Computational Fluid Dynamics Lab

Brandon Lim
10/21/2024

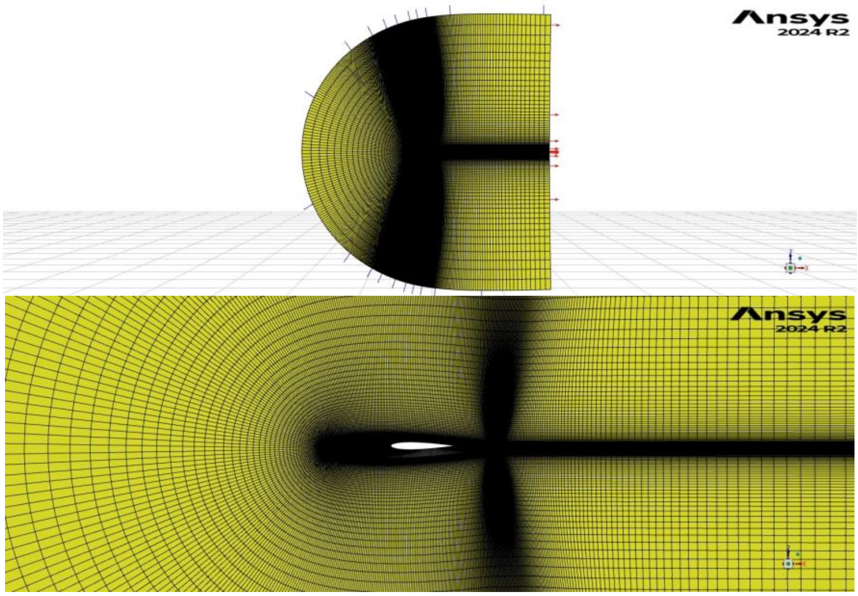

Figure 1a. Computational mesh of the NACA0012 air foil at 400 nodes. Both images are for the case of 400 nodes with an angle of attack of zero degrees. The top image is the entire domain of the computational fluid dynamics model while the bottom image is a zoomed in domain of the airfoil.

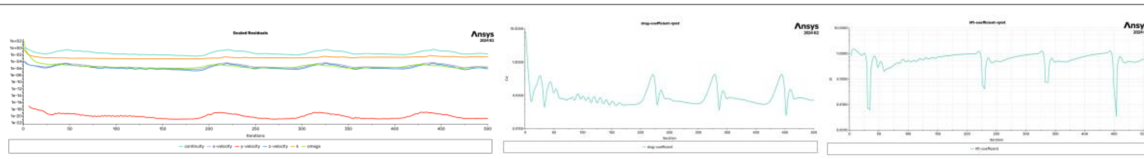

Figure 1b. Three convergence plots of the NACA0012 Airfoil at an angle of attack of 12-degrees. The left plot is the plot of the residuals continuity, x-velocity, y-velocity, k, and omega on the y-axis and number of iterations on the x-axis. The middle plot is the plot of drag coefficient on the y-axis and number of iterations on the x-axis. The right plot is the plot of lift coefficient on the y-axis and number of iterations on the x-axis.

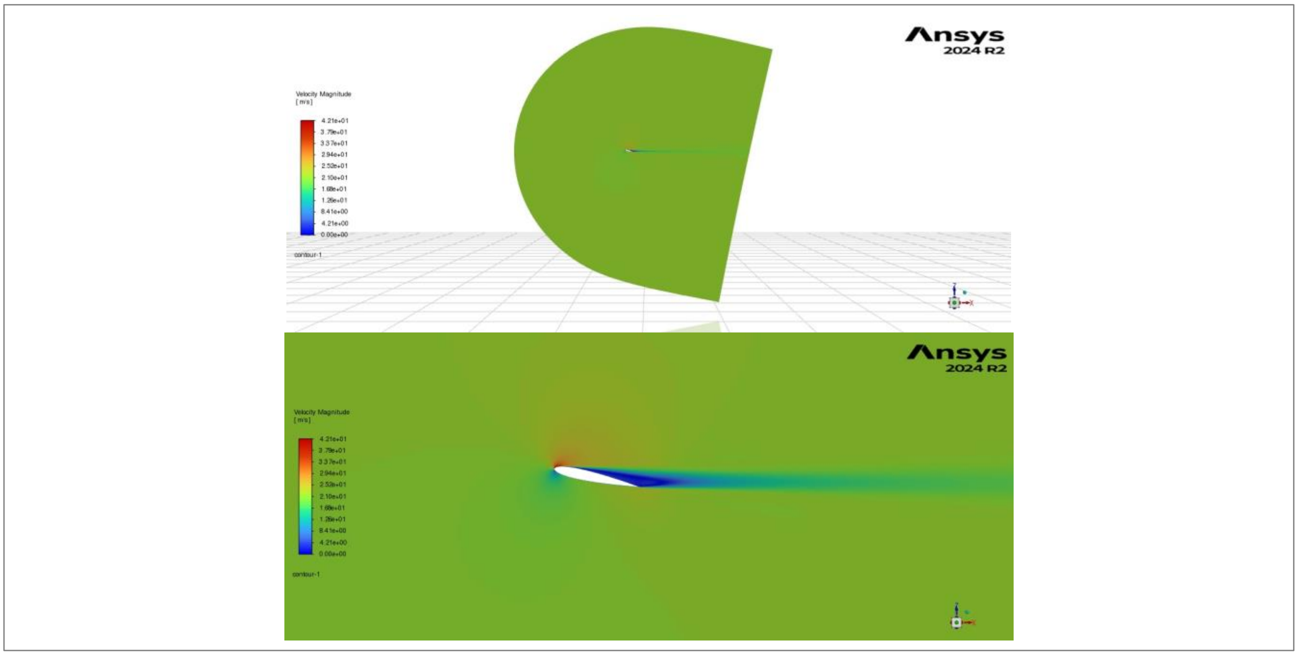

Figure 1c. Velocity magnitude contour plot of an NACA0012 air foil at an angle of attack of 12-degrees. The top figure is of the entire computational domain while the bottom figure is a zoomed-in picuture near the airfoil. In this image, flow is moving from left to right.

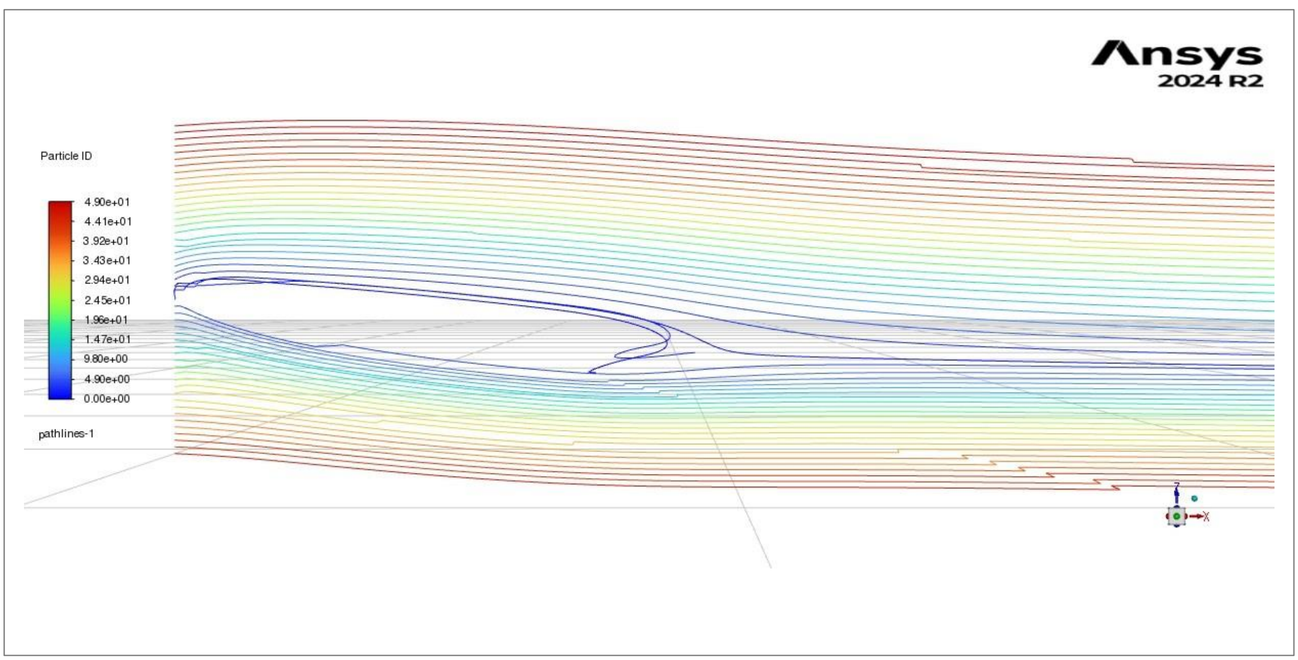

Figure 1d. Pathlines near the NACA0012 air foil at an angle of attack of 12-degrees. In this image, flow is moving from left to right.

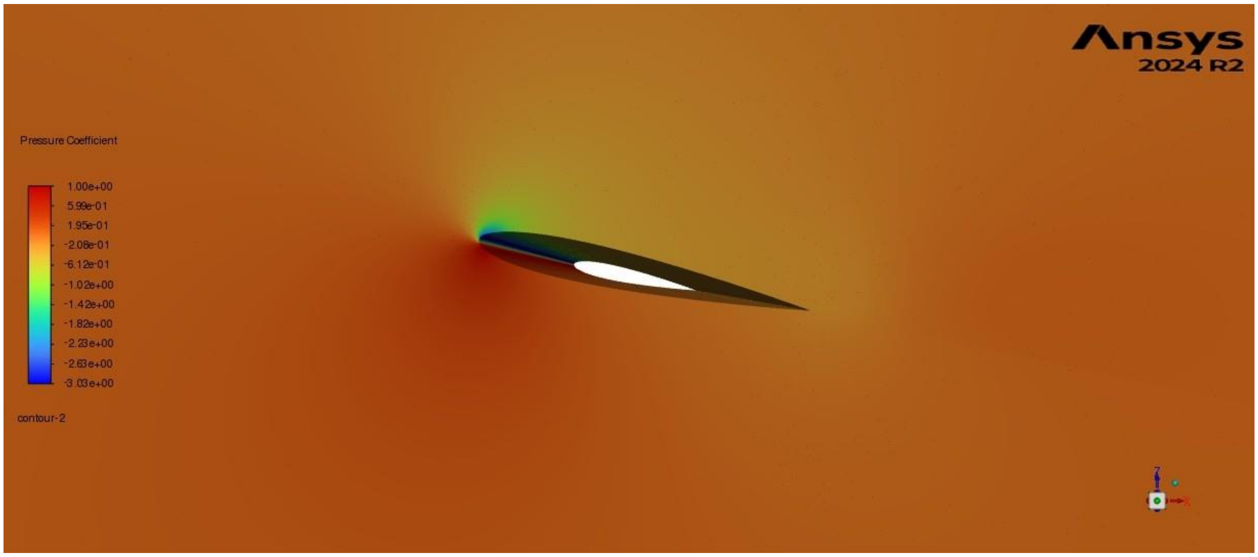

Figure 1e. Pressure coefficient contour plot near the NACA0012 air foil at an angle of attack of 12-degrees. In this image, flow is moving from left to right.

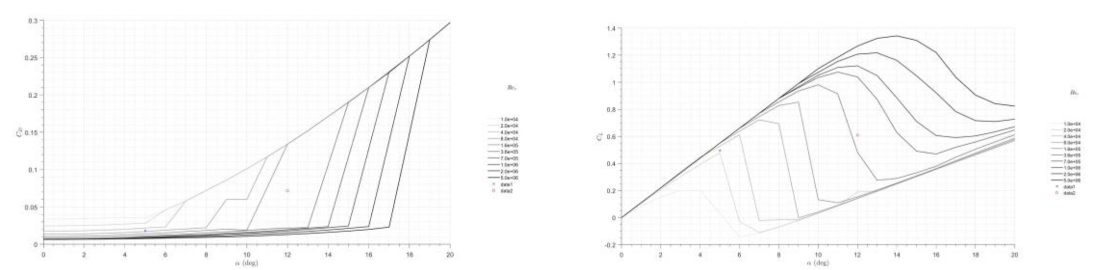

Figure 1f. Plots of drag and lift coefficient on the y-axis and angle of attack in degrees on the x-axis for a NACA0012 air foil. The left plot is the plot of coefficient of drag and the right plot is the plot of coefficient of lift. The blue markers represent the simulated results for an angle of attack of 5 degrees, and the red markerse represent the simulated results for an angle of attack of 12 degrees.

Table 1g. Lift and drag coefficient for the NACA0012 air foil at different angles of attack. The simulated, experimental, and percent difference between simulated and experimental values are reported based on the angle of attack at similar Reynolds numbers.

α (deg)	C_D			C_L		
	Simulation	Experiment	ε (%)	Simulation	Experiment	ε (%)
5	0.017857103	0.014	27.55%	0.49614851	0.55	9.79%
12	0.071687275	0.134	46.50%	0.60869602	0.1533	297.06%

Short-Answer Questions

2a. *Discuss how the CFD simulation results vary with mesh spacing. Use the results from the tutorial to answer this question. Be sure to be specific in your response. [4–6 sentences]*

Computational fluid dynamics simulation results can vary its accuracy based on the discretization of its domain or the mesh spacing. Mesh spacing should be fine enough to ensure simulation results do not vary changes in fluid properties significantly across the entire volume elements in the mesh. For example, in the AnsysFluent tutorial, an independent study of Drag Coefficient and Lift Coefficient versus number of mesh elements was performed. From the results, we can see that at a lower number of mesh nodes, the coefficient of drag and lift are both relatively further away from the convergence of the coefficient of drag and lift values when the number of mesh nodes are increased. When the mesh nodes in the experiment are increased to 100, 400, and 800 the coefficient of drag converges to around 0.0294 while the coefficient of lift converges to around 0.9200. Now looking at the mesh node of 50, we can see that the coefficient of drag is around 0.035 and the coefficient of lift is around 0.860 which are values relatively innacuracte compared to the convergence value at higher mesh nodes.

2b. *Discuss what is "validation" in the context of CFD simulations and why it is necessary. [2–4 sentences]*

Validation in the context of CFD simulation is the comparison of simulation results against real experiemtal data under the same conditions. This is important to do to ensure that the conditions at which the simulation was run under are accurate for reliable analysis. By comparing the simulated results against real experimental data, discrepancies can be noted and potentially reveal errors in the simulation like number of mesh nodes or number of iterations. If discrepancies in the validation are small, engineering judgement can be used to declare if discrepancies are acceptable for simulation analysis.