RhinoCFD Tutorial

3D Aerofoil

RhinoCFD Official document produced by CHAM
February 10, 2017

Introduction

This tutorial focuses on 3D flow over a wing created using the NACA 6409 profile. This tutorial assumes that you understand how to create a mesh with smooth transitions between regions using expansion ratios.
In order to save time, completed geometry is provided. Create a new folder which can be used as the working directory. Download the file 3Dwing.3dm and save it in the working directory. When you open the file it should look like figure 1.

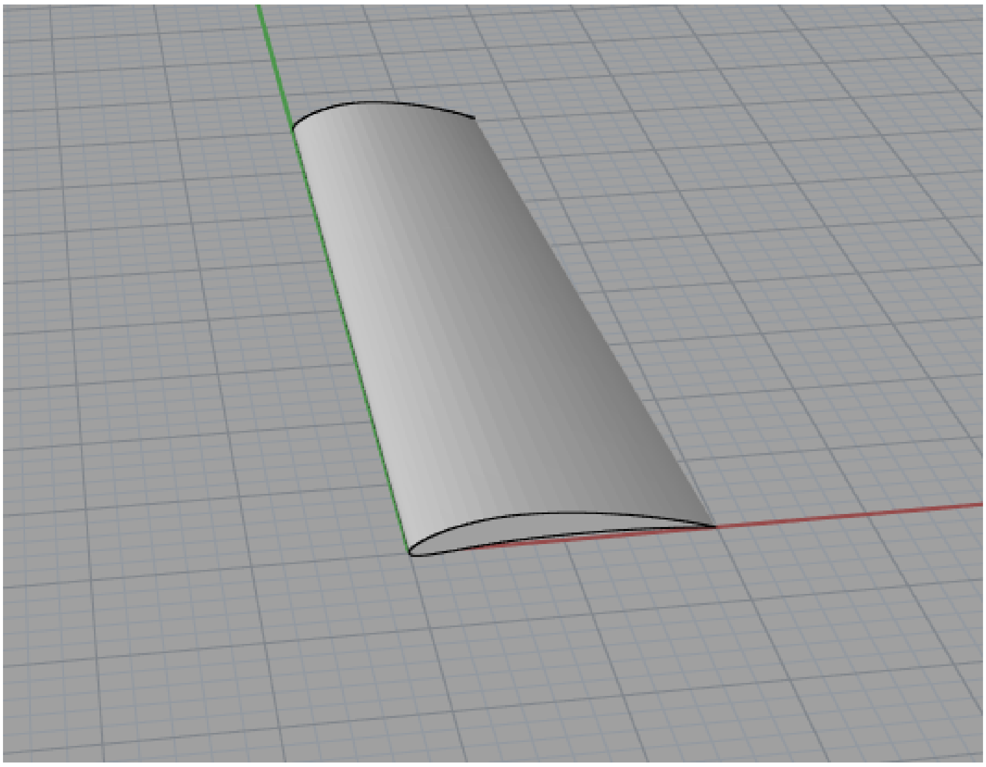

Figure 1: Iso View of Wing Geometry

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.

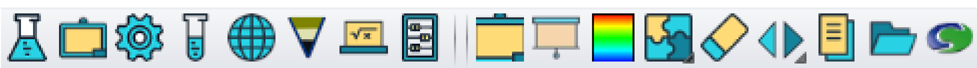

Figure 2: RhinoCFD Toolbar

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.

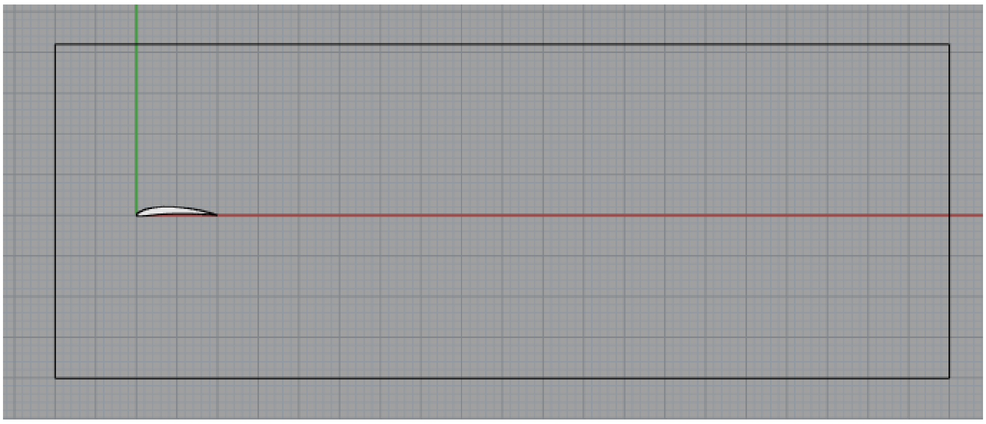

Figure 3: Side View of Aerofoil and Domain

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.

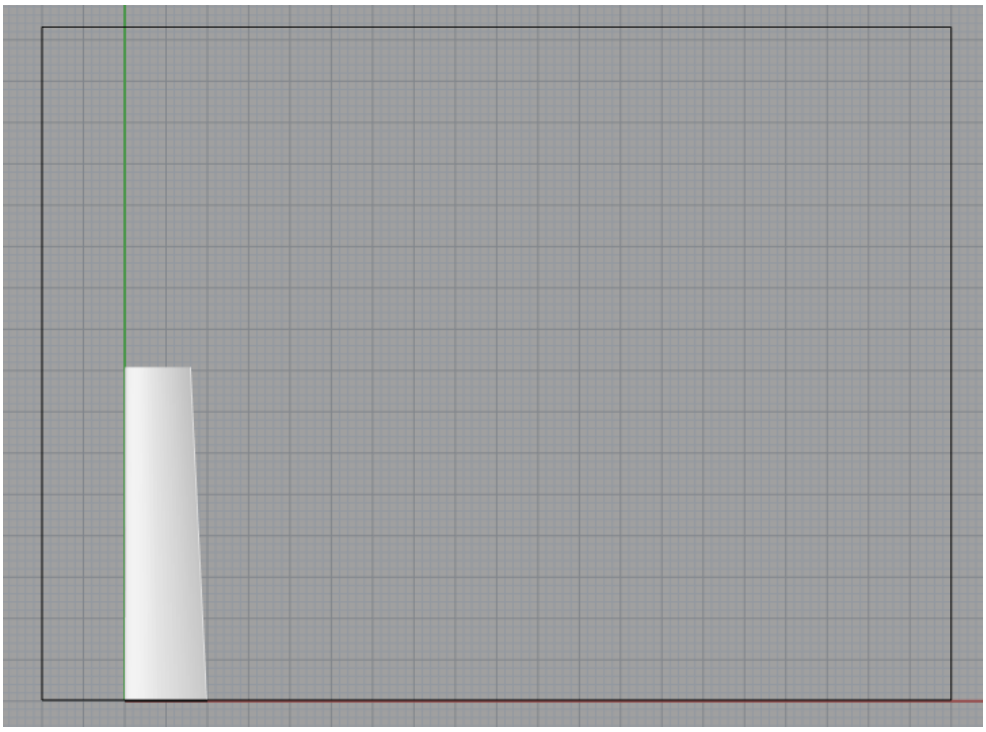

Figure 4: Top View of Aerofoil and Domain

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 flow 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.

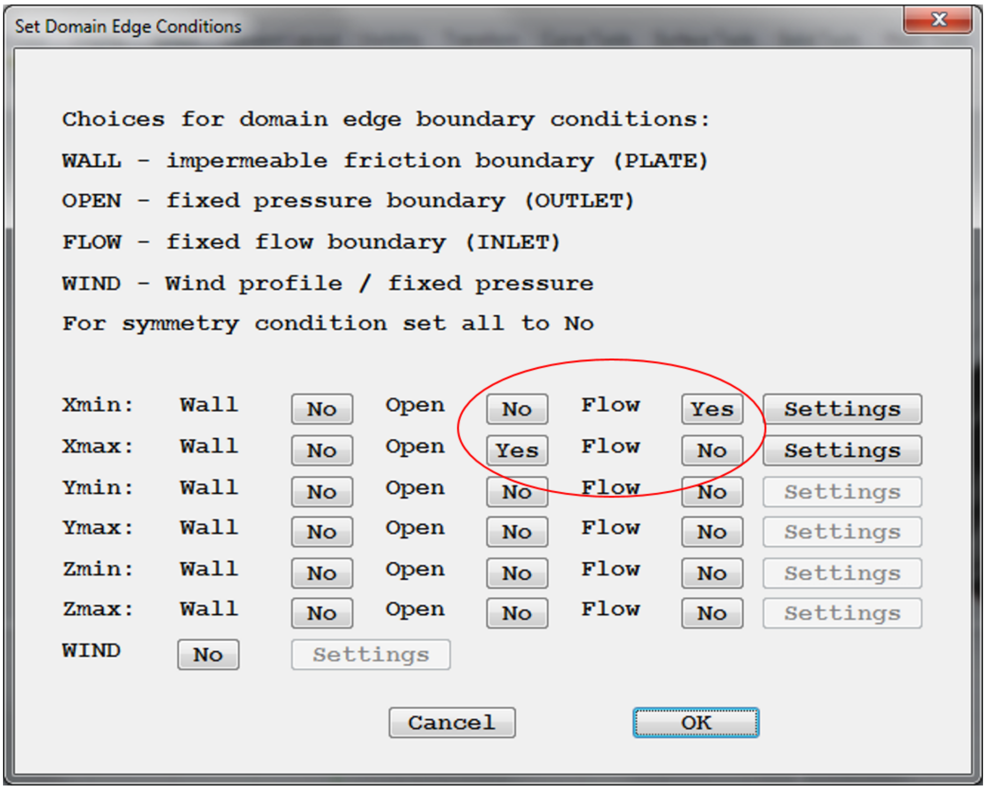

Figure 5: Domain Faces Menu

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.

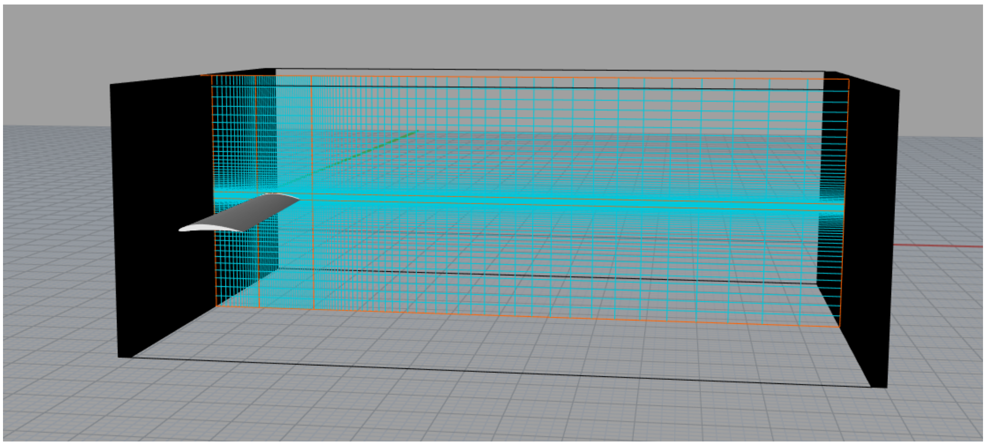

Figure 6: Iso View of Grid

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.

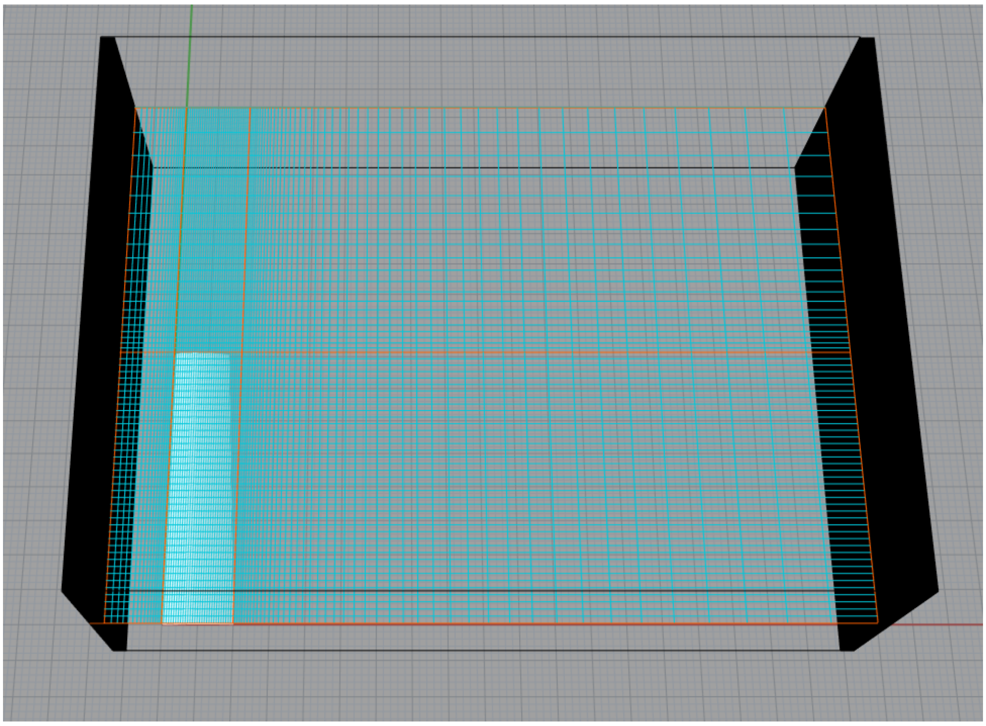

Figure 7: Top View of Grid

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.

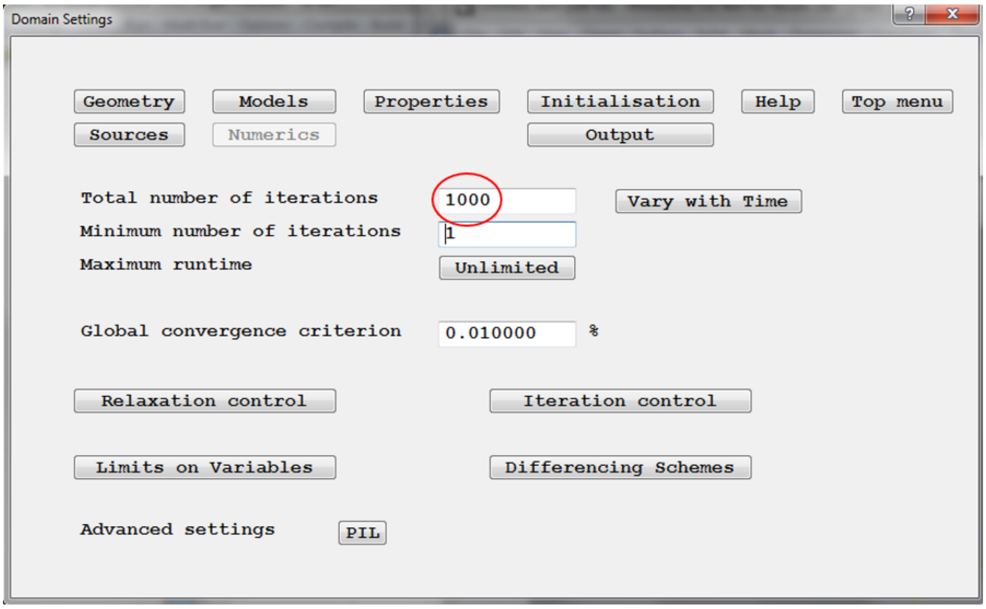

Figure 8: Numerics Menu

Go to the "relaxation control" panel and turn 'Automatic Convergance 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.

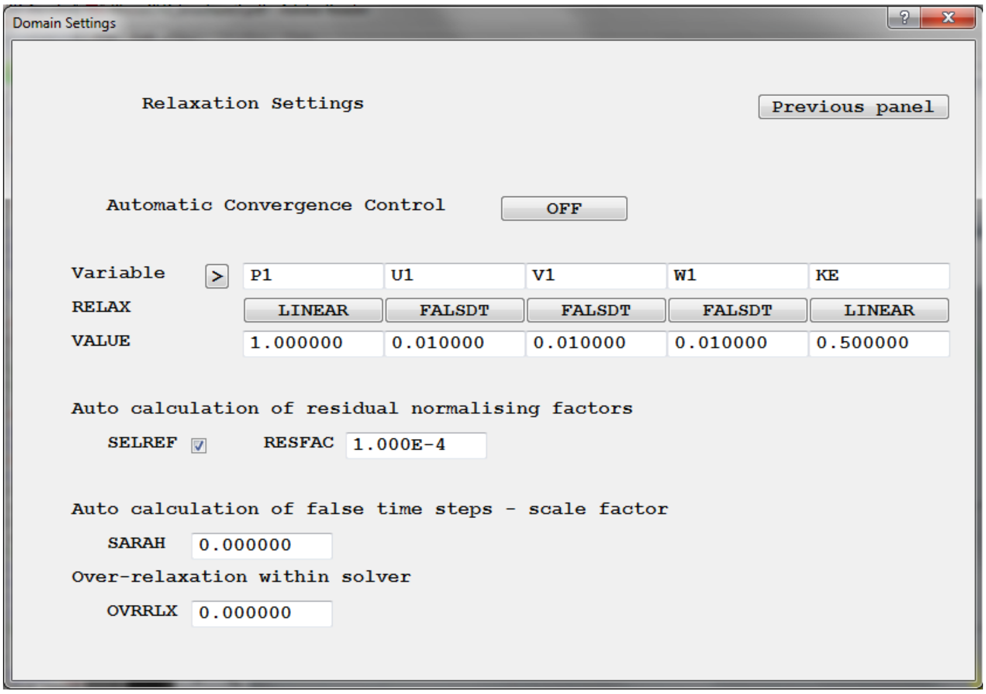

Figure 9: Relaxation Settings

Running the Simulation

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

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

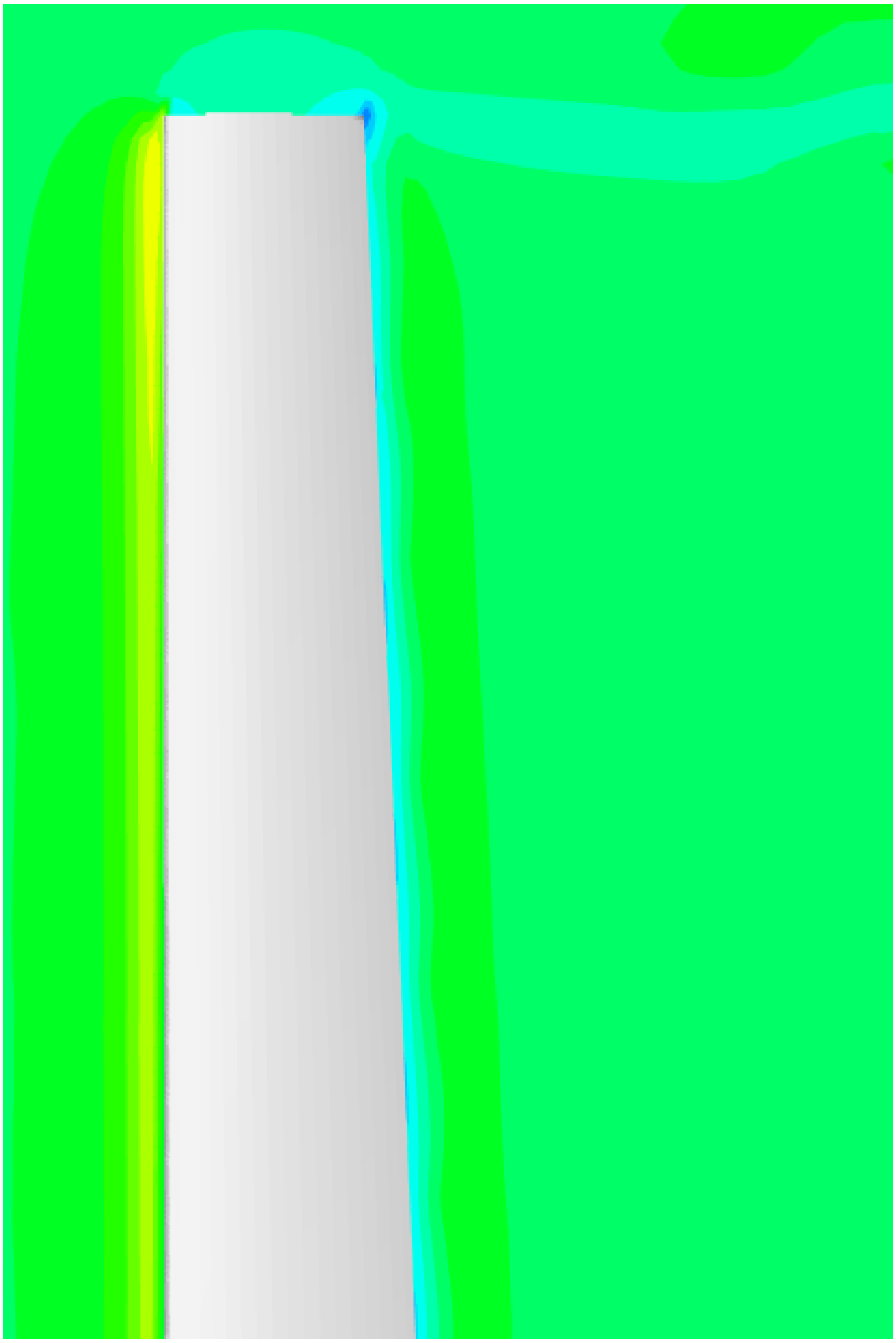

Figure 10: Pressure Contour Top View

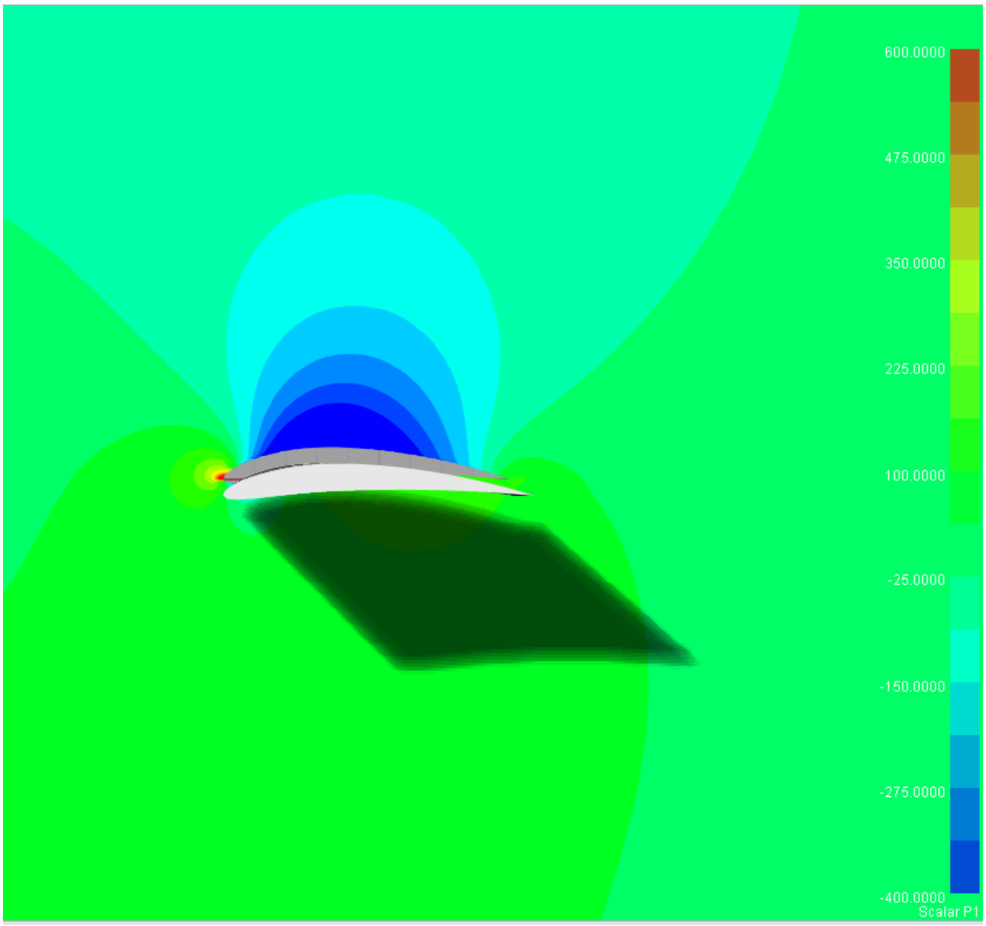

Figure 11: Pressure Contour Side View

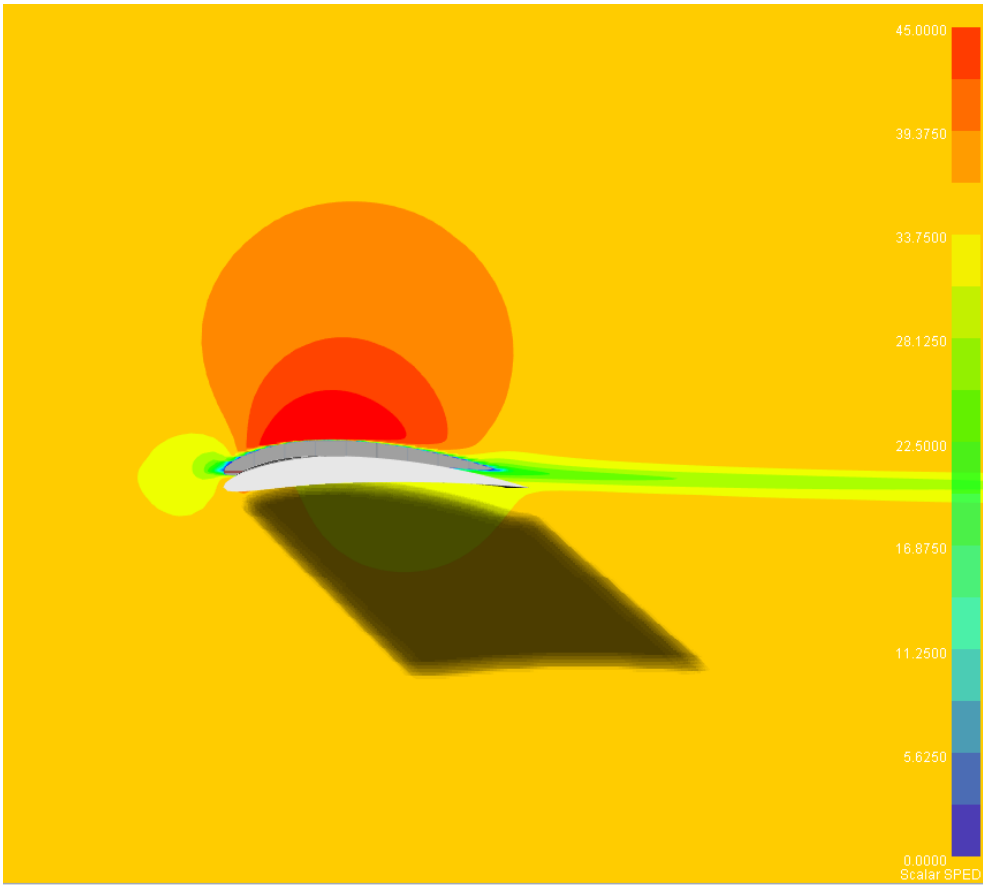

Figure 12: Velocity Contour Side View

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