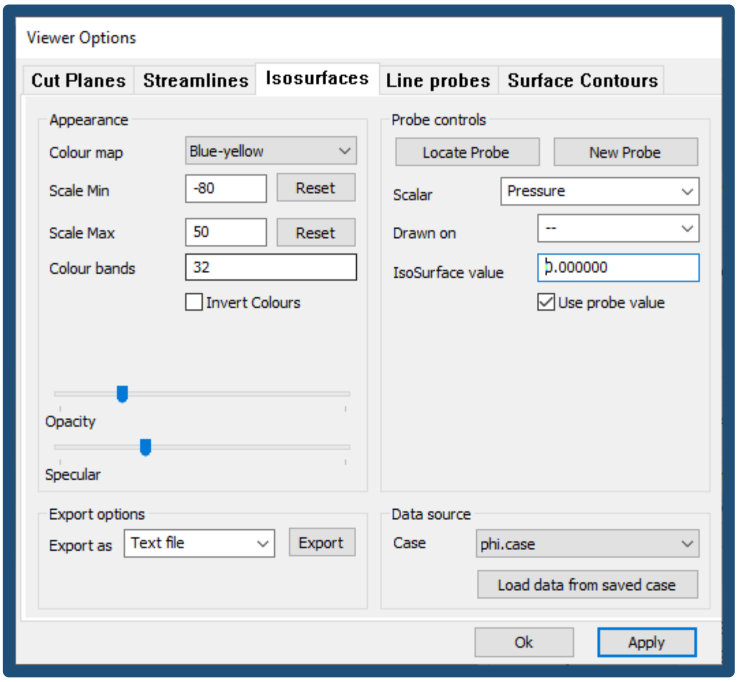

Figure 21: IsoSurface Results Panel

On the third row, click in the tick box to use a user de ned value to colour and locate the IsoSurface. If this box is left ticked, RhinoCFD will use the value the ‘mesh probe’ is reading at its current location, and the IsoSurface value is changed by moving the probe. In the box labelled IsoSurface, edit the value to be in between the max and min values of the selected variable. The IsoSurface will then change to show regions of the specified value. Figure 22 shows typical results.

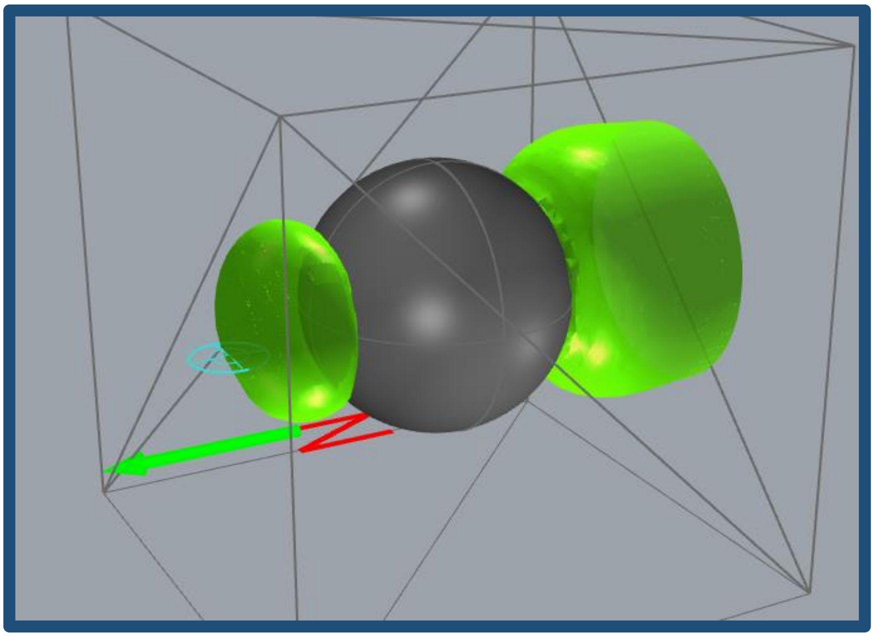

Figure 22: IsoSurface

To create a surface contour, click on the 'select and contour blockage' in the probe drop down list and then select the sphere. This should produce something similar to the following figure:

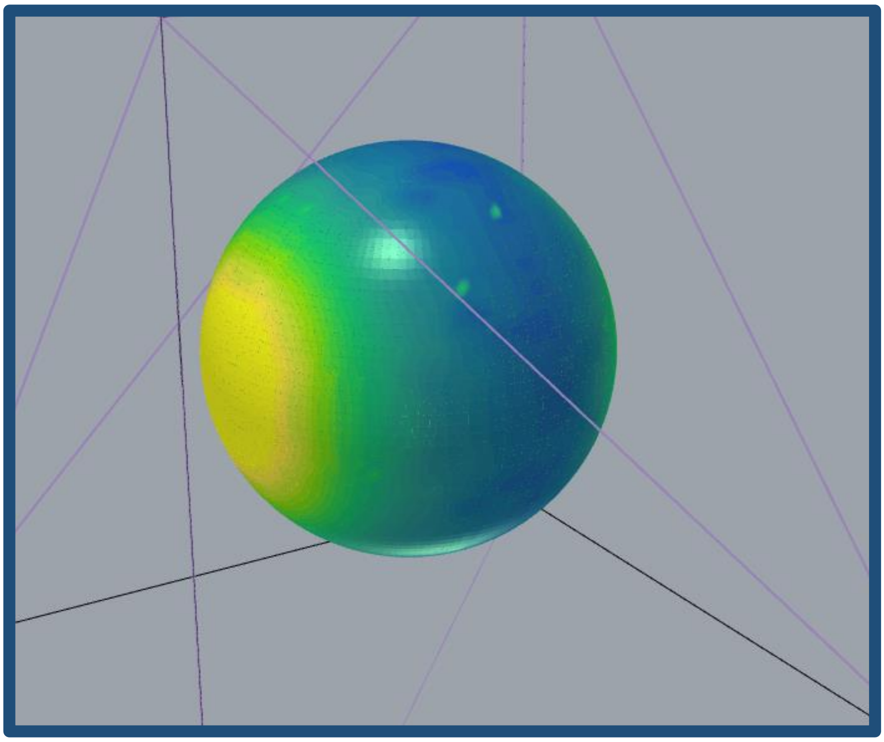

Figure 23: Surface Contour

The transparency of the surface contour can be changed by choosing an option from the 'Opacity' list. The data used to colour the surface can be exported as a .csv le, by clicking on 'export results' and selecting the desired le location.

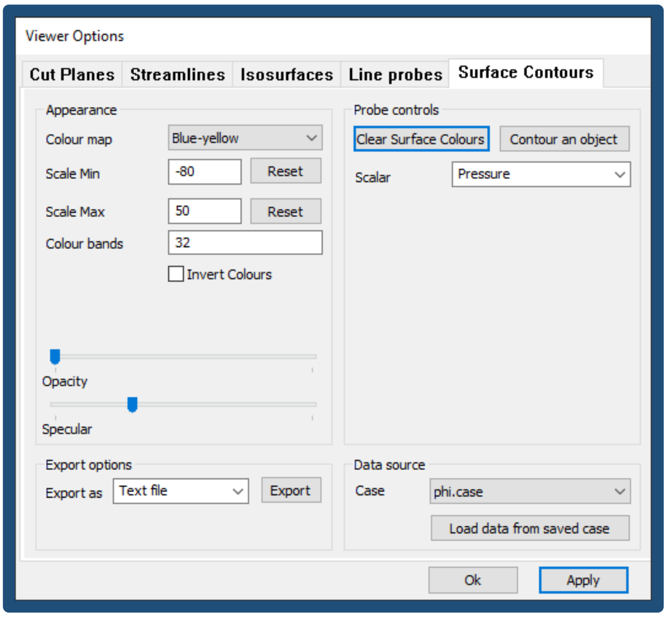

Figure 24: Surface Contour Results Panel

The data at any point, or along a string of points, in the domain can be exported as a .csv le using the line probe.

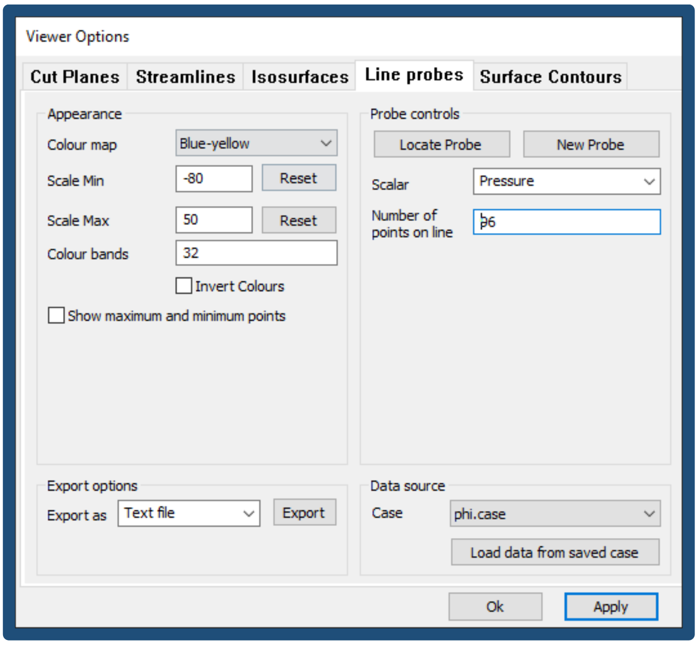

Figure 25: Line Results Panel

Using the 'line' tab of the results panel click on 'Add Probe' and then move the generated probe around the domain:

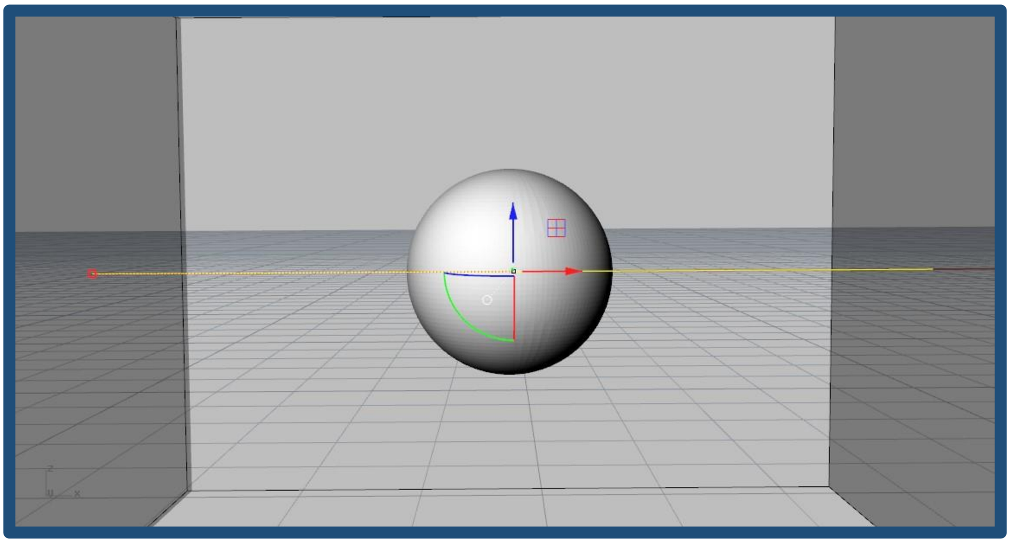

Figure 26: Line Probe

Then enter 100 as the number of 'points on line graph' and select 'Export results'. This can then be opened and plotted in a program such as Excel.

It is possible to display multiple cutting planes, iso-surfaces and stream lines at the same time. To create a new probe either click on the desired probe button from the RhinoCFD toolbar or 'create probe' from the results panel; or to duplicate a probe, press and hold down Alt and then drag the desired probe using the Gumball.

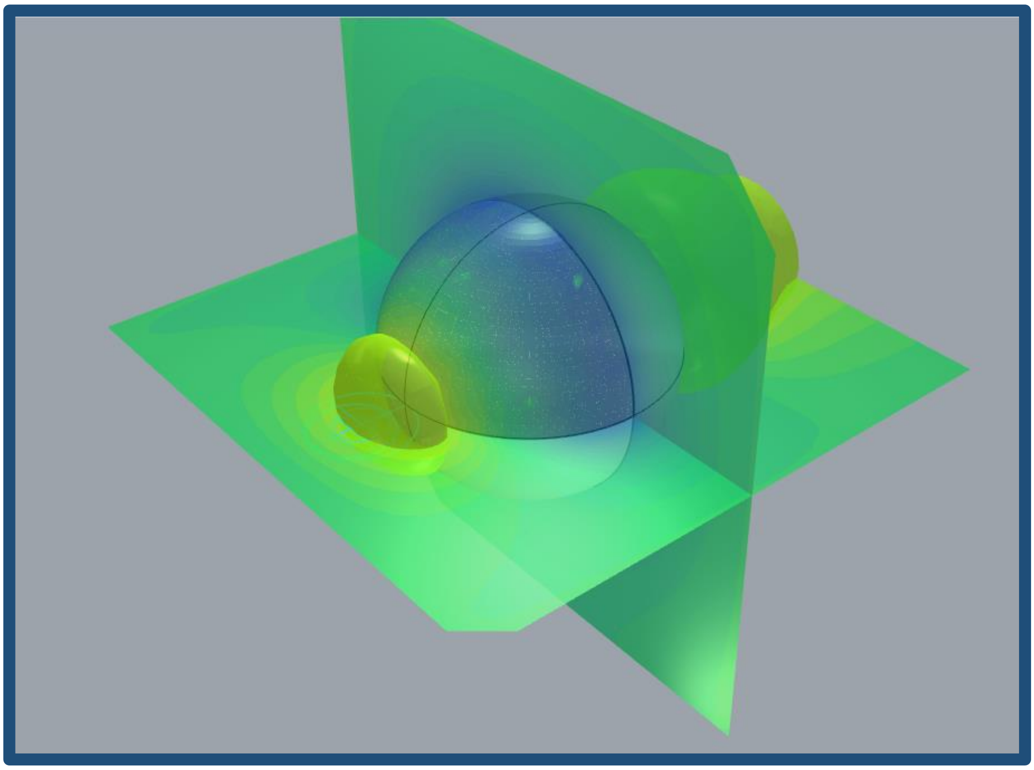

Figure 27: A Range of Cutting Planes and Probes

To remove the visualizations and return to the Pre-processor stage, right click on 'load results'. From here you can change any settings and run the simulation again.