

Velocity and Pressure Contour Plot in ANSYS Fluent Simulation

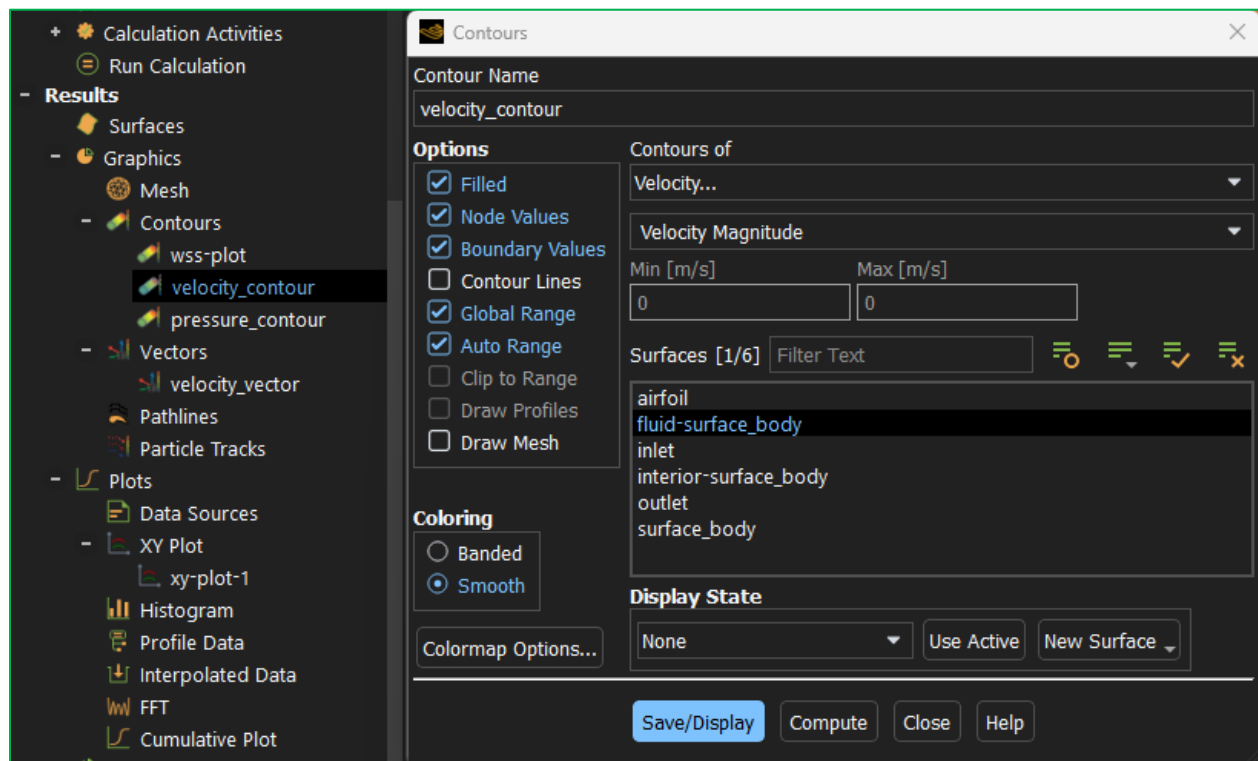
Description: Contours of velocity and pressure to understand fluid dynamic properties of a drag (and lift) inducing airfoil in a given fluid domain

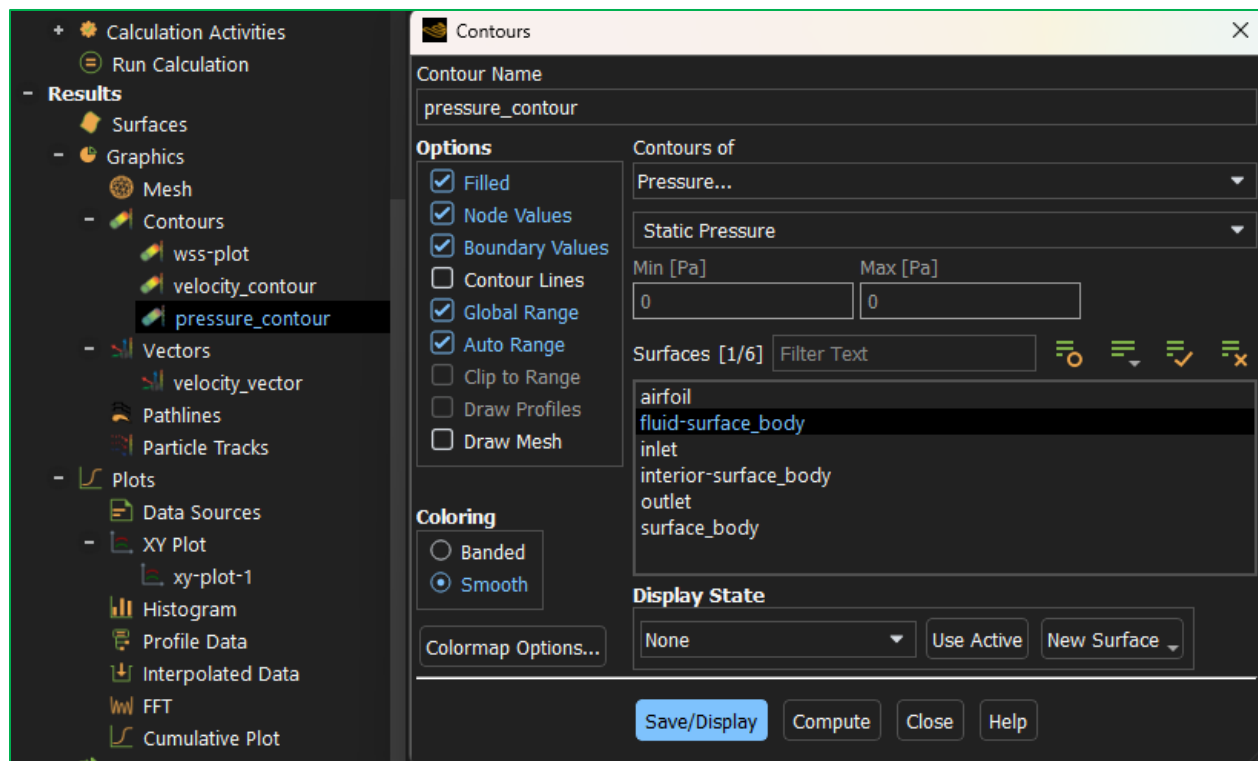
Keyword: Aerodynamics, CFD, ANSYS, Simulation, Velocity, Pressure, Airfoil

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!

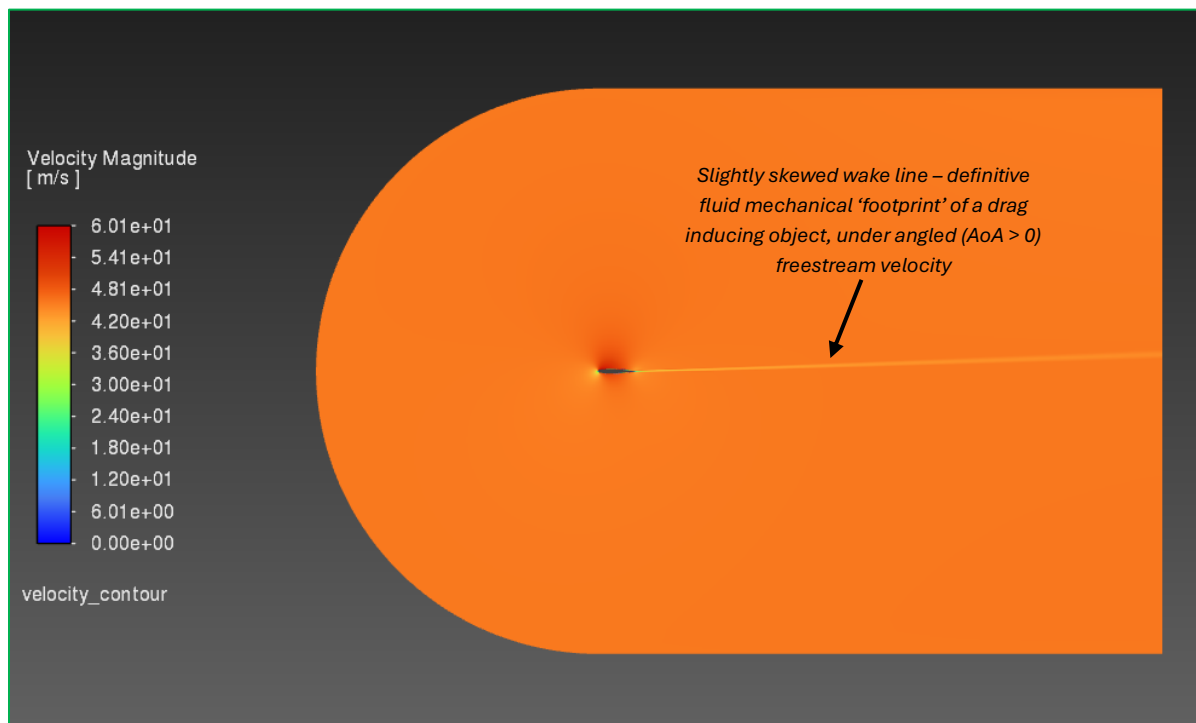
In the simulation module, create new contour for velocity and pressure –

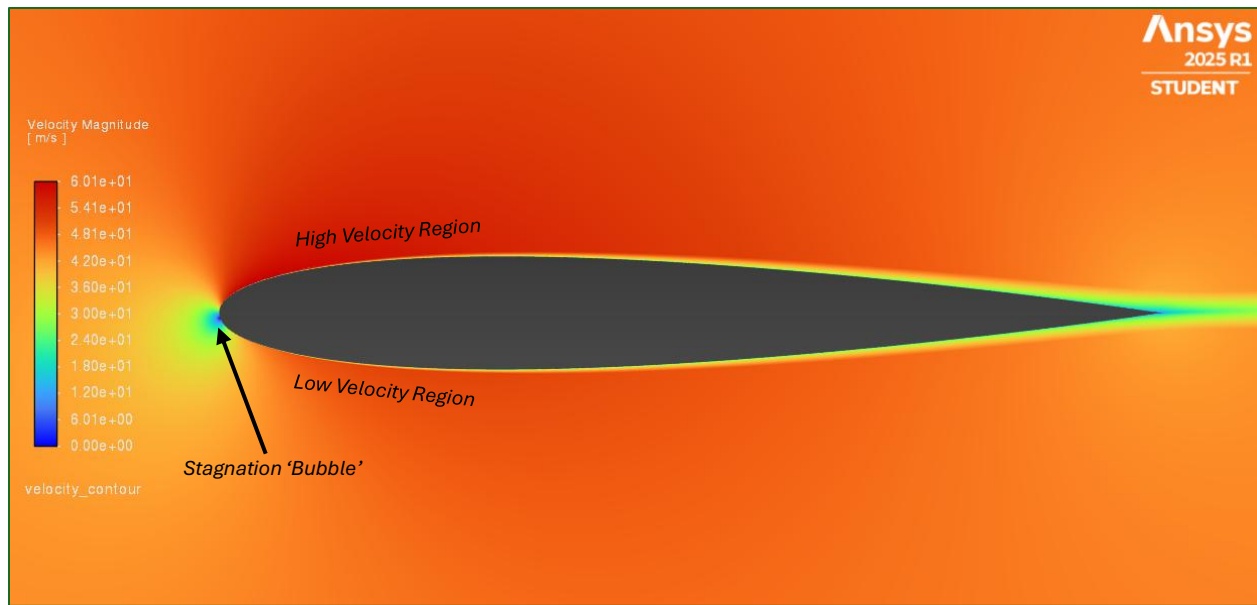




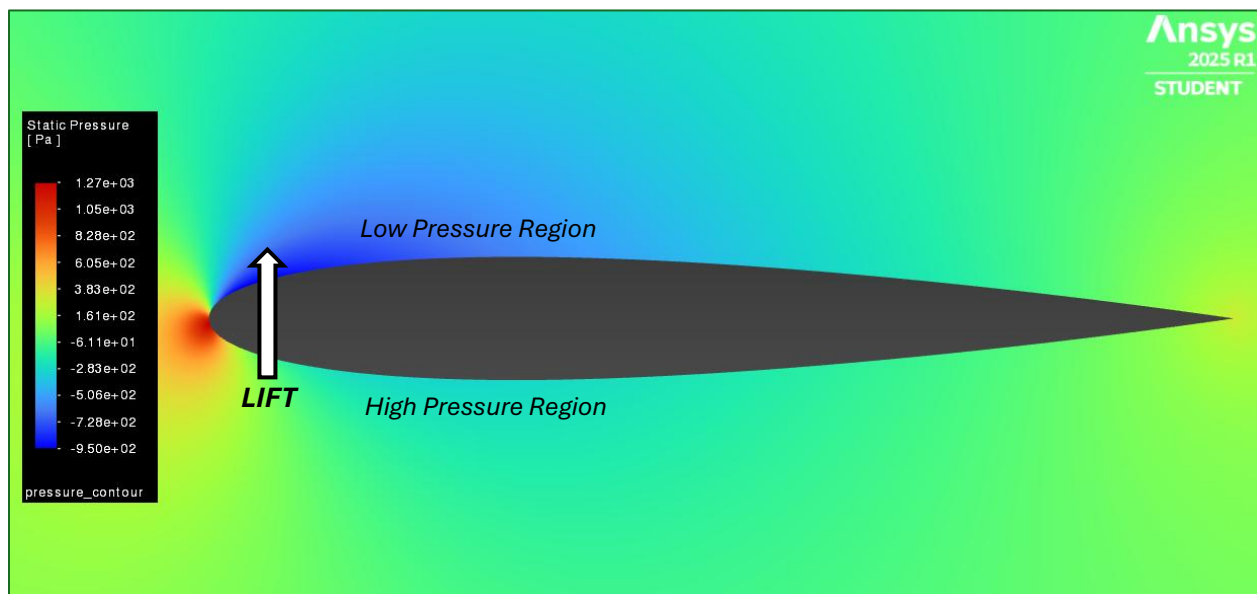
Once done, we are ready to dissect the ‘painting’ these two contours will provide us.
(Ignore the *wss-plot* contour, we will get to it in another discussion)

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 on the 'Velocity Vector Plot' result discussion – you can find that in my website.

Omar Saif
 CFD Enthusiast
 July 9, 2025