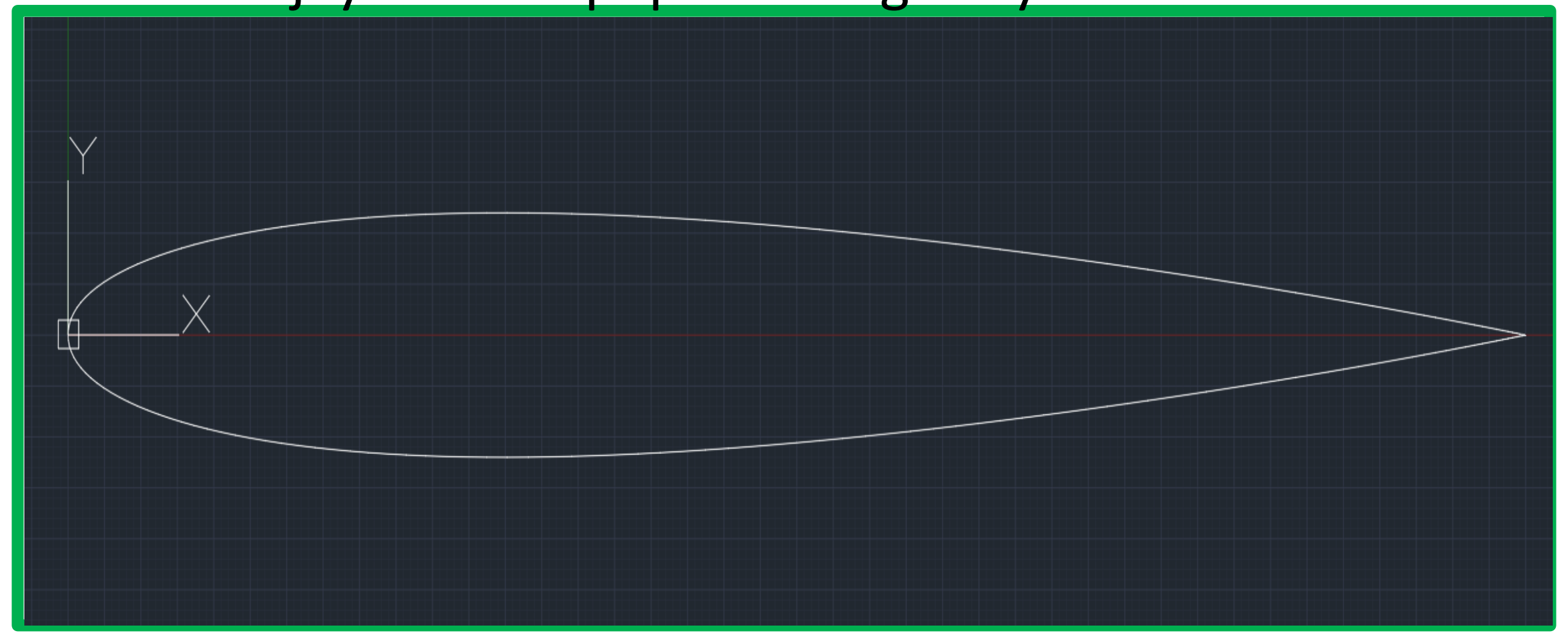
Numerical Study of Wing Morphing Aerodynamic Properties in Low Reynolds Number Flow

Benjamin Diaz Villa, Dr. Ralph Metcalfe
Department of Mechanical Engineering, University of Houston

UNIVERSITY of HOUSTON

Introduction

Wing morphing is inspired by the flexible biological mechanisms that many birds, insects, and aquatic organisms use to travel through air or water. Unlike these creatures, almost all aircraft control their flight by using rigid materials and joints to move the wing ailerons, flaps, slats, and rudders. This poses potential flight control failures due to loose hinges, jammed joints, and slacked pulley cables, just to name a few. In contrast, wing morphing technology, such as that involving shape-memory alloys (SMAs), can be used to even out the aerodynamic pressure so that no one part is over-stressed without having to use rigid, moving parts. SMA's can open up new wing shapes not possible with present technology. Recent studies have shown that wing morphing can improve the lift/drag ratio under turbulent flow condition. However, this technology has not been carefully studied in low Reynolds number flows such as those encountered by drones and unmanned aerial vehicles. This research will determine what morphing deformations are best suited to enhance the efficiency of a drone flying in low Reynolds number flow. However, for much of the summer, the research focused on validating that the ANSYS Fluent code works in generating high-fidelity simulations of flow around airfoil. Thus, I based my validation on Sanjay Mittal's paper on flight hysteresis.



General Cross-Sectional Schematic of the NACA0012 Airfoil

Method of Computation

According to Smits' Introduction to Fluid Mechanics, in order to model the incompressibility of the air flow, the relative velocity must be less than 0.3 Mach and the divergence of the flow velocity must be zero. From the same source, I found that the following simplification of the Navier-Stokes equation models turbulent, time-dependent flow, as shown:

$$\frac{\partial \rho}{\partial t}+\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho u)}{\partial y}+\frac{\partial(\rho u)}{\partial z}=0$$

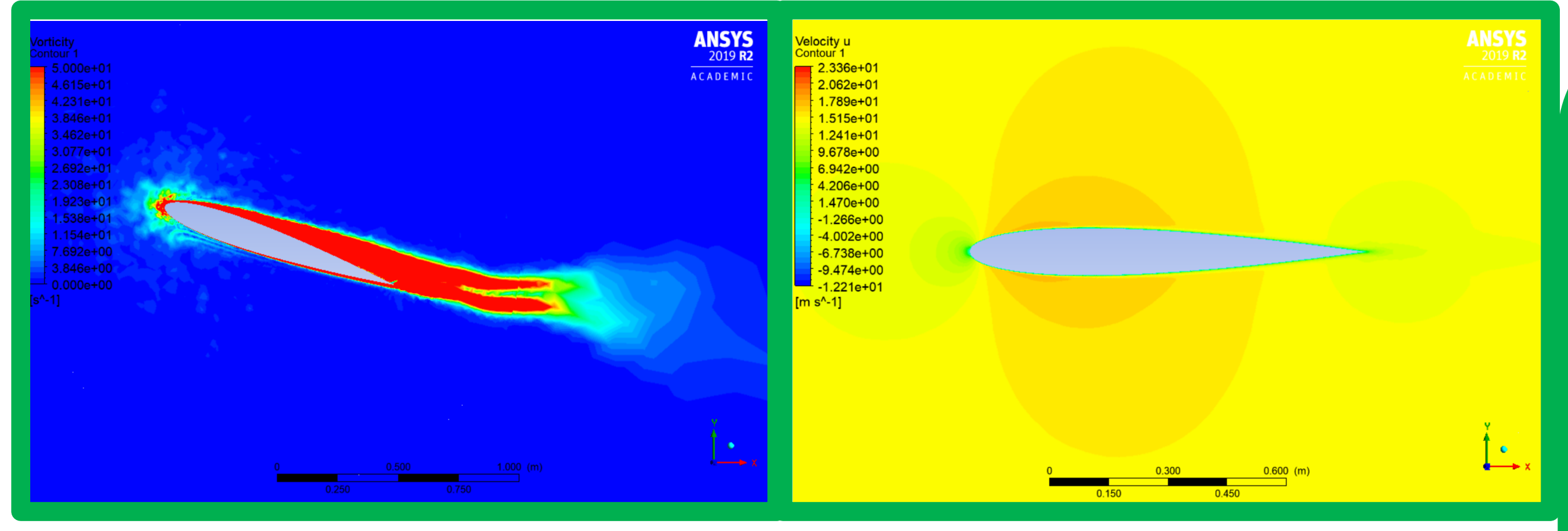
$$\text{X-Momentum: } \frac{\partial(\rho u)}{\partial t}+\frac{\partial(\rho u^2)}{\partial x}+\frac{\partial(\rho uv)}{\partial y}+\frac{\partial(\rho uw)}{\partial z}=-\frac{\partial p}{\partial x}+\frac{1}{Re_r}\left[\frac{\partial \tau_{xx}}{\partial x}+\frac{\partial \tau_{xy}}{\partial y}+\frac{\partial \tau_{xz}}{\partial z}\right]$$

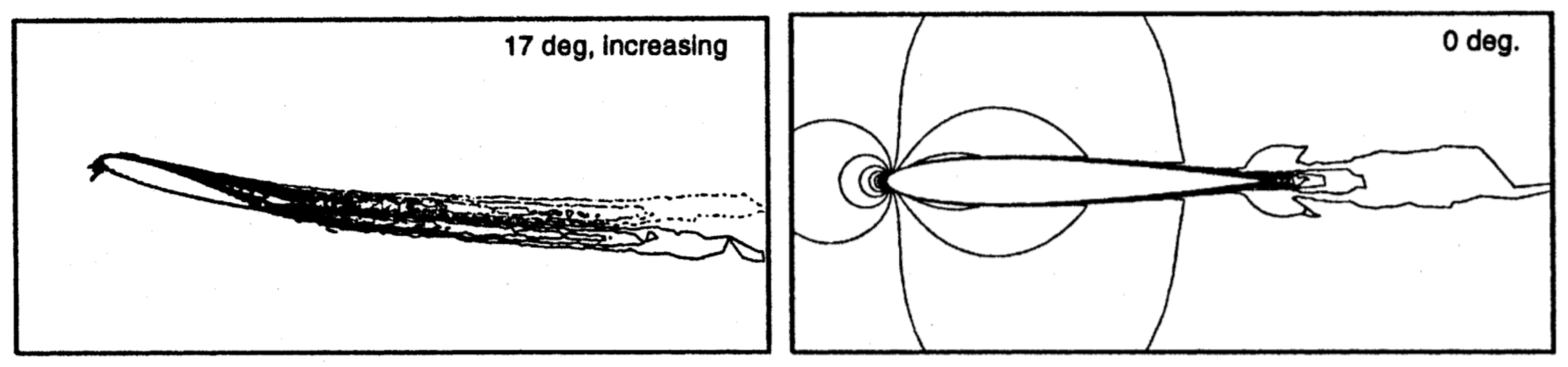
$$\text{Y-Momentum: } \frac{\partial(\rho v)}{\partial t}+\frac{\partial(\rho uv)}{\partial x}+\frac{\partial(\rho v^2)}{\partial y}+\frac{\partial(\rho vw)}{\partial z}=-\frac{\partial p}{\partial y}+\frac{1}{Re_r}\left[\frac{\partial \tau_{xy}}{\partial x}+\frac{\partial \tau_{yy}}{\partial y}+\frac{\partial \tau_{yz}}{\partial z}\right]$$

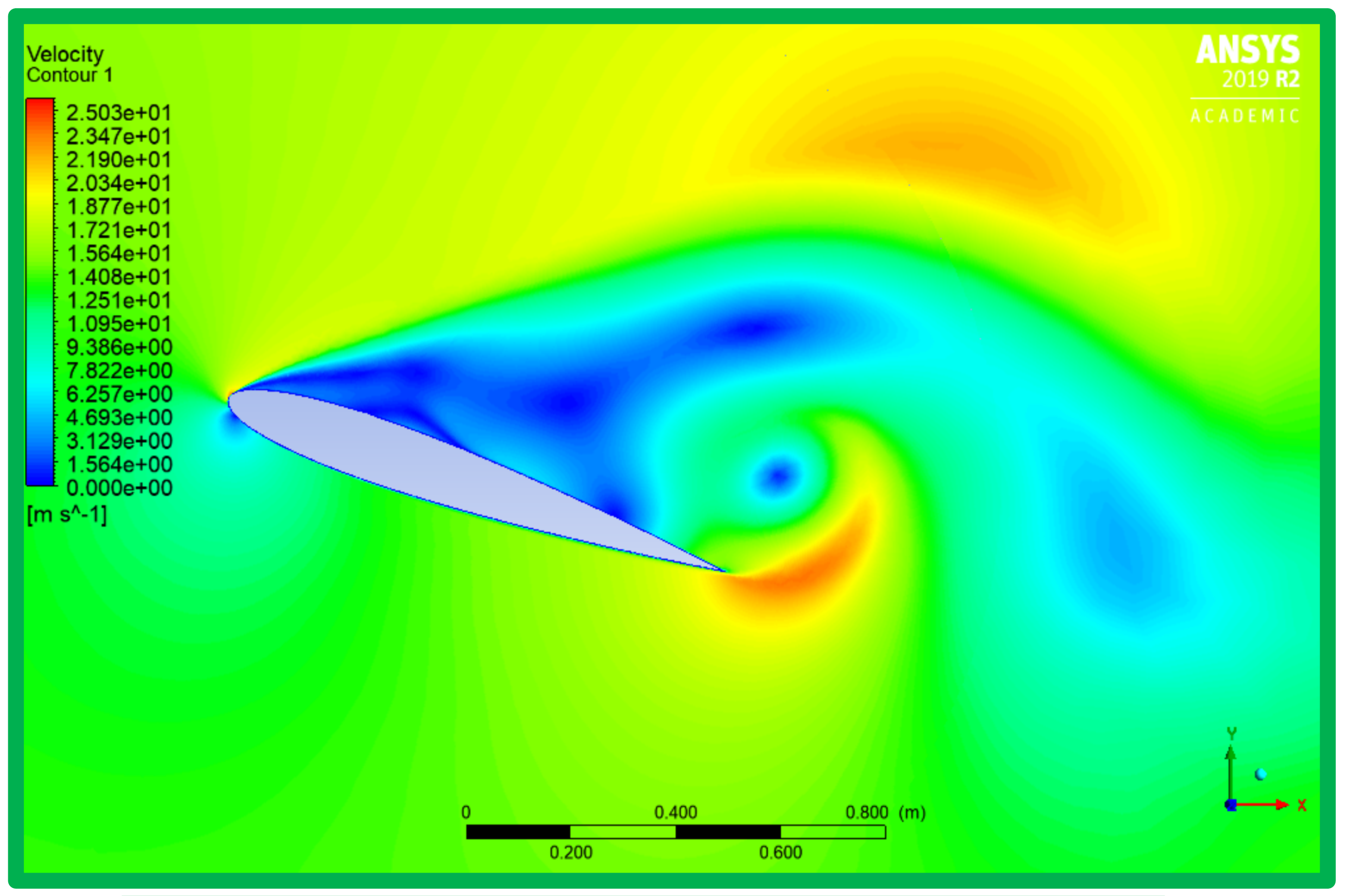
$$\text{Z-Momentum: } \frac{\partial(\rho w)}{\partial t}+\frac{\partial(\rho uw)}{\partial x}+\frac{\partial(\rho vw)}{\partial y}+\frac{\partial(\rho w^2)}{\partial z}=-\frac{\partial p}{\partial z}+\frac{1}{Re_r}\left[\frac{\partial \tau_{xz}}{\partial x}+\frac{\partial \tau_{yz}}{\partial y}+\frac{\partial \tau_{zz}}{\partial z}\right]$$

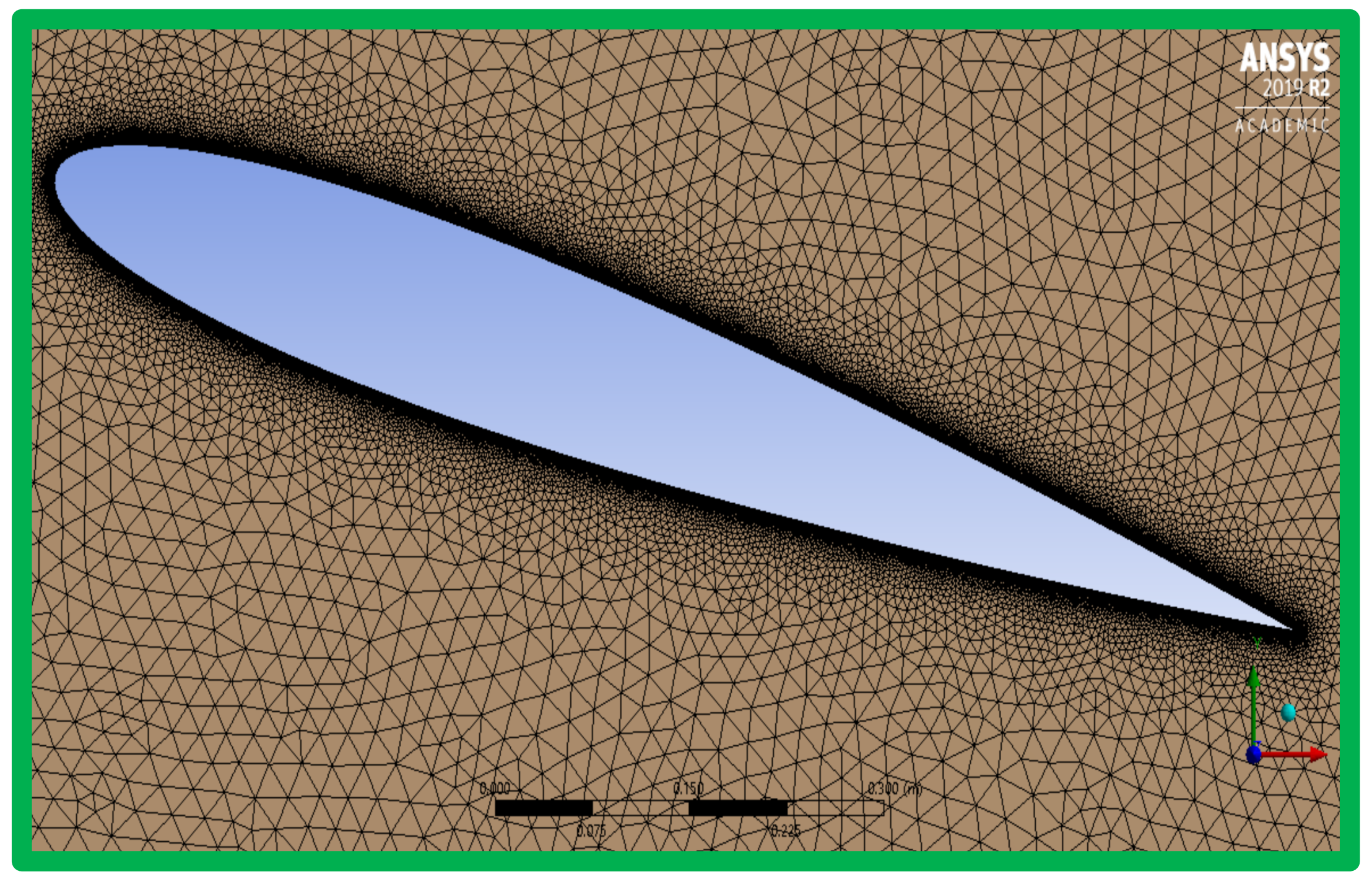
Where u, v, w are the velocity vectors, ρ is the fluid density, τ is the tensor stress, and v is the kinematic viscosity. Because the Navier-Stokes equation could not be solved analytically and the geometries of the discs were complex, I had to use ANSYS Fluent, which implements the finite-volume method to solve the conservation equations. In ANSYS Fluent, I implemented the pressure-velocity coupling by means of the SIMPLE-type fully implicit algorithm. I then approximated the steady solution using second-order implicit method. And finally, I made sure that the solution was second-order accurate in space and time.

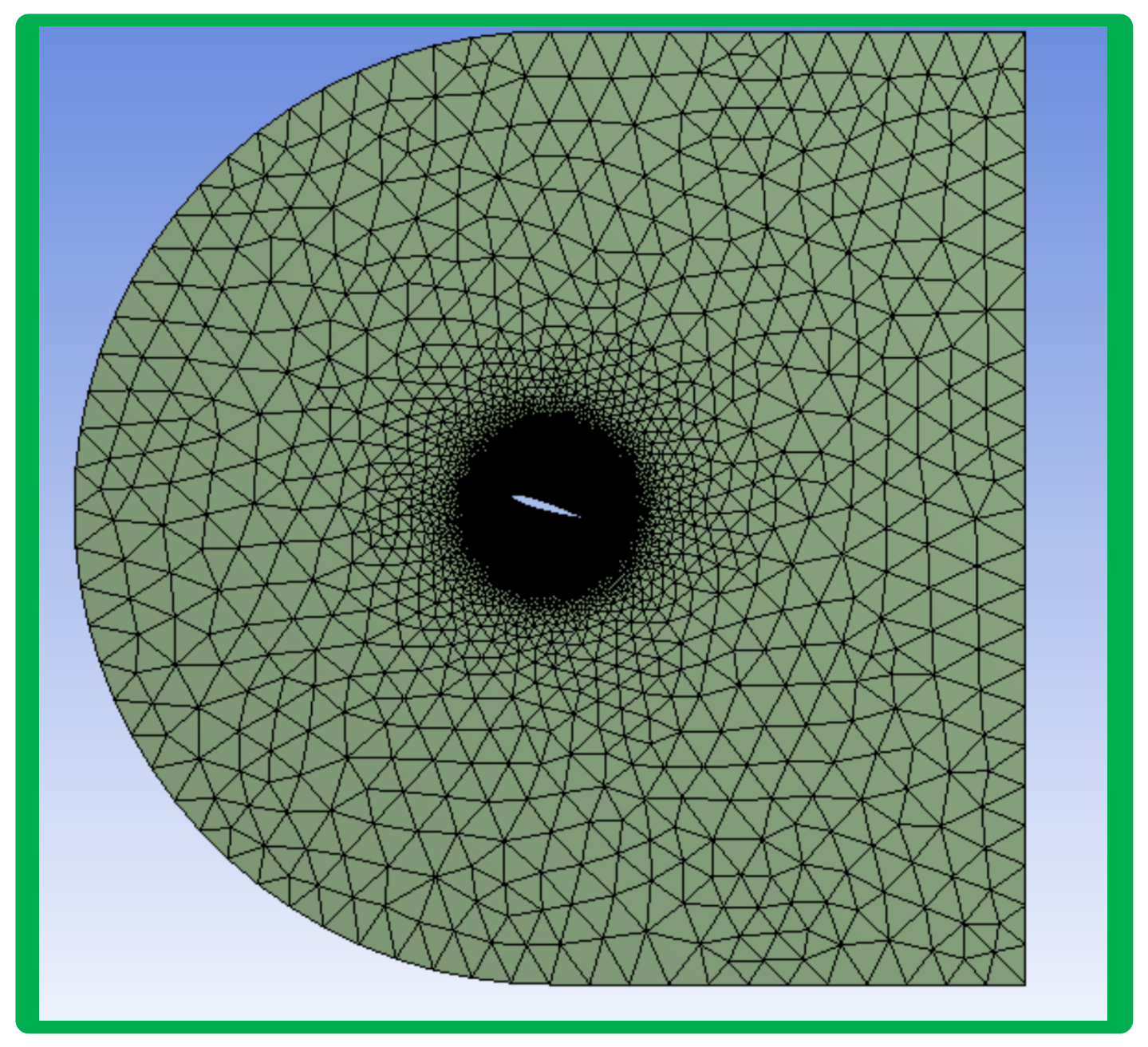
Spalart-Allmaras Model



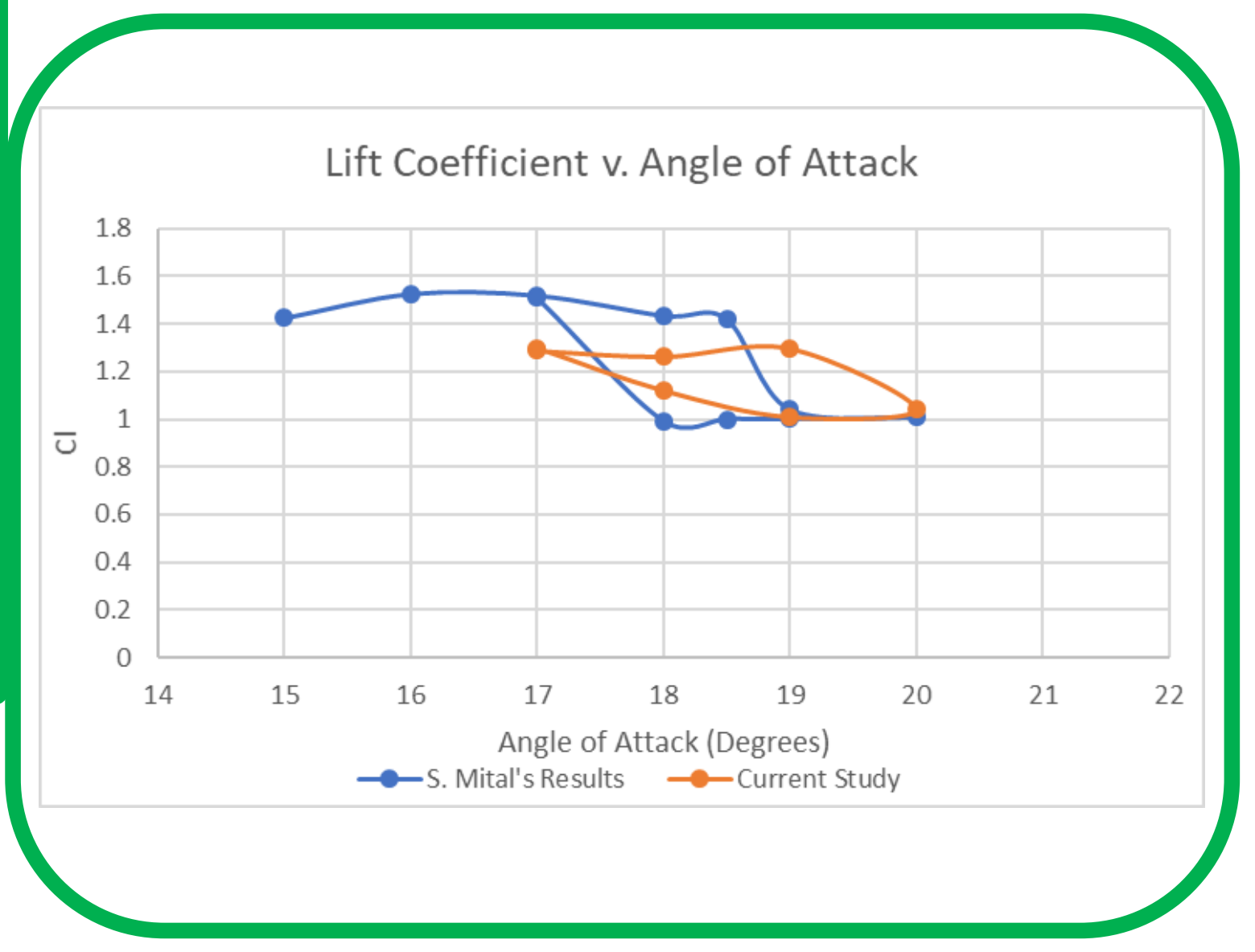


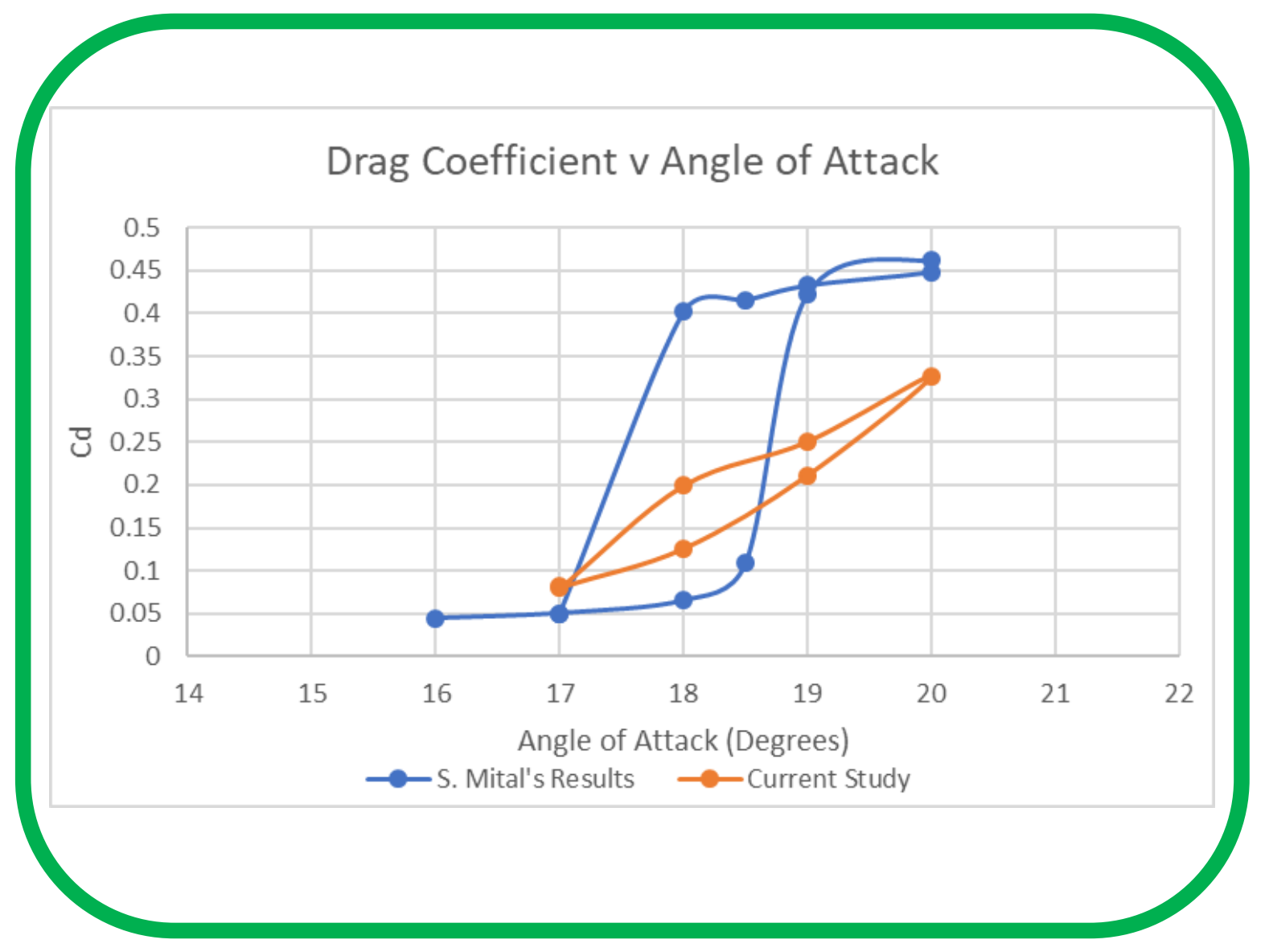


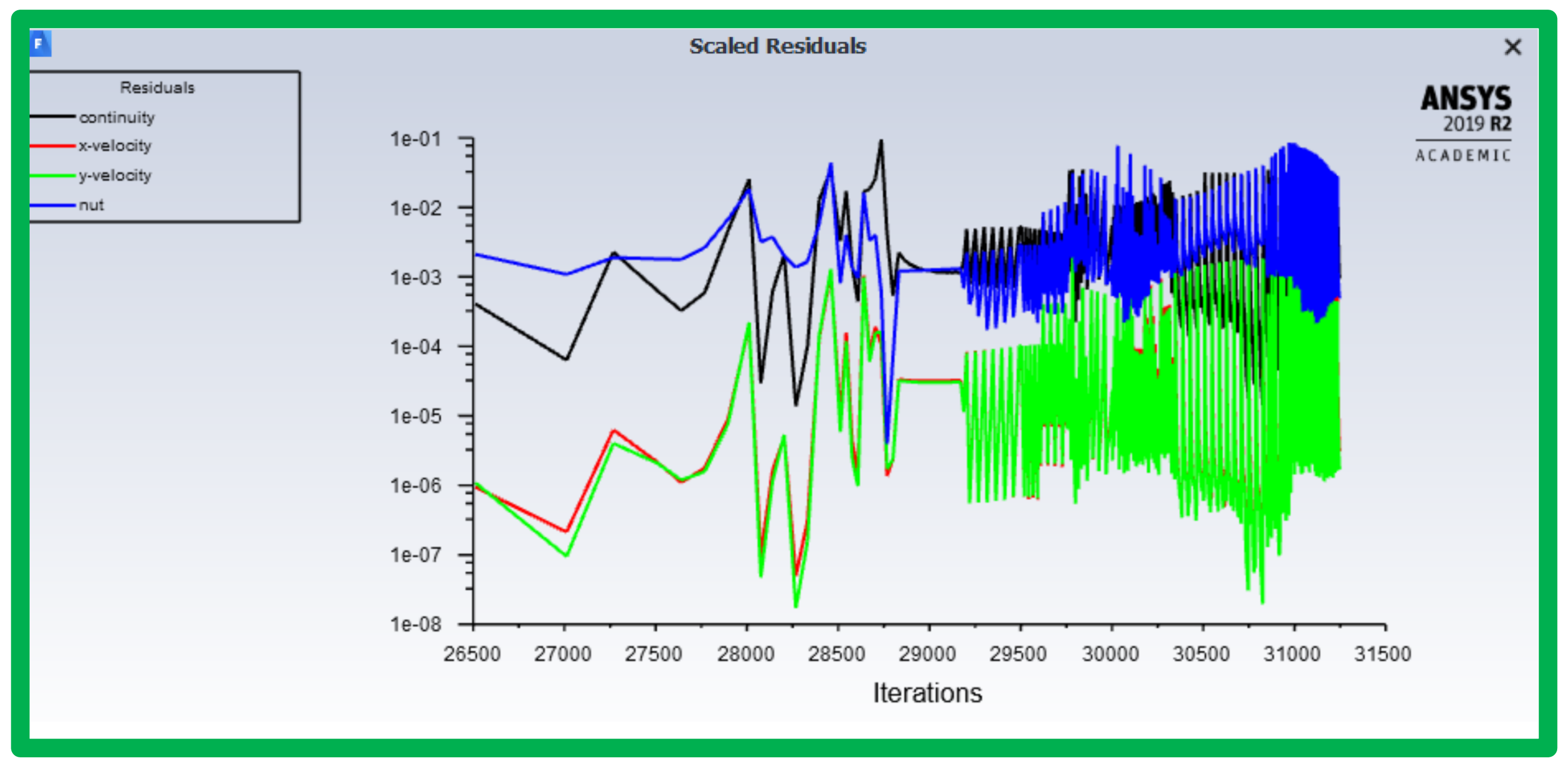




Results







Future Work

Although this study's results do not exactly calculate the values close to the published values, the computational code does in fact validate the existence of flight hysteresis. This means that the discrepancies are accounted by the spatial and temporal discretization, given that this work was done on elemental computers at home and not at the university. In addition the geometry of the domain can be excluded from influencing the discrepancy, because this simulation was done in two ways: sliding mesh and variable inlet velocity boundary conditions. And both produced the same results. Future work involves using other turbulence models such as k-omega and SST, in order to determine the best turbulence model to use for the wing deformation simulations.

References

[1] Barbarino, S, et al. "A Review on Shape Memory Alloys with Applications to Morphing Aircraft." Smart Materials and Structures, vol. 23, no. 6, 2014, p. 063001., doi:10.1088/0964-1726/23/6/063001.

[2] Fincham, J.h.s., and M.i. Friswell. "Aerodynamic Optimisation of a Camber Morphing Aerofoil." Aerospace Science and Technology, vol. 43, 2015, pp. 245–255., doi:10.1016/j.ast.2015.02.023.

[3] Barbarino, Silvestro, et al. "A Review of Morphing Aircraft." Journal of Intelligent Material Systems and Structures, vol. 22, no. 9, 2011, pp. 823–877., doi:10.1177/1045389x11414084.

[4] Mittal, Sanjay, and Priyank Saxena. "Hysteresis in Flow Past a NACA 0012 Airfoil." Computer Methods in Applied Mechanics and Engineering, vol. 191, no. 19-20, 2002, pp. 2207–2217., doi:10.1016/s0045-7825(01)00382-6.

[5] Koreanschi, Andreea, et al. "New Numerical Study of Boundary Layer Behavior on A Morphing Wing-with-Aileron System." 32nd AIAA Applied Aerodynamics Conference, 2014, doi:10.2514/6.2014-3170.

[6] Abdessemed, Chawki & Yao, Yufeng & Abdessalem, Bouferrouk & Narayan, Pritesh. (2017). Morphing Airfoils Unsteady Flow Analysis Using Dynamic Meshing. International Journal of Numerical Methods for Heat and Fluid Flow.