Velocity and Pressure Contour Plot in ANSYS Fluent Simulation

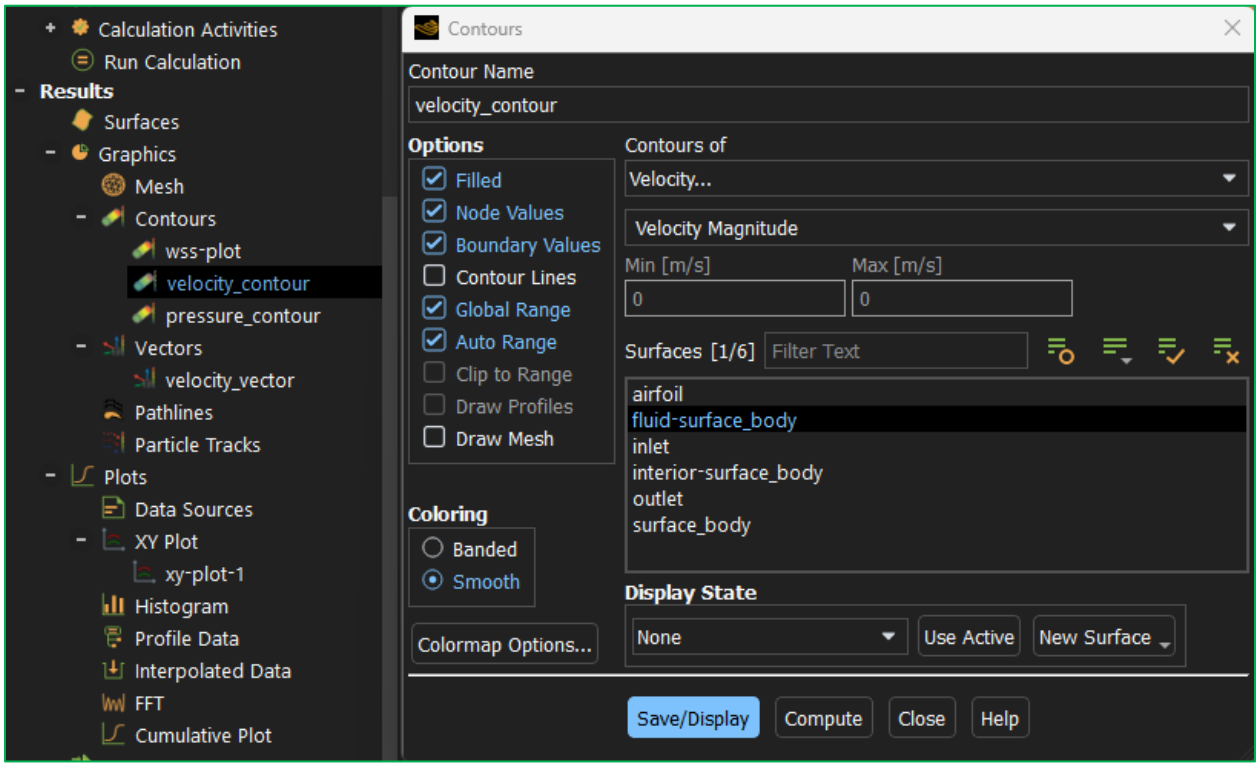
Description: Contours of velocity and pressure to understand fluid dynamic properties of a drag (and lift) inducing airfoil in a given fluid domain

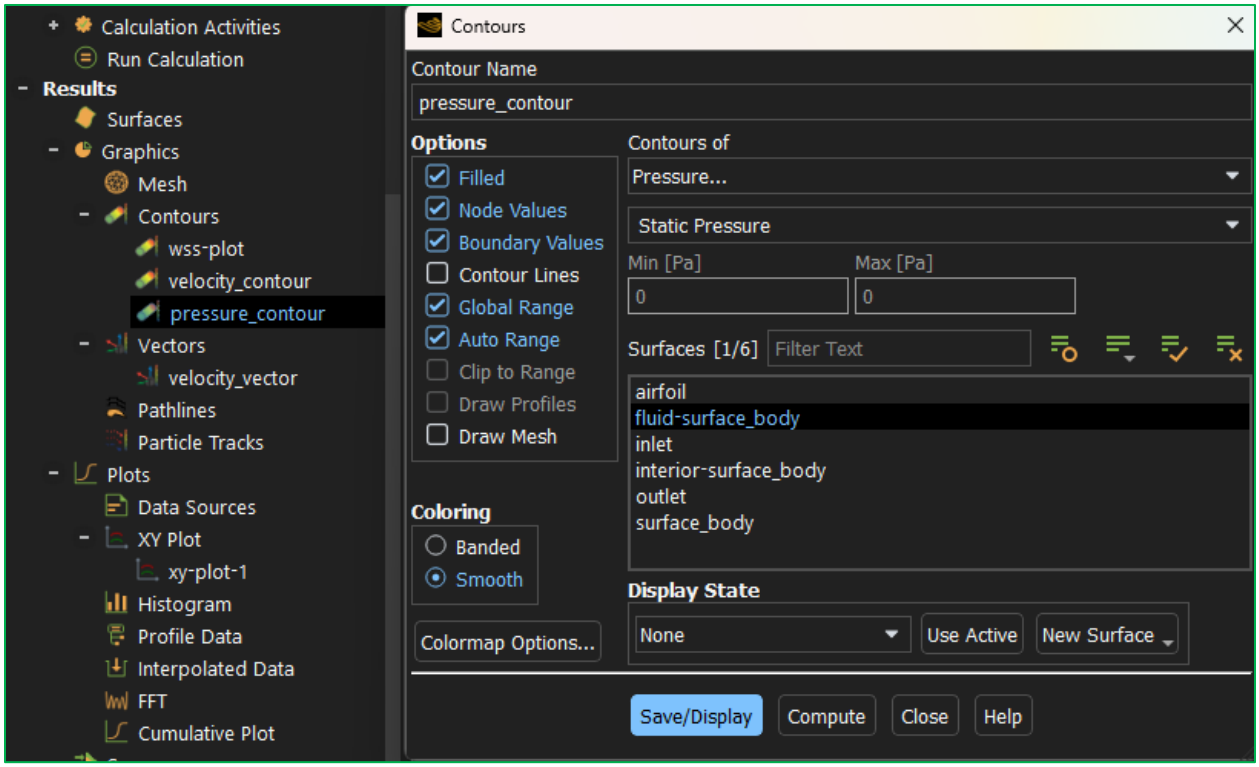
Keyword: *Aerodynamics, CFD, ANSYS, Simulation, Velocity, Pressure, Airfoil*

Setting Up the Plot

Once we are done with the simulation of the airfoil, it's time to discuss some of the results and understand flight dynamics – the fun part!

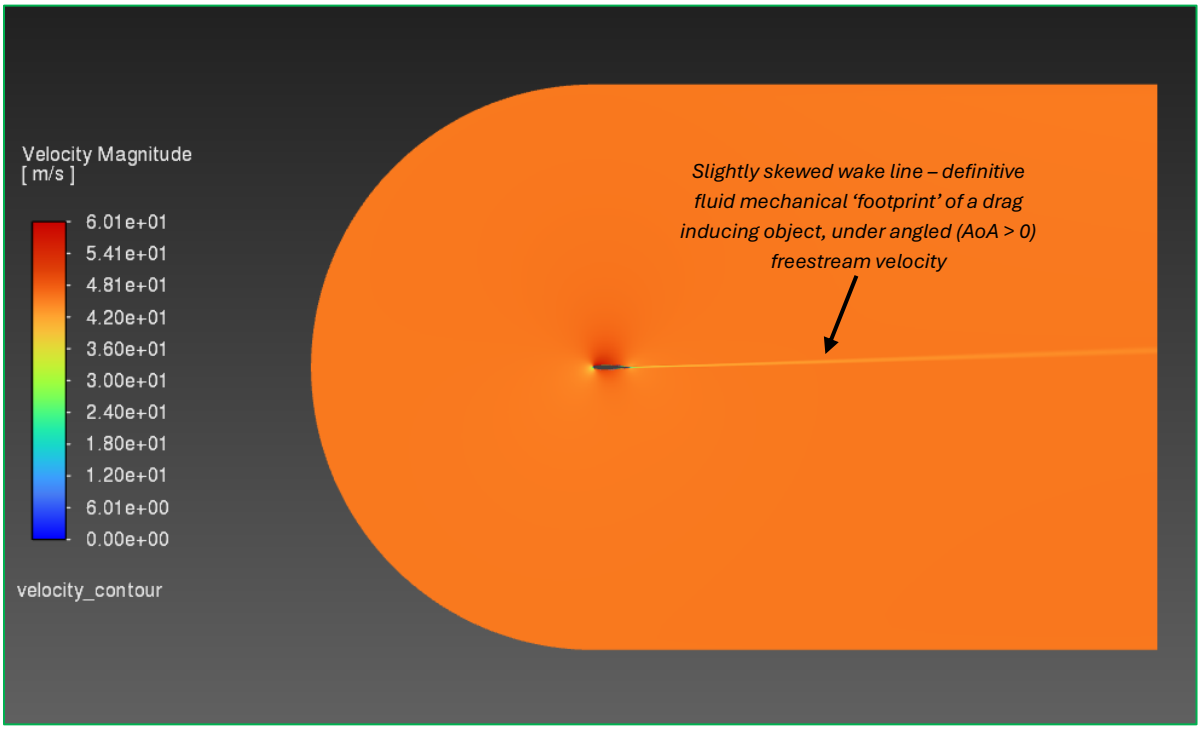
In the simulation module, create new contour for velocity and pressure –

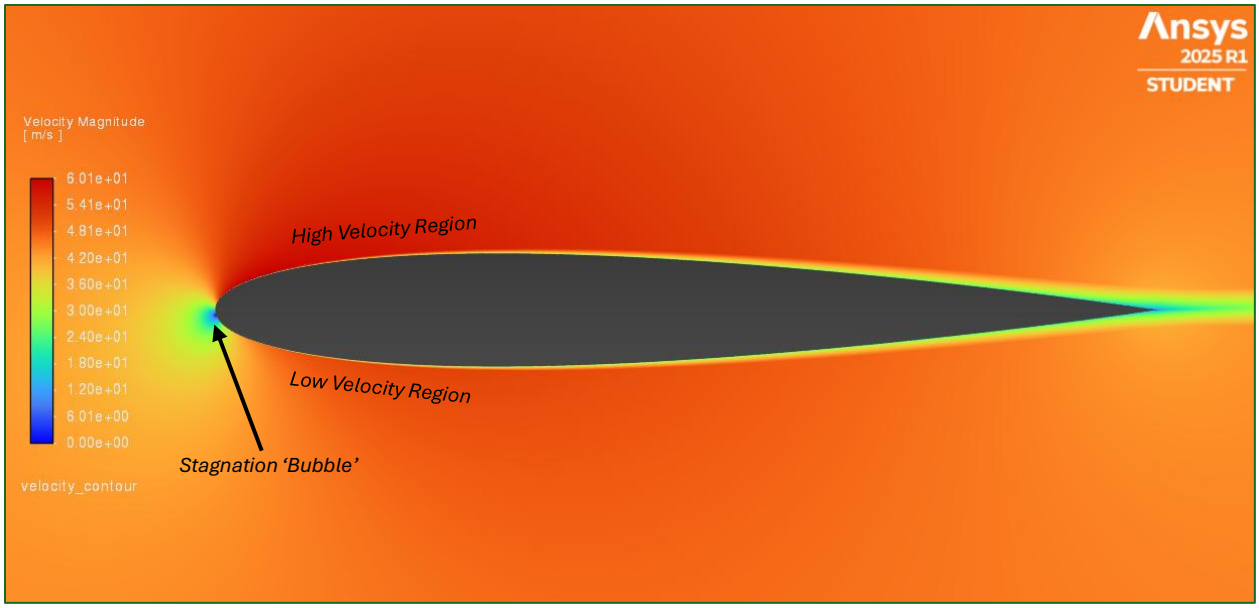




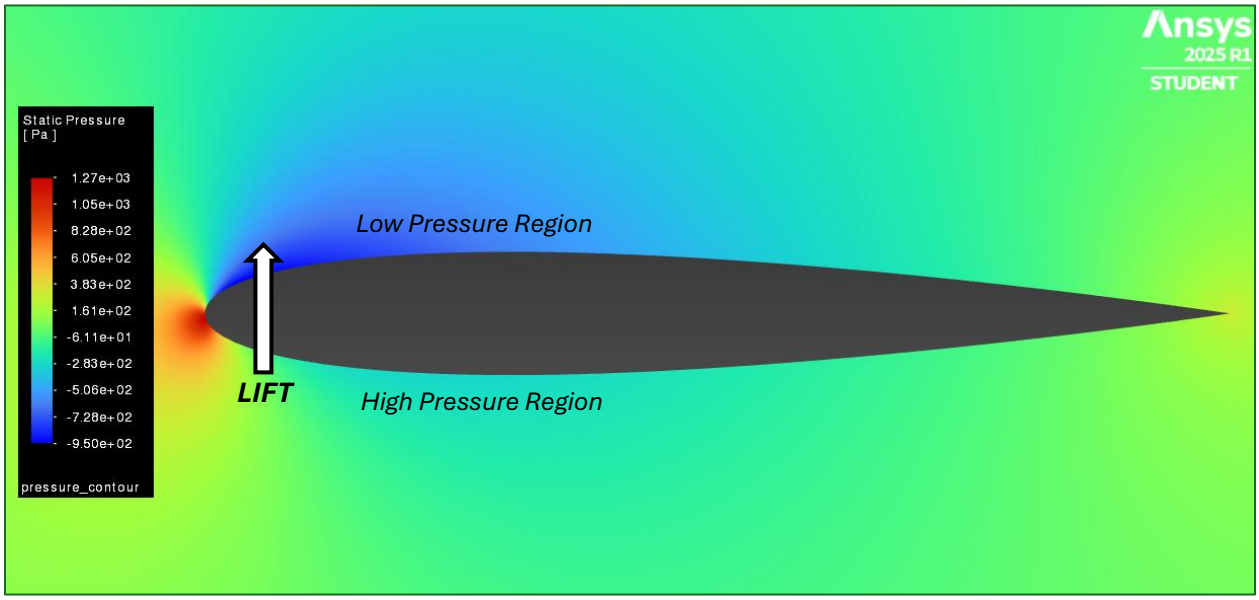
Once done, we are ready to dissect the 'painting' these two contours will provide us. (Ignore the *wss-plot* contour, we will get to it in another discussion)

Velocity Contour





Pressure Contour



All the images are pretty self-explanatory. But in case you missed it (since you're probably doomscrolling TikTok), the airfoil has achieved lift here. There is more discussion about the aerodynamic phenomena on the Velocity Vector Plot result discussion.

Omar Saif
CFD Enthusiast
July 9, 2025