

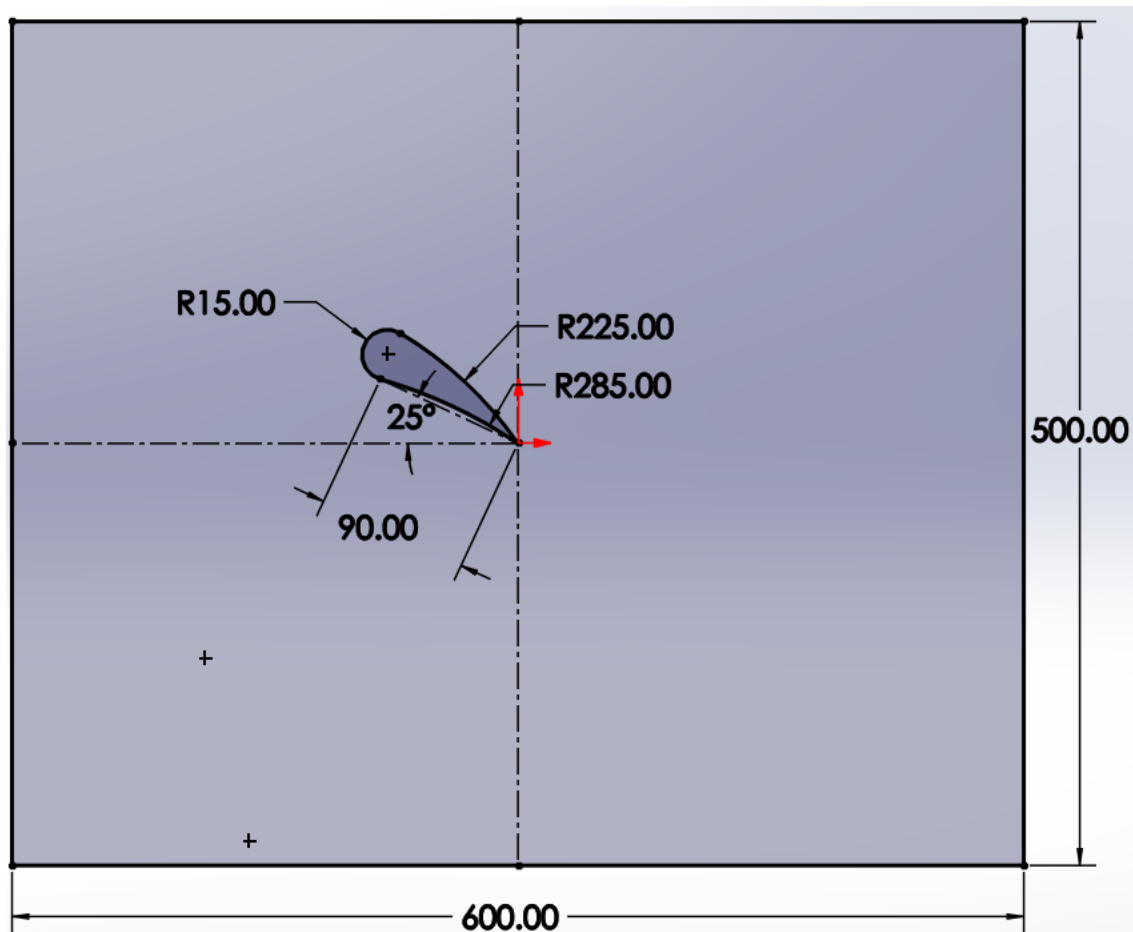
Geometry:

Figure 1. Dimensioned Sketch of Airfoil with Surrounding Box of Air. Besides the angle dimension, all dimensions are in mm.

To generate the above airfoil geometry on Solidworks I used 3 arcs to form the surface of the airfoil: one on the bottom edge of the wing, one on the leading edge of the wing, and one on the top edge of the wing. The arc at the leading edge of the wing is tangent at the contact points with the two other arcs. Adding the 90 mm length of the bottom edge of the wing and the 15 mm radius of the leading edge arc, the total chord-length is found to be 105 mm.

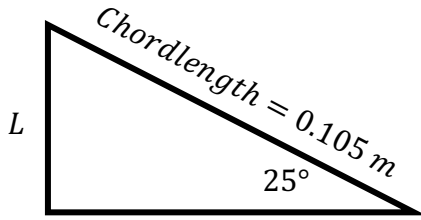
Boundary Conditions:

To achieve a velocity magnitude at the inlet boundary that resulted in a Reynolds number between 100 and 1000 for the airfoil, the velocity magnitudes for a Reynolds number of 100 and 1000 were calculated and then a value was selected for the velocity magnitude that fell between the two calculated values.

$$Re = \frac{L \times u_{avg}}{\nu} \rightarrow u_{avg} = \frac{Re \times \nu}{L}$$

The kinematic viscosity ν of the air was approximated as $\nu = 1.5 \times 10^{-5} \text{ m}^2/\text{s}$.

The characteristic length L of the airfoil was approximated using the projected length of the airfoil normal to the incoming air.



$$\sin 25 = \frac{L}{0.105} \rightarrow L = 0.105 \sin 25 \approx 0.0444 \text{ m}$$

$$\frac{100 \times (1.5 \times 10^{-5} \frac{\text{m}^2}{\text{s}})}{0.0444 \text{ m}} \leq u_{avg} \leq \frac{1000 \times (1.5 \times 10^{-5} \frac{\text{m}^2}{\text{s}})}{0.0444 \text{ m}}$$

$$0.0338 \text{ m/s} \leq u_{avg} \leq 0.338 \text{ m/s}$$

A nice round value of $u_{avg} = 0.1 \text{ m/s}$ was selected from this range to be used as the inlet velocity boundary condition.

Creating Meshes:

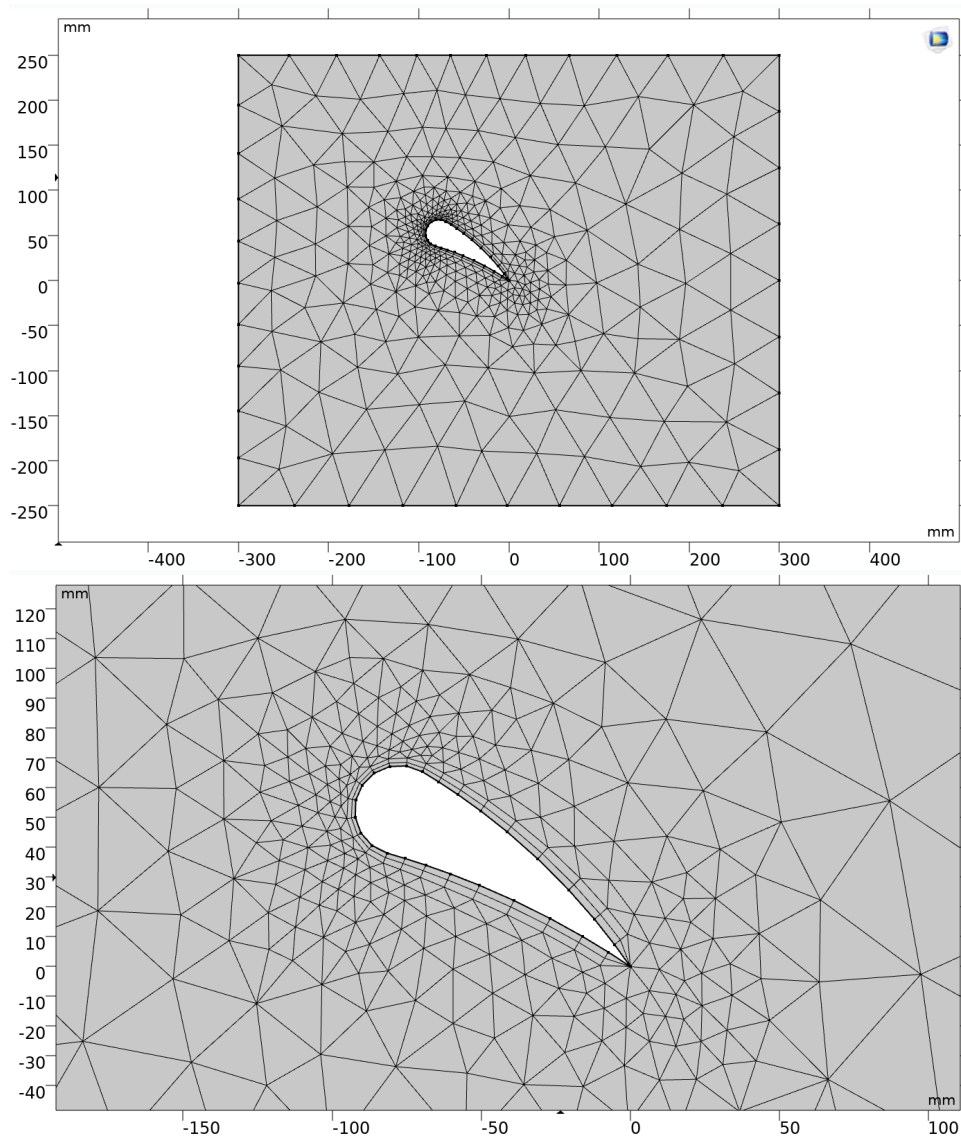


Figure 2. “Coarser” Mesh Size. Entire domain is shown at top and a magnified view near the airfoil surface is shown at bottom.

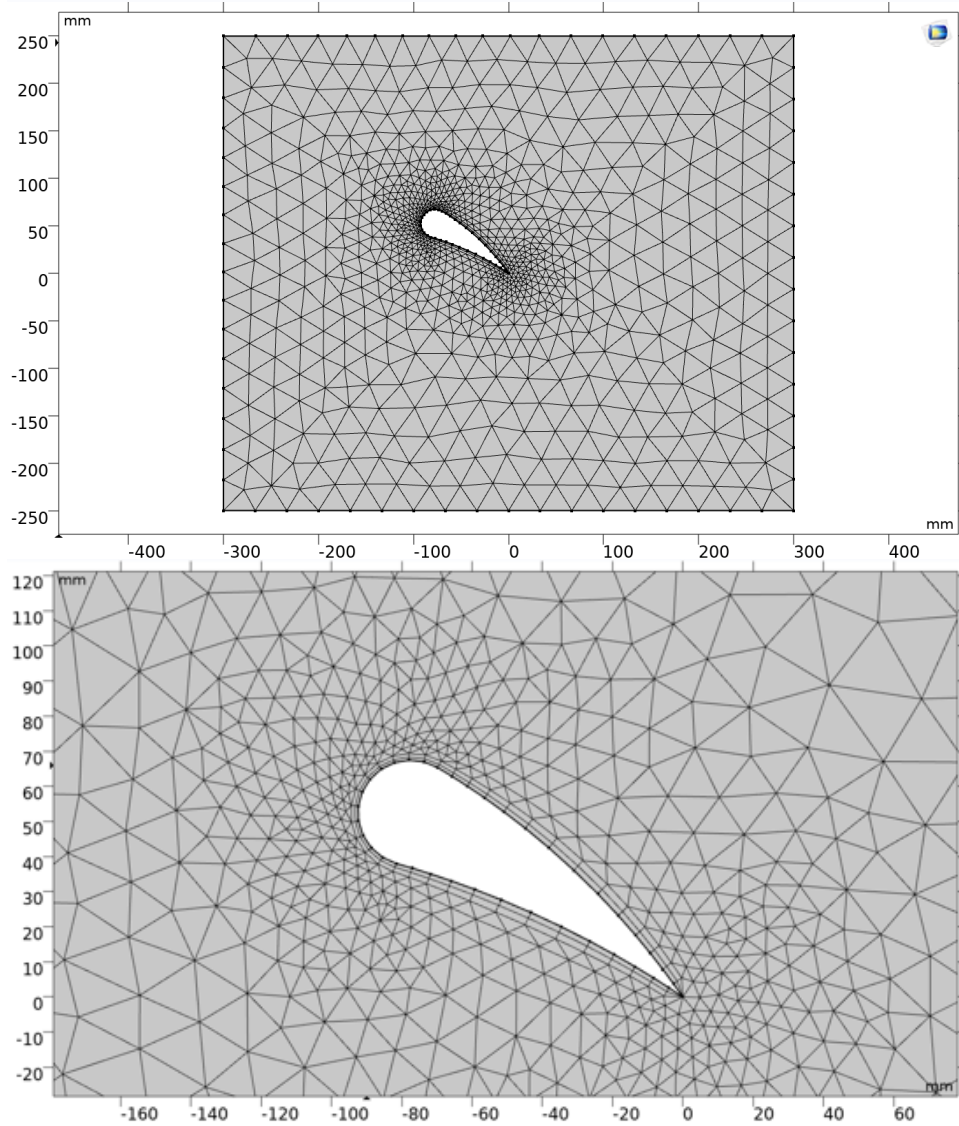


Figure 3. “Normal” Mesh Size. Entire domain is shown at top and a magnified view near the airfoil surface is shown at bottom.

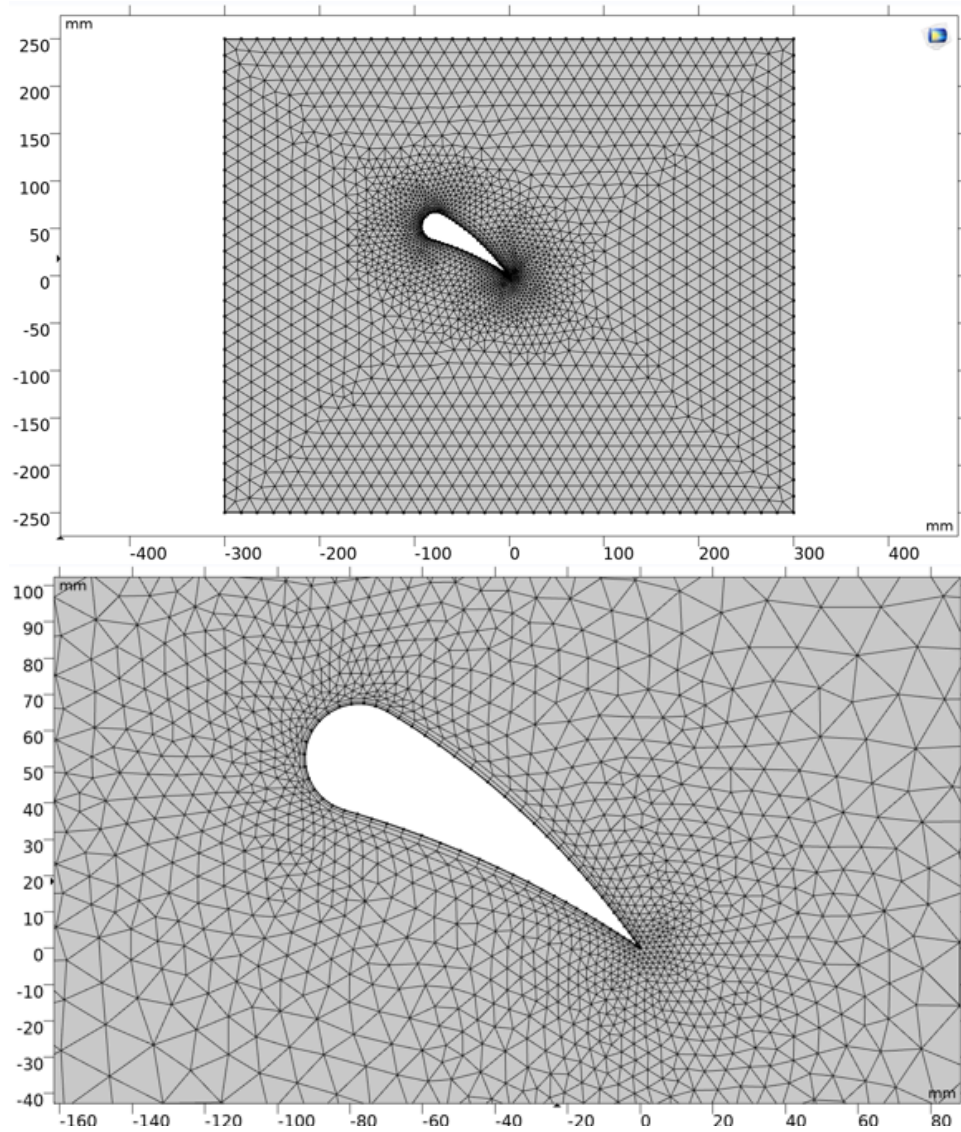


Figure 4. “Finer” Mesh Size. Entire domain is shown at top and a magnified view near the airfoil surface is shown at bottom.

Solution Computation:

Computation time using “normal” mesh size: 4 seconds

Note: During later trials this computation time dropped to 2 seconds. It is unclear why, but one possibility may be that Comsol is storing the study results from previous simulations. Therefore when asked to rerun the same study or a similar study with a slightly different mesh size, it may be retrieving some already calculated values instead of recalculating the values at all nodes, thereby lowering the overall computation time.

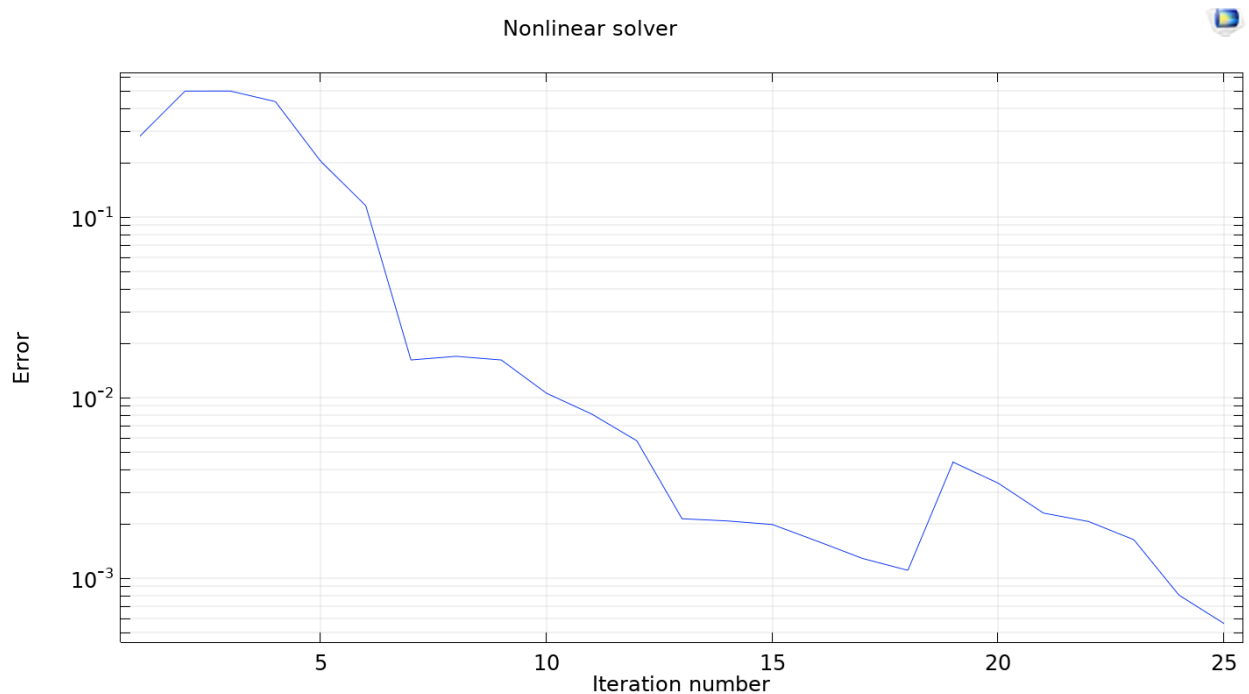


Figure 5. Convergence Plot for Comsol Nonlinear Solver Computing Study Results using “Normal” Mesh Size

Results:

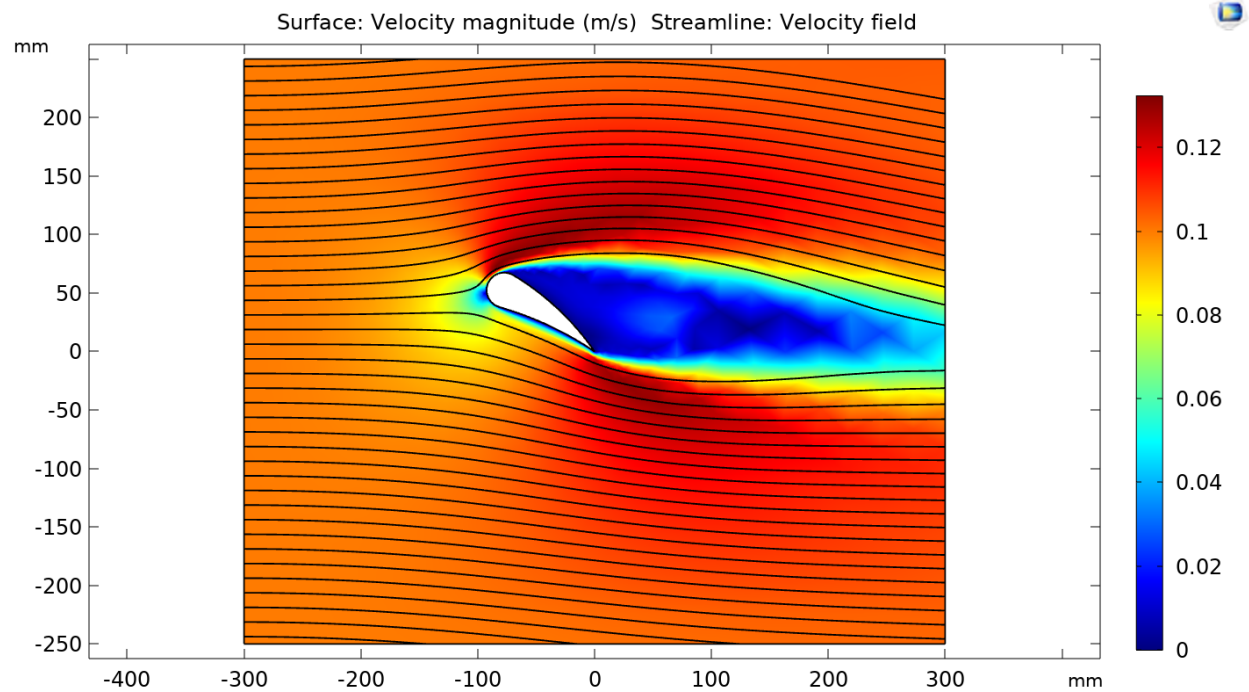


Figure 6. Velocity Color Contour Plot with Streamlines from Simulation Results

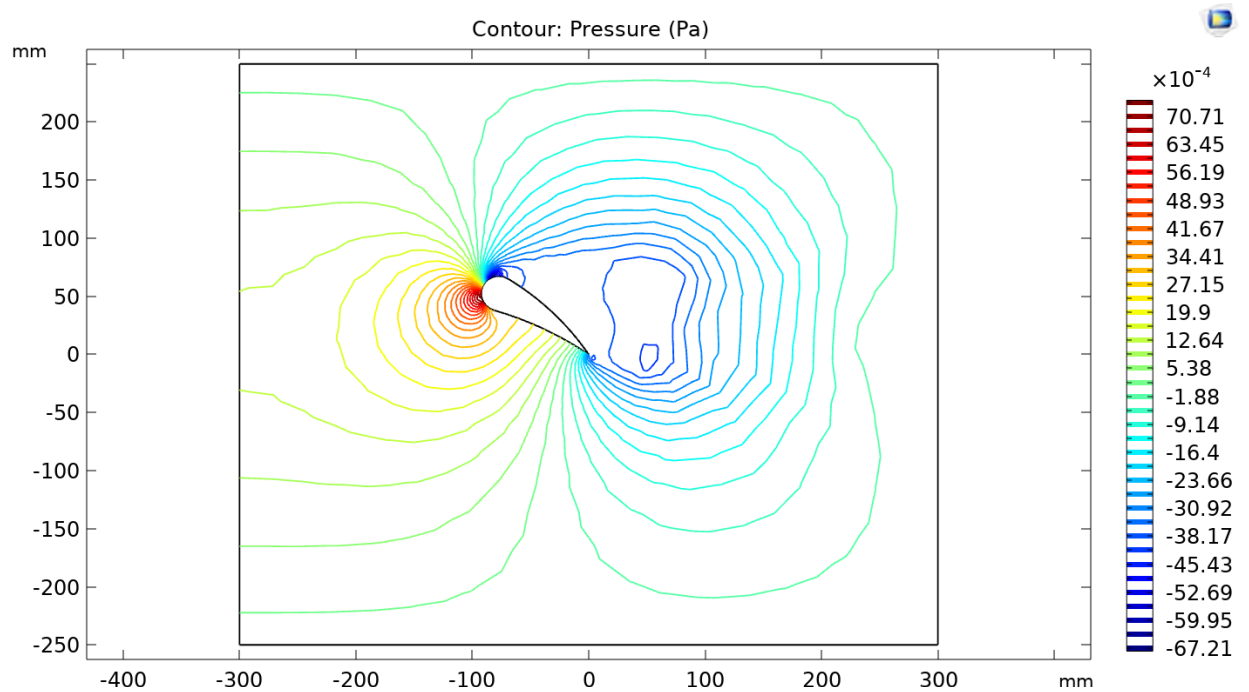


Figure 7. Pressure Color Contour Plot from Simulation Results

Analysis:

Examining Figure 6, the flow field is fastest in the regions shedding off of the leading edge and trailing edge of the airfoil, reaching approximately 0.13 m/s at a maximum. The occurrence of these fast regions can be explained by looking at the pressure gradient at the leading edge and trailing edge of the airfoil in Figure 7. Around those points, the pressure is decreasing quickly as it moves over the airfoil causing the flow in those regions to accelerate.

The flow field is slowest near the airfoil surface, in the region behind the airfoil (to the right), and at a point near the leading edge of the airfoil where the streamlines split to move around the wing, reaching 0 m/s at a minimum. Since a no-slip condition was put into the simulation for the airfoil surface, it makes sense that the flow near the airfoil surface is slow as the airfoil is fixed in place (has zero velocity). It is also intuitive that the flow behind the airfoil is slow as the airfoil blocks and reroutes streamlines that were travelling level with the airfoil. The circular region of slow flow at the leading edge of the airfoil where the streamlines split to move around the wing is a little more interesting to analyze. The region appears to be directly adjacent to the part of the airfoil that is fully perpendicular to the flow. Therefore all incoming flow would have to stop moving in the horizontal direction at that point as it cannot pass through the airfoil. Instead, the flow would be slowed and rerouted vertically around the airfoil.

While most of the flow is directed primarily in the horizontal direction, especially far from the airfoil, flow breaking across the leading edge of the wing and pulling in behind the wake of the airfoil has a significant vertical component. The significant vertical component of the flow at the leading edge of the wing can be explained simply by the fact that air cannot pass through the airfoil. Instead of moving horizontally, the flow must either move up and around or down and around the airfoil. On the other hand, the vertical component pulling the flow together behind the wake of the airfoil is explained by the negative vertical pressure gradient acting inwards on either side of the wake.

Aerodynamic Forces:

Simulation Author	Lift Force (N/m)	Drag Force (N/m)
Professor Kemmerling	1.55×10^{-4}	1.41×10^{-4}
Me (Rónán Gissler)	5.40×10^{-4}	4.47×10^{-4}

Table 1. Comparison of Aerodynamic Forces Between Simulations Using Different Airfoil Geometries

My aerodynamic forces were approximately three times larger than the aerodynamic forces returned by Professor Kemmerling's simulation. These significantly larger aerodynamic forces could be the result of my airfoil camber or slightly higher angle of attack which would result in a larger portion of the airfoil area facing the incoming flow. Thinking about the equations for lift and drag, the larger airfoil area facing the flow would lead to larger aerodynamic forces. Another way to think about it would be that when a larger portion of the airfoil area is facing the incoming flow, the average pressure on the underside of the airfoil would increase thereby also leading to increases in aerodynamic forces. Nonetheless, it seems reasonable that these differences in aerodynamic forces are due to differences in airfoil geometry and not a mistake on my part as the results are of the same order of magnitude.

Grid Convergence:

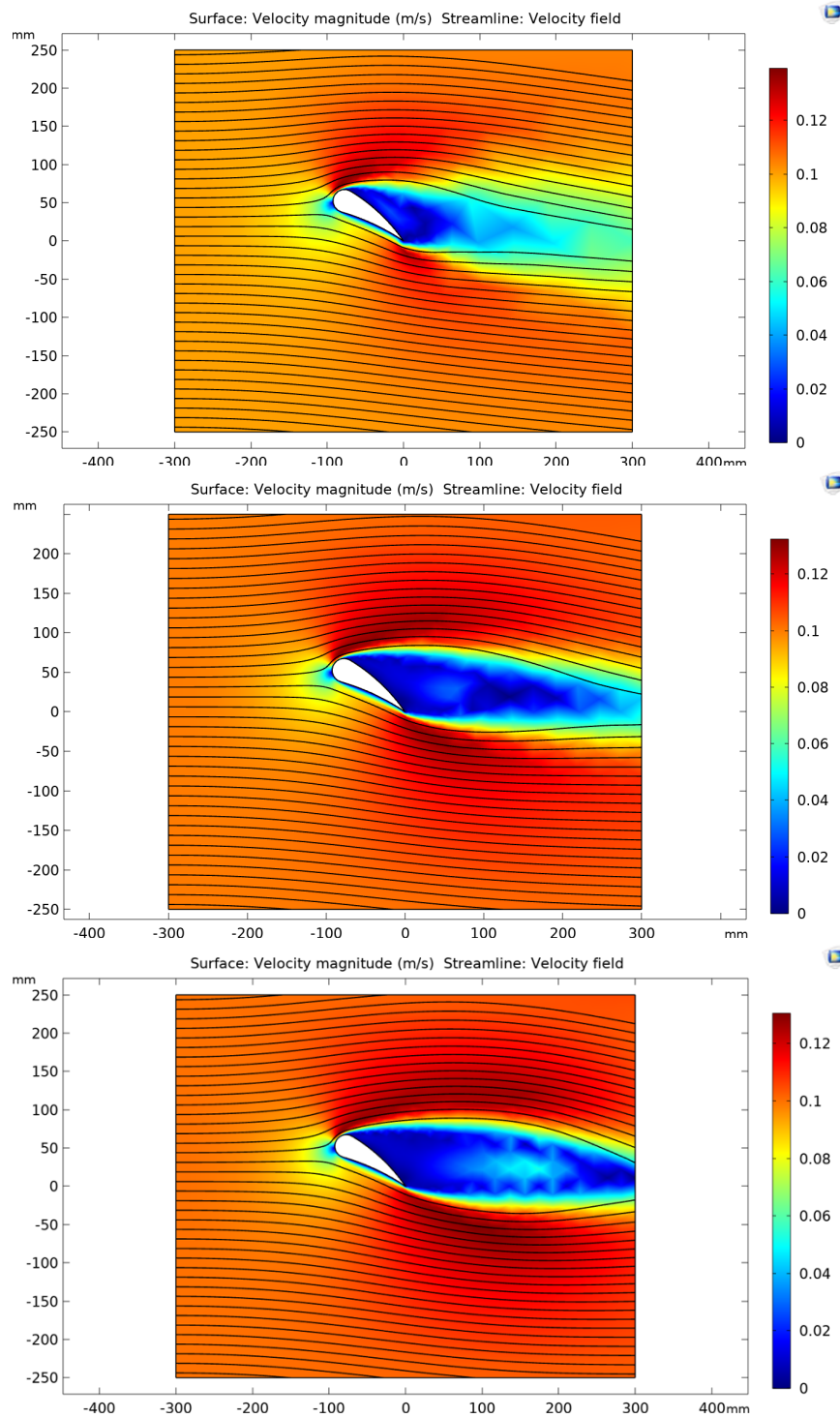


Figure 8. Velocity Color Contour Plots Running Simulation with Different Mesh Sizes. From top to bottom: “coarser”, “normal”, and “fine”.

The velocity plots shown in Figure 8 for different mesh sizes appear to mostly agree with one another. It appears that the finer the mesh, the longer the region of high velocity flow shedding off of the leading and trailing edge of the airfoil. There is also some significant variation in the velocity plots in the wake region behind the airfoil. It seems that the finer the mesh, the tighter and less diffuse the wake becomes. However, in the bottom “fine” mesh size plot a region of fluid motion also appears in the middle of the wake that didn’t seem to be present in the other velocity plots.

Mesh Size	Computation Time (s)	Domain Elements	Boundary Elements
“coarser”	2	767	67
“normal”	2	1791	107
“fine”	4	3197	149

Table 2. Comparison of Mesh Sizes

Note: The “Fine” mesh size was used instead of the “Finer” mesh size since the simulation did not converge when the “Finer” mesh size was applied.

When choosing a mesh, finer meshes provide higher resolution snapshots of the behavior of the fluid and are therefore better unless the high resolution is at the expense of a significantly longer computation time. When more points are available at which flow solutions can be determined, a more wholesome understanding of the flow scenario is possible. Of the meshes ran, “fine” was the finest as it had the most mesh elements if we look at Table 2. Consequently, I would apply the “fine” mesh when running the simulation. Even if the solution accuracy is only increased slightly, the additional computation time of 2 seconds compared to other mesh sizes isn’t a concern. I would consider using even finer meshes if the solver was able to converge for finer meshes. Since the solver cannot return simulation results for any meshes finer than “fine”, the results provided by the “fine” mesh size will provide a “good enough” solution.

Reflection:

Overall, following the procedure for this lab was relatively straightforward. The one challenge I did encounter with Comsol was the solver failing to converge when I used the “finer” mesh size. However, just decreasing the mesh size to “fine” solved this problem. There was also some weirdness with inconsistent computation times when running the simulation using the “normal” mesh size. I’m still not totally sure why this is. I’d be interested to learn under what circumstances these simulation results could be applied. Based on Professor Kemmerling’s comment in the canvas discussion regarding translating these results to 3D, I would imagine that the results of the simulation would be better approximations for long, high aspect-ratio airfoils where the inconsistencies at the edge of the airfoil would have less of an effect on the overall airfoil performance.