CHAM

RhinoCFD

Powered by PHOENICS

RhinoCFD Tutorial – 3D Aerofoil

Document release date: May 2023

Software version: 2.1.5 January 2022

Solver version: PHOENICS 2022 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

This tutorial focuses on 3D flow over a wing created using the NACA 6409 pro le. This tutorial assumes prior knowledge in creating a mesh with smooth transitions between regions using expansion ratios.

To save some time, we've got the geometry all set for you. Let's start by making a fresh folder to use as our working directory. Grab the file named "3Dwing.3dm" and pop it into the same folder. When you open the file, it'll resemble figure 1.

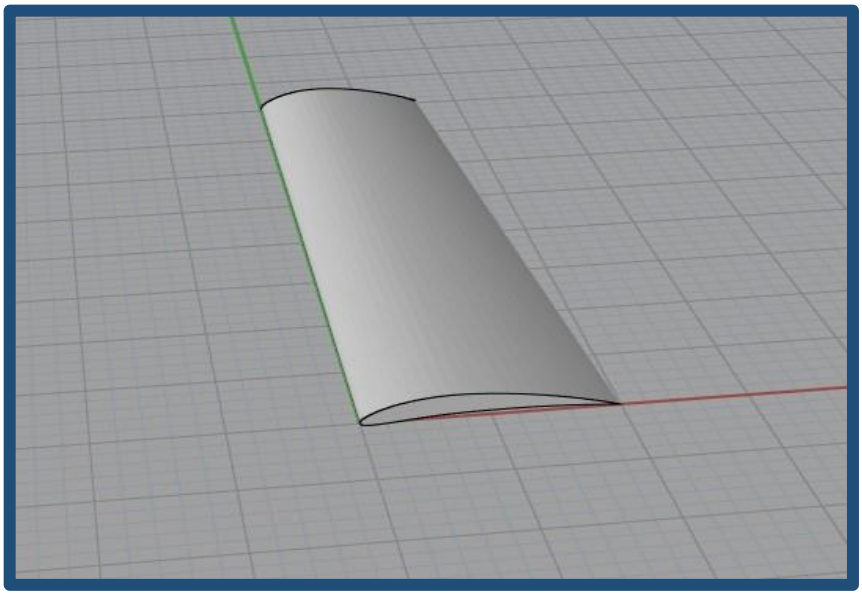

Figure 1: Iso View of Wing Geometry

2 CFD Analysis

First, create a domain around the wing. This can be done by clicking the first button on the toolbar. When promoted set the working directory to the correct folder.

Figure 2: RhinoCFD Toolbar

Next, use the gumball tool to resize and reposition the domain in X and Z. We shall use approximately 1 chord length upstream of the aerofoil, 5 chord lengths downstream and 2.5 chord lengths above and below.

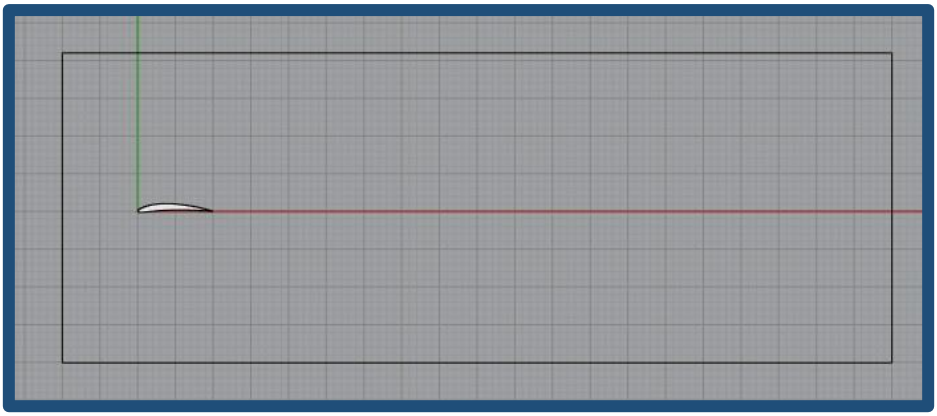

Figure 3: Side View of Aerofoil and Domain

As this case is 3D, we also need to resize and reposition the domain in Y. Size wise; the domain should be approximately twice the span of the wing in the Y direction. The domain should be positioned such that the thickest part, the root, should be touching the wall of the domain as shown below.

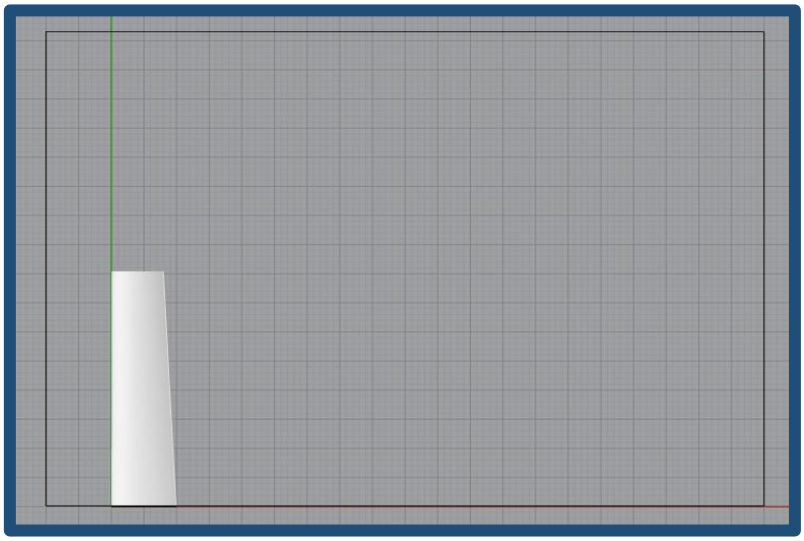

Figure 4: Top View of Aerofoil and Domain

2.1 Creating Fluid Boundaries

The next stage is to add an inlet and outlet to each end of the domain. This can be done by right clicking on the second toolbar icon. Go to the domain faces tab and add ow to the Xmin end of the domain and make Xmax open, see image below. Using the settings button for Xmin, add an X velocity of 35m/s.

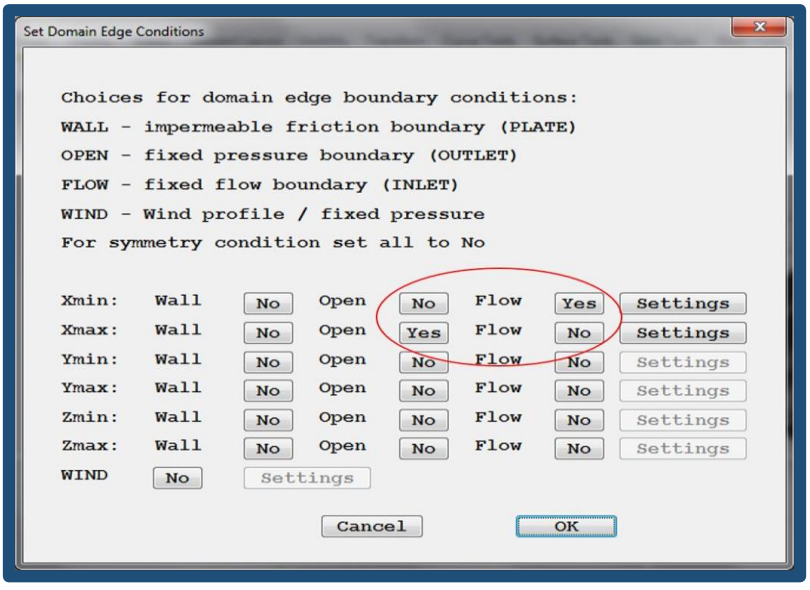

Figure 5: Domain Faces Menu

2.2 Refining the Mesh

Next we need to create the mesh; you can edit mesh settings by left clicking on 'show grid dialog'. A fine mesh is required in the region of the aerofoil and a coarse mesh further away.

When viewing the mesh, you should have something like Figure 6.

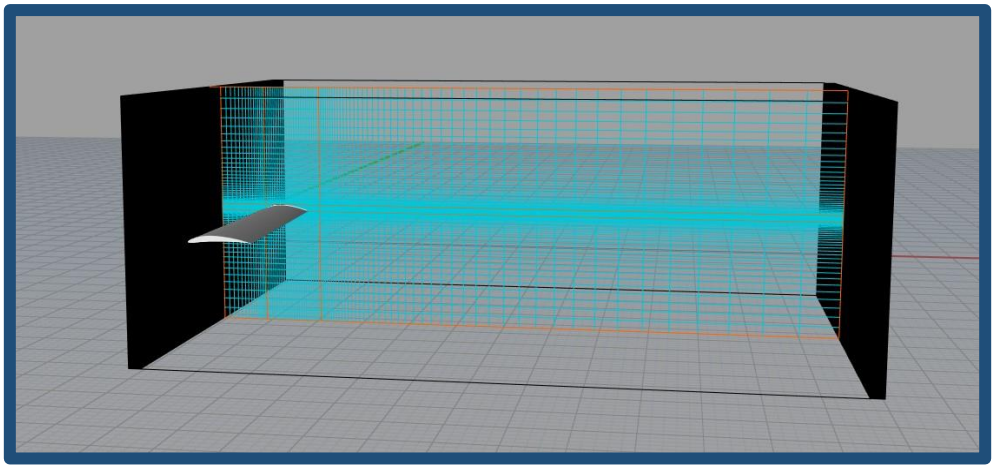

Figure 6: Iso View of Grid

Next you need to set up the mesh in y. 50 cells in the region containing the aerofoil and 20 cells in the other region should be sufficient. Adjust the expansion ratio of the region with fewer cells to achieve a smooth transition.

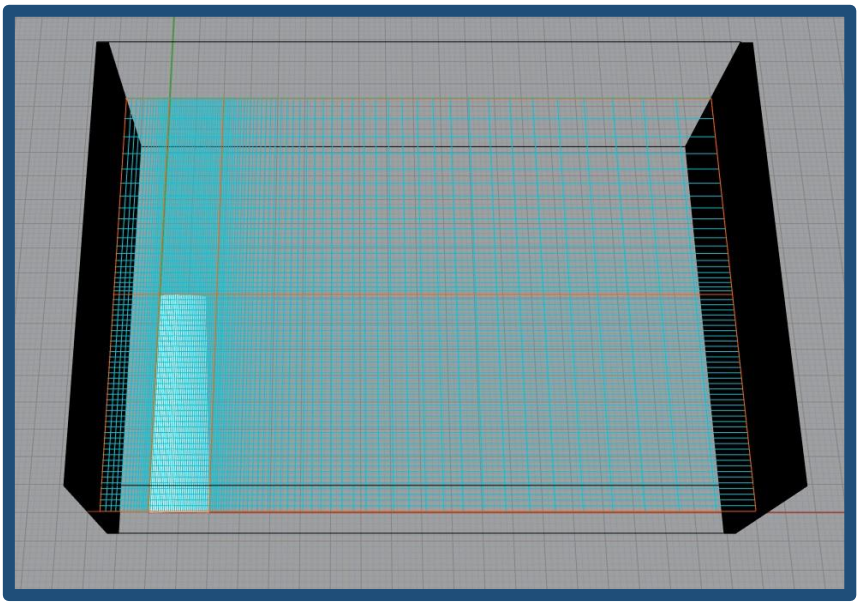

Figure 7: Top View of Grid

2.3 Numerics Settings

Left click on the second toolbar icon to access the main menu. Under the 'numerics' tab set the number of iterations to 1000.

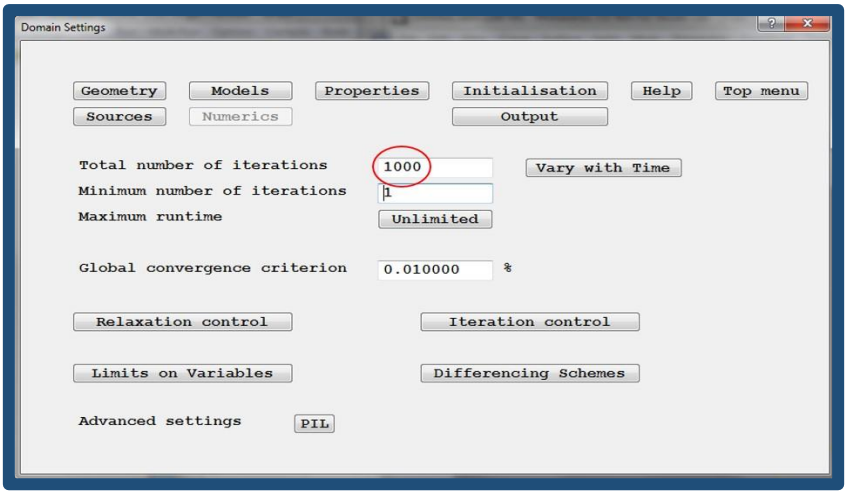

Figure 8: Numerics Menu

Go to the relaxation control panel and turn 'Automatic Convergence Control' to OFF and set u1, v1, and w1 to 0.01. If the solution struggles to converge these values may need to be reduced further.

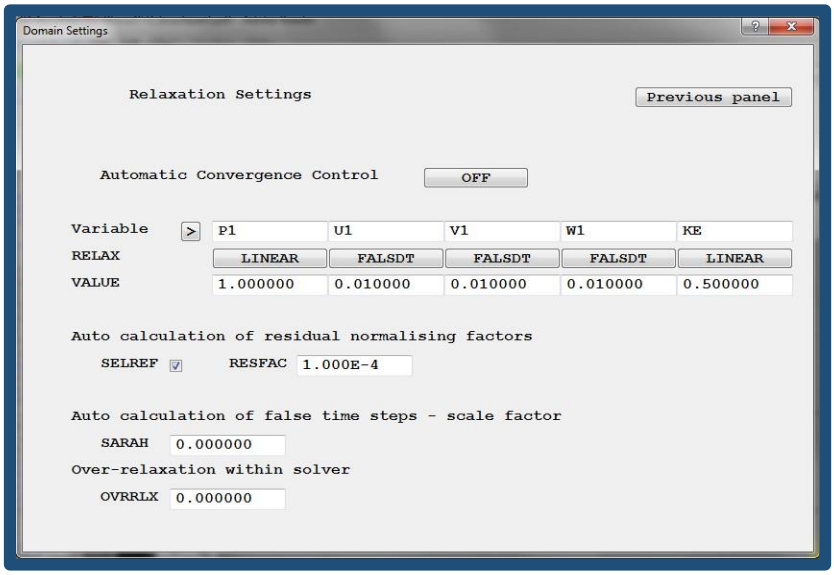

Figure 9: Relaxation Settings

2.4 Running the Simulation

Click on 'Run Solver' and the simulation will start.

2.5 Results

Once the solver has finished, results can be viewed by clicking on 'Load Results'.

You can change the plotting plane by selecting the cutting and rotating by the desired angle. The location of the slice can be changed by using the gumball tool to move the plotting plane in X, Y or Z.

The figures 10 and 11 show examples of plots produced by plotting pressure.

You can also produce similar plots of velocity, as can be seen in figure 12

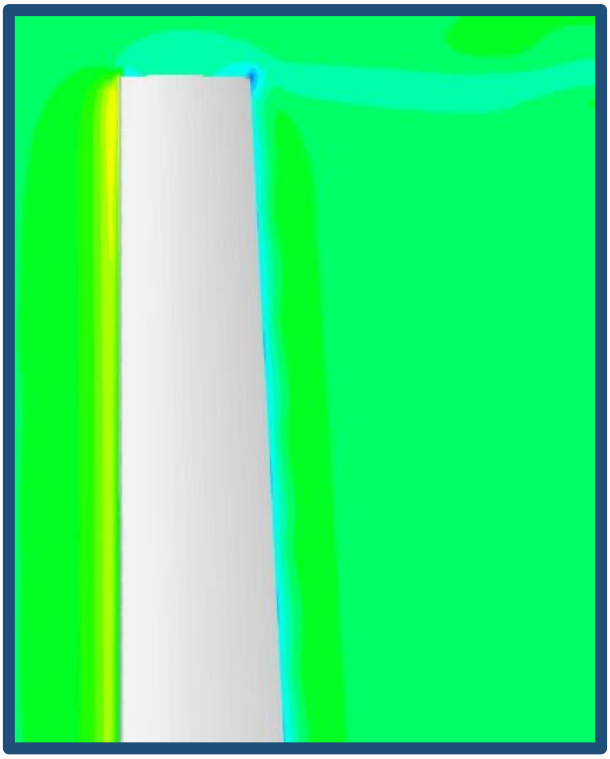

Figure 10: Pressure Contour Top View

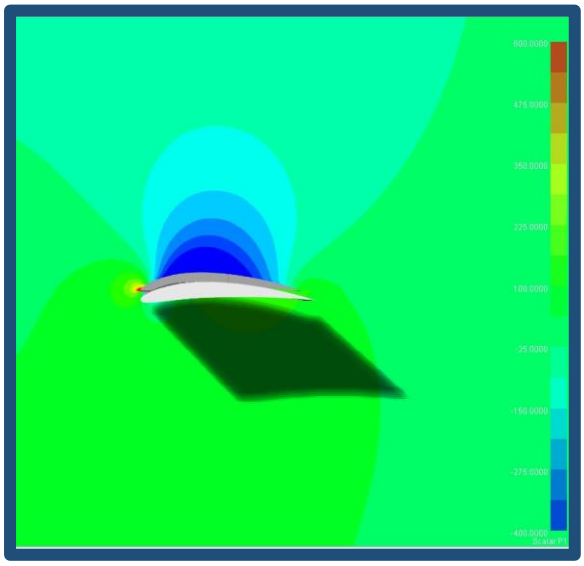

Figure 11: Pressure Contour Side View

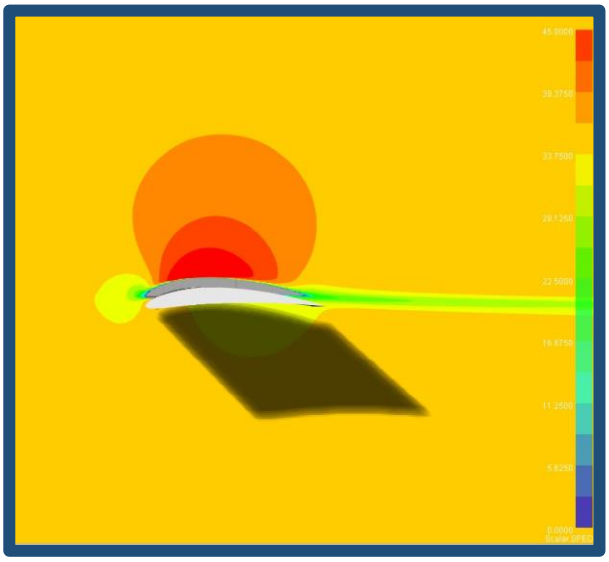

Figure 12: Velocity Contour Side View