Velocity Vector Plot in ANSYS Fluent Simulation

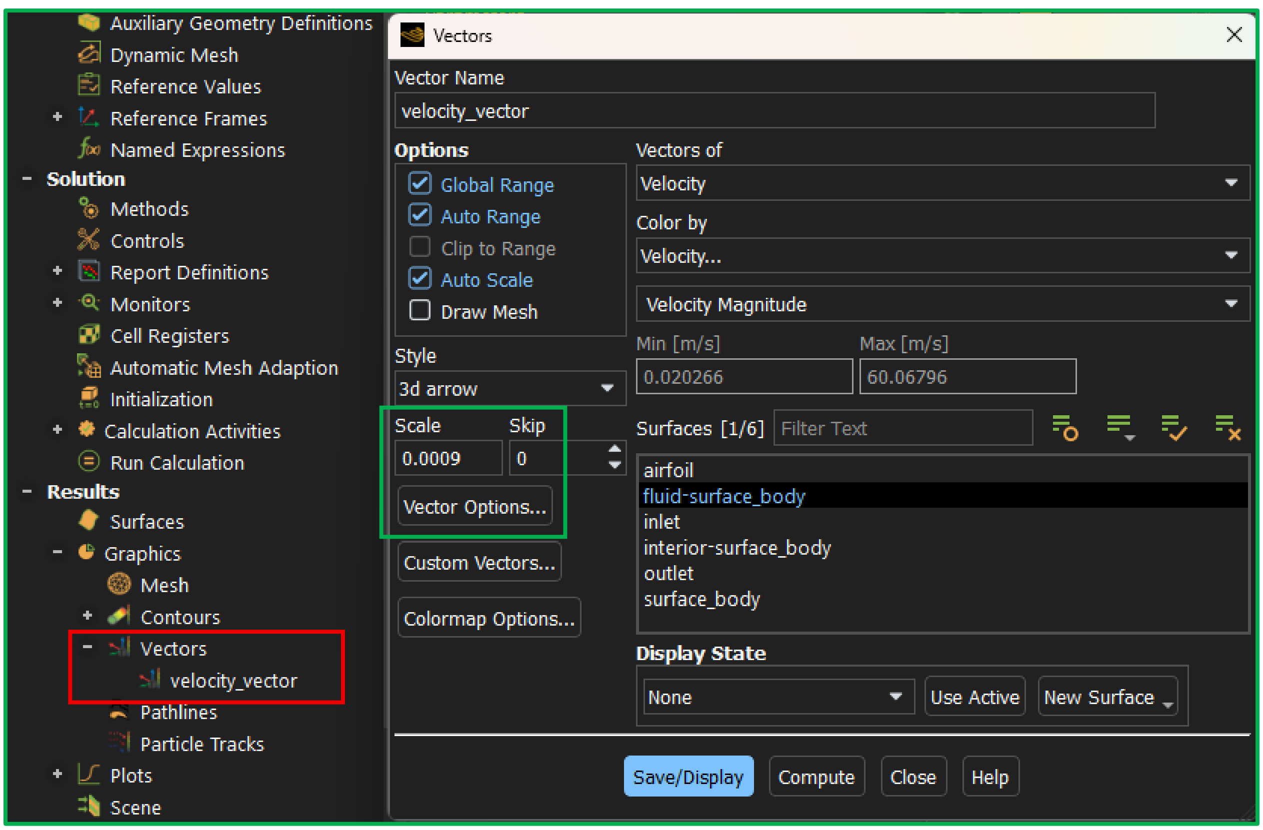
Description: Plotted Velocity Vectors that elucidate the flow pattern and behavior of regional fluid domains around the airfoil

Keyword: *Aerodynamics, CFD, ANSYS, Simulation, Vector, Airfoil*

Setting Up the Plot

I assume you have finished the simulation and looked into the '*velocity and pressure contour plots*' result discussion. This is the perfect follow up – with a little more details to the whole CFD magic!

Make sure you are still in the simulation (we are yet to get into the CFD-Post, or the Result module; this is a result discussion right in the simulation module). Consult the following screenshot –

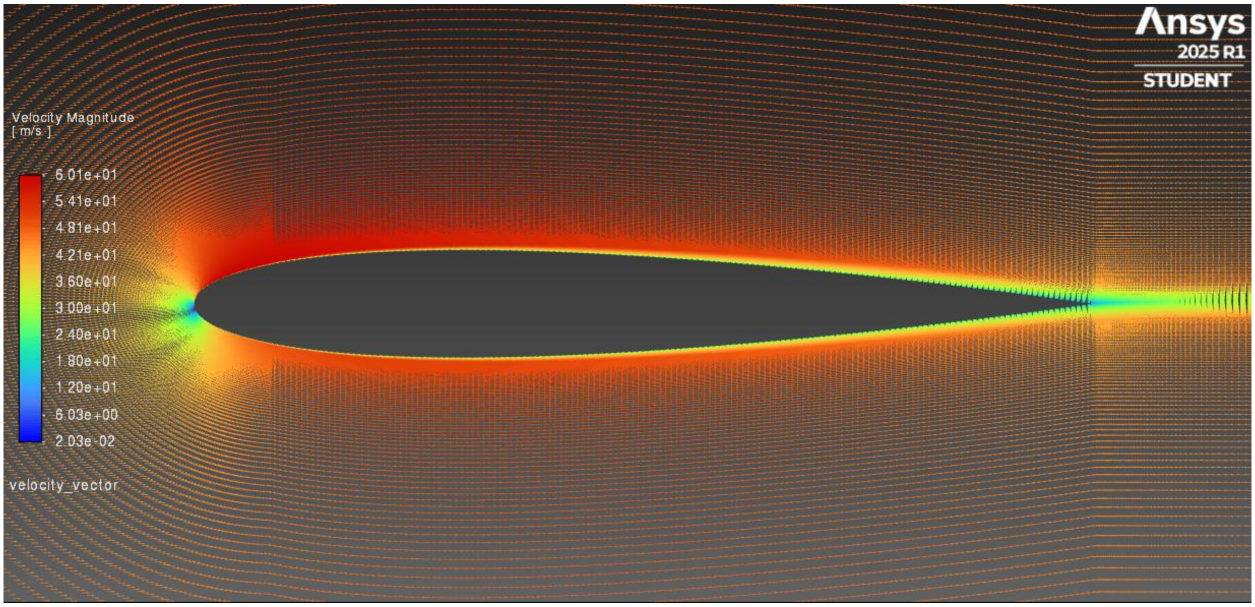


Right-click on Vectors – New. Follow all the options shown. However, feel free to play with Scale and Vector Options (marked green); as they might impact the shape and visuals of the plots you may try to implement. They are airfoil shape, AoA and size dependent.

Once all the options are set and locked, hit Save/Display.

Vector Plot Discussion

Velocity vectors show the attitude of fluid in small domains (say a bunch of fluid molecules grouped together and acting as units) in a fluid mechanical analysis. Say you want to understand the fluid flow around an airfoil. Fluid vectors are the tool for you. It's so easy to understand that even a child can get the full picture of Navier-Stokes and Bernoulli's equations. Yes, a child, understanding velocity and pressure head. And as an added bonus, knows exactly how airplanes work!



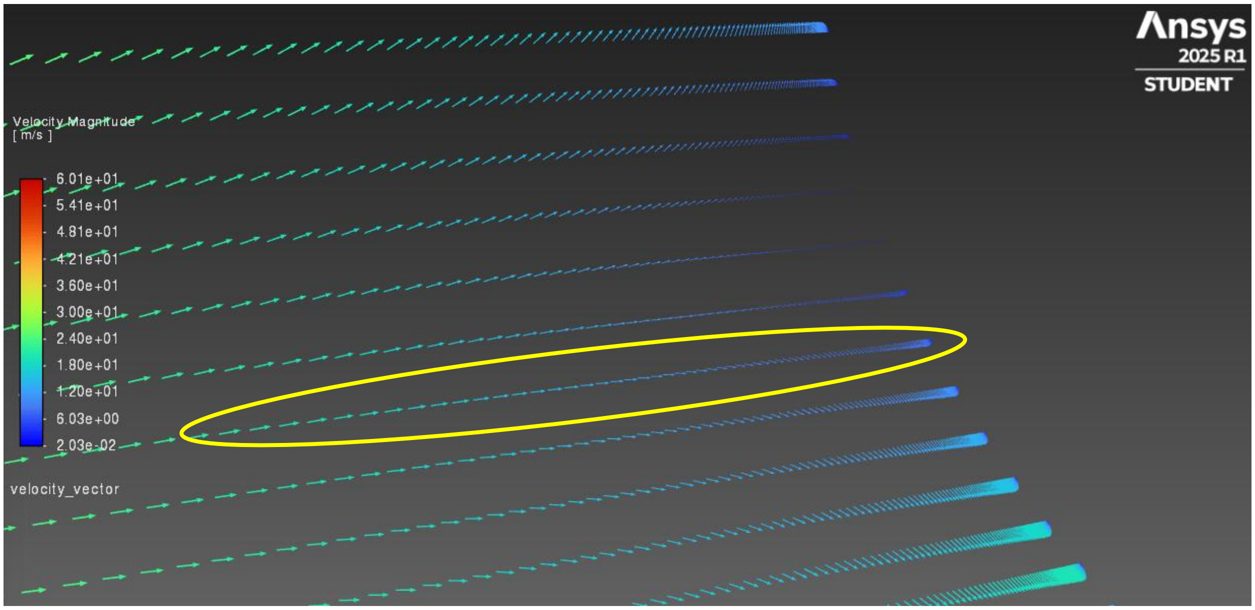
What you are seeing my friend is Physics, stripped naked.

Pretty, isn't she?

How do Airplanes Generate Lift

Let's unpack, shall we?

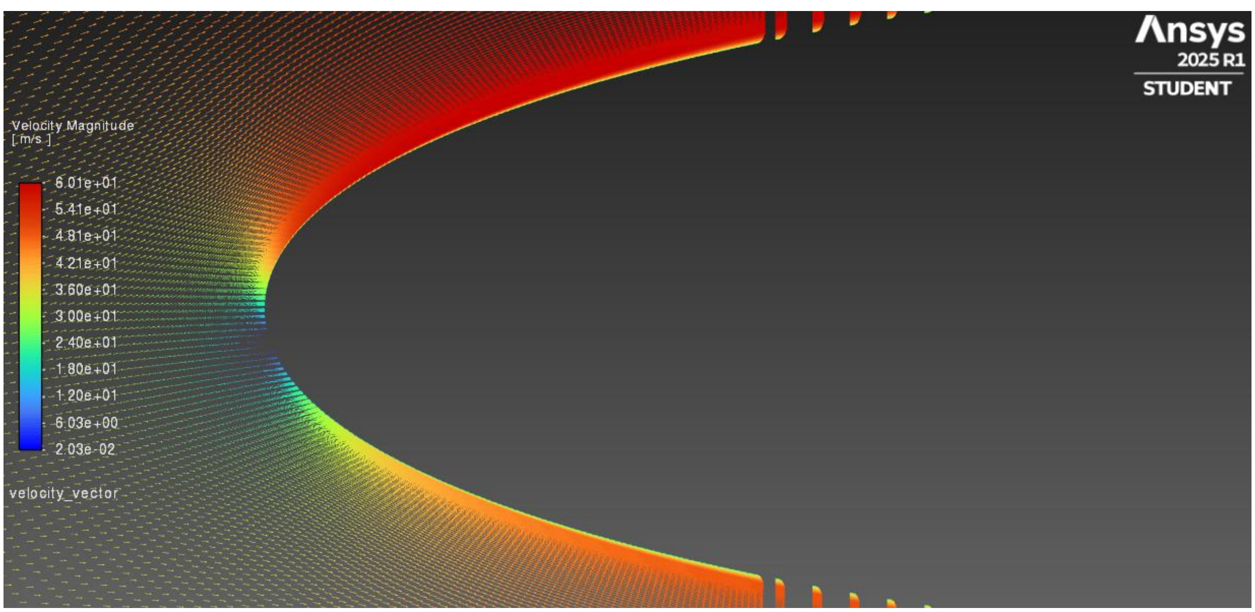
If you look at the leading edge of the airfoil, you will see a small patch of blue. That is the stagnation point where the flow comes to a regional halt and separates along the airfoil. See the different colors on top and bottom of the airfoil? The top has a slightly warmer color compared to the bottom – meaning local velocity in that top region is higher than that on the bottom. If you plug Bernoulli's equation here, clearly the bottom has slightly higher pressure. Meaning? The airfoil will be 'pushed' upwards, in other words, lift!



Look at the marked vectors, they are hitting the airfoil straight. Above that, see how upwardly skewed the vectors are, downwardly below. This is a picture of that stagnation zone – zoomed in. Now, even a child will know exactly where the flow is hitting the airfoil straight, and then splitting into two flows, one at the top and the other at the bottom.

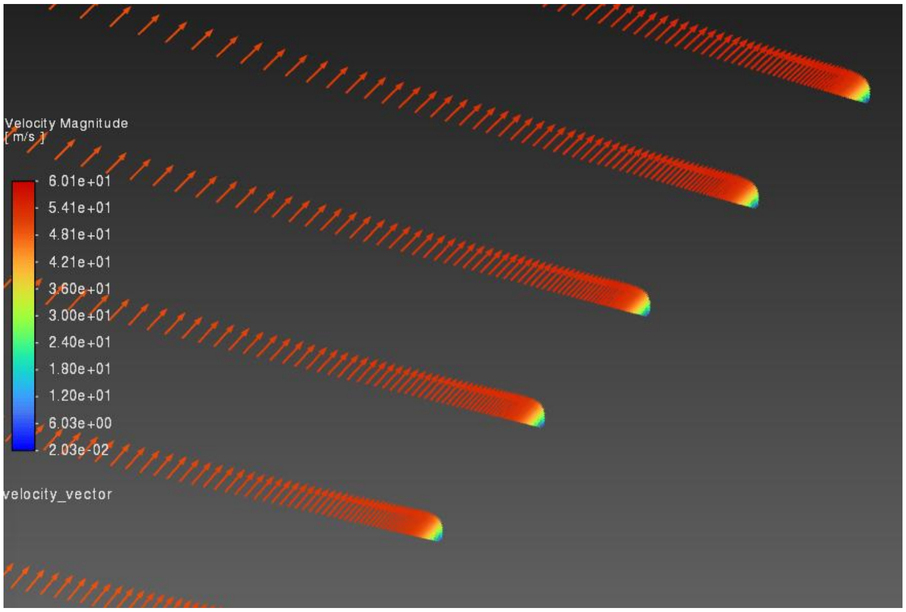
You're welcome. Go tell your mommy.

Now, if we zoom out that upper and lower part to understand the delta, we can clearly see how the velocity is acting in their respected regions.

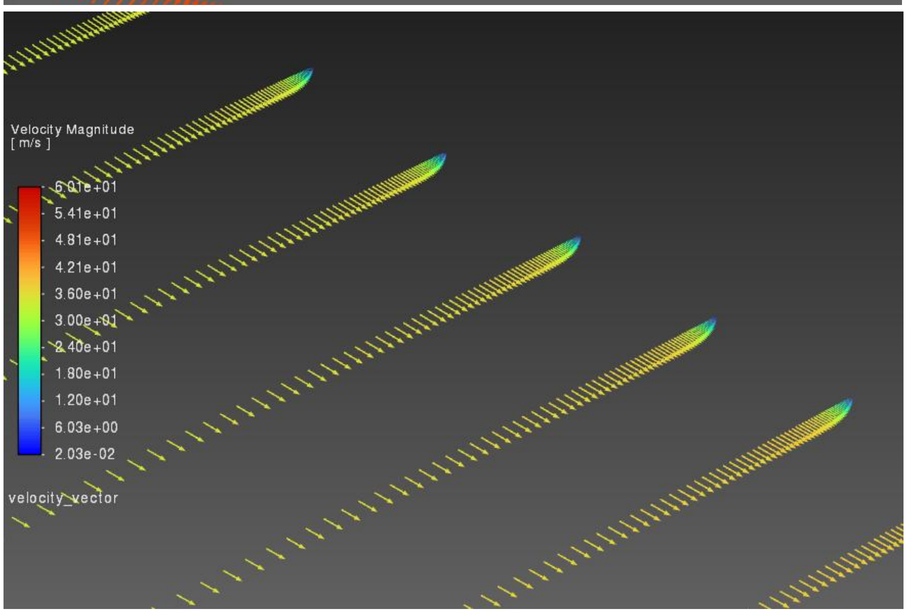


You can clearly see the velocity difference, with the stagnation bubble splitting the flow.

Zooming in, you can see how fluid is behaving in the leading edge, with different velocity in the top and the bottom.



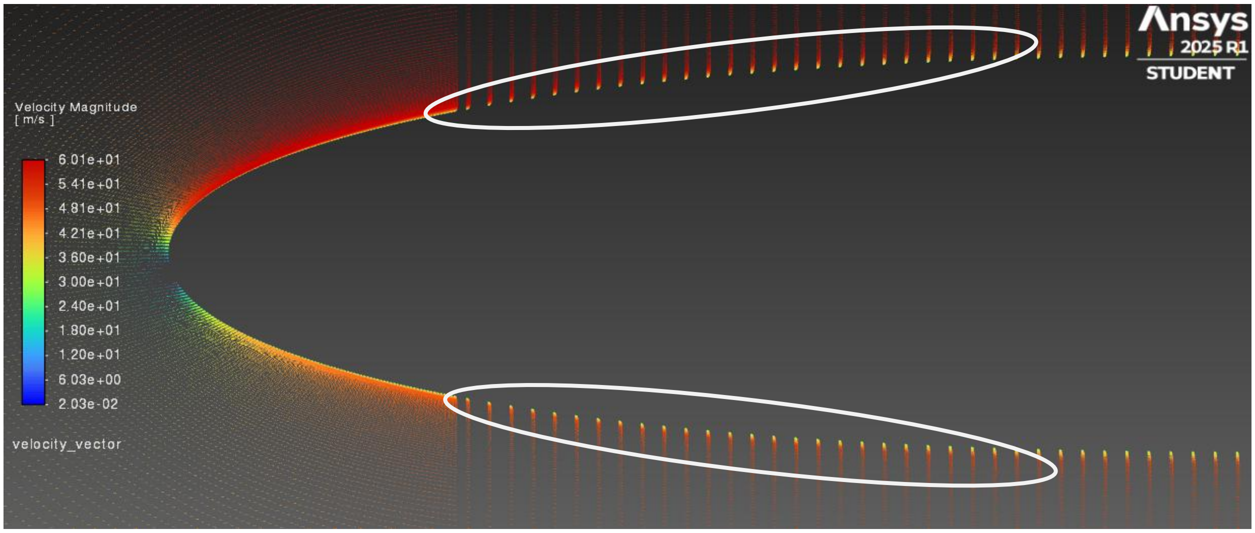
Top surface layer, higher velocity going upwards and around



Bottom surface layer, lower velocity going downwards and around

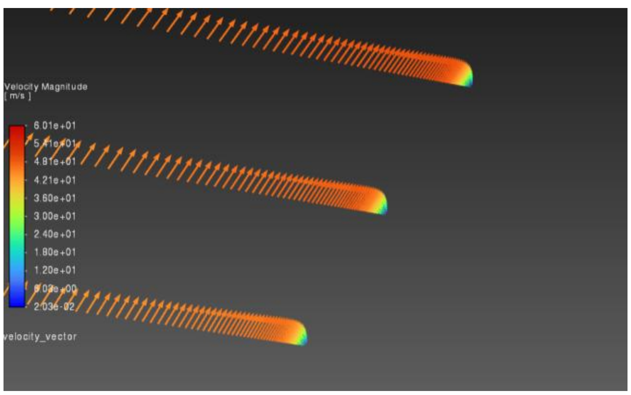
Notice anything else?

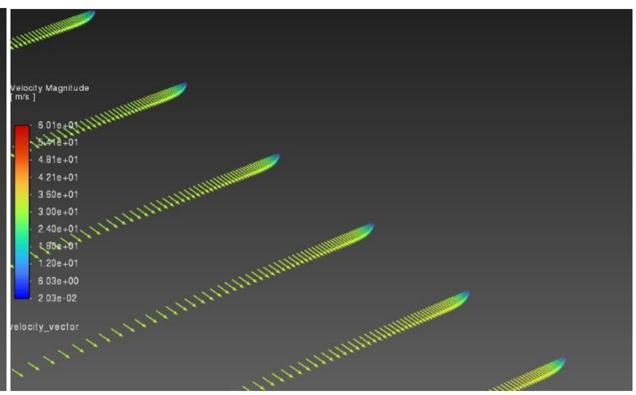
Let me show you, by framing it a little differently.



See the “Match-stick” like pattern with the same color gradient on both top and bottom?

Meaning that, around the airfoil, a very thin layer has been formed with fluids exerting the exact same velocity, regardless if they are on the top or bottom layer. They are exactly the same, exerts the same pattern at the leading edge. It’s just too congested at this zoom level. But why not, see the pictures below –





Look for those blue patches at the very start of the vector fields. Yeah, you’re right, that’s boundary layer – ‘no slip condition’ regardless if it’s on the top or bottom surface.

Bernoulli, Navier and Stokes would be spinning in their graves!

Omar Saif
CFD Enthusiast
July 12, 2025