2023 Annual Conference & Exposition
Baltimore Convention Center, MD | June 25 - 28, 2023
The Harbor of Engineering
Education for 130 Years
Paper ID #40039

Using Free Software as Computational Wind Tunnels to Teach Students About Airfoils

Dr. Jason Andrew Roney, University of Denver

Dr. Roney is currently a Teaching Professor of Mechanical and Materials Engineering. Dr. Roney joined the University of Denver (DU) in Autumn 2014. Prior to joining DU, Dr. Roney held both industry and academic positions.

Using Free Software as Computational Wind Tunnels to Teach Students About Airfoils

Two-dimensional infinite airfoils are a fundamental concept in Aerodynamics and Aircraft design. Studying airfoils provides an estimate of the lift force and drag force for an aircraft. The Wright Brothers were revolutionary in their use of wind tunnels to design airfoils that helped provide the first powered flights at Kitty Hawk, North Carolina. The path forward from then on was to use wind tunnels to study fundamental airfoils shapes for use on aircraft and the different shaped airfoils were categorized and studied by the National Advisory Committee for Aeronautics (NACA), the precursor to the National Aeronautics and Space Administration (NASA). For instance, to this day, we still talk about the NACA 0012 airfoil. However, rarely does a university have all the wind-tunnel resources necessary to educate their students about all the different flow regimes from incompressible to compressible effects, nor the means to make or purchase all the different shaped airfoil models. Fortunately, today various free software packages are available to give students a fundamental understanding of the effects of flow regimes on different attributes such as coefficient of lift and drag of airfoils. As part of an introductory aerospace engineering course, the students at the University of Denver are given a project to study the NACA 0012 airfoil in the incompressible and compressible flow regimes with JavaFoil, primarily a vortex panel method with add-on models for viscous and compressible effects. And, in addition, the students study the compressible flow regime with the Ansys Fluent Student Version. In this project the students use the two software tools as computational wind tunnels where they study different angles of attack and flow conditions. Upon completion of their analysis, the students then compare their result with each method and with the known NACA Handbook values. This project thus provides a means for the students to synthesize the theory and concepts about aerodynamics taught in the first half of an introductory aerospace course by using a computational wind tunnel.

introduction

In Fall 2014, as a newly hired professor at the University of Denver in the Mechanical and Materials Engineering Department, I was given the opportunity to create new technical electives for our undergraduate students. At the University of Denver when I arrived there were no courses in Aerospace Engineering for our students, so I created two new electives known as "Introduction to Aerospace Engineering I" and "Introduction to Aerospace Engineering II". Coming from an industry position, and having attended a few pedogeological workshops, I was certain that the route to engaging the students was to engage them in Project Based Learning (PBL) and Experiential Learning (EL) [1], [2]. At the time, the experimental facilities at our university were not entirely conducive to hands-on Aerospace applications, so as I developed these courses, I looked to software options for introducing the students to Aerospace Engineering concepts. In this way, they could explore and synthesize the theoretical topics that were given as lectures in class. The first course which I developed was "Introduction to Aerospace Engineering I" which included course content on aerodynamics with some fundamental flight dynamics.

The basis for the project discussed in this paper grew from some initial first year PBL software explorations with a MATLAB® [3] Vortex Panel Code where the students described the aerodynamic characteristics such as coefficient of lift and drag for various airfoils in the "Introduction to Aerospace Engineering I" course. The type of analysis that was done with the code was typical of what would have been found using an experimental wind tunnel to find the lift-drag polar for an airfoil. Knowing that these were simply simulations, the students were asked to compare their solutions to the NACA Handbook [4], [5] values for their specific airfoil. The next year, I realized that by logging into our engineering server the students could also use our licensed version of Ansys Fluent Computational Fluid Dynamics (CFD) [6] software to also solve a compressible flow problem to predict lift and drag on a 2D airfoil in the compressible flow regime. At some point in the evolution of this project, the MATLAB® vortex panel method code stopped working with the latest version of MATLAB®, and at the same time due to popularity of the aerospace course, the server was unable to handle both faculty research projects and student projects.

However, I was able to pivot the PBL project to use free and downloadable software which is described in this paper. The first free software used is called JavaFoil [7]; JavaFoil is primarily a vortex panel method with add-on models for viscous and compressible effects and is fully documented [8]. Around this same time, Ansys also began allowing students (not just university students, but anyone who wanted to learn more about CFD software) to download the Ansys Fluent Student Version which allowed the students to install Ansys Fluent and Ansys Workbench [6] directly on their computer. The Ansys Fluent Student Version is limited in the number of computational cells that can be used in any given model but is effective at evaluating 2D airfoils. During my teaching evolution, the University of Denver also became part of The Kern Entrepreneurial Engineering Network KEEN, so I also used the KEEN Entrepreneurial Mindset (EM) [9] to further develop this project, and turned this project into a Curiosity, Connections, Creating Value (3 C's) project [9]. My project used a story (role-playing) to excite the students about using their computational wind tunnels. The next section describes this basic pedagogical methodology including the project basics along with some typical results. In addition, some general comments on the course are included.

methods

Two of the most important topics for the students to synthesize through PBL or EL was to understand how to determine lift and drag coefficients on a given airfoil design. The scaffolding for this project included in-class lectures on airfoils, wind tunnels, aerodynamics, turbulent flow, and compressible flow [10], [11]. Typically, the types of experiments that are done to understand lift and drag are done with low-speed wind tunnels in the incompressible flow regime. In bigger aerospace engineering departments, they may even be able to study the flow at transonic and supersonic flow regimes where there are new effects. The tools (free software) that the students were asked to download to complete this project were JavaFoil for subsonic flow and Ansys Fluent for transonic flow on an airfoil. The students were given the following prompt considering the EM mindset to create Curiosity:

"You are part of an engineer team of new hires in the Airfoil branch of the 'Aerodynamics Store'. Before moving on to the more sophisticated proprietary company airfoils, your branch manager wants you to show that you have the 'Right Stuff' when it comes to analyzing airfoils.

They suggest that you evaluate the NACA 0012 Airfoil which is shown in Figure 1 as your first engineering assignment. This airfoil has been around for years and is used in all sorts of Aerospace applications from low speed to high-speed applications. This project involves only computational analysis; the company has found that this is less cost prohibitive than doing wind-tunnel studies, but this type of work must also be eventually validated experimentally. The company has the following computational tools available for your study: 1) JavaFoil for Inviscid/Viscous 2D Flow 2) Fluent Computational Fluid Dynamics Tools for Compressible Flow"

Figure 1. Aerodystore NACA 0012 Airfoil

JavaFoil: incompressible flow

The JavaFoil interfaces are rather intuitive and do not require a lot of training to start producing immediate results. So, at a Reynolds number of $R_e = 6.0 \times 10^6$ the students are asked to provide or find the following for the NACA 0012 airfoil using JavaFoil:

1) A panel plot of the airfoil showing that they have the correct airfoil input to the program.
2) C_p, Coefficient of Pressure C_l, Coefficient of Lift, and C_d, Coefficient of Drag over the chord length as a function of the angle of attack for angles of attack from -20 to 20° for increments of 4°.
3) Plots of the flow field with flow vectors and streamlines, and pressure contours for select angles of attack.
4) C_l, Coefficient of Lift, with the stall model turned off (the idea is that turning the stall model off reverts the solution to the inviscid solution for lift).
5) The boundary layer height at the given Reynolds number and at a lower Reynolds number $R_e = 1.0 \times 10^5$ (the idea being that they can perceive that turbulent boundary layers are thinner than laminar boundary layers)

In addition, the students are asked to find the appropriate equivalent experimental wind-tunnel data that is available in the NACA Handbook to compare their computational values to the handbook values [4], [5]. The idea is that the JavaFoil calculations are at a flow speed that makes the aerodynamic coefficients Reynolds number independent up to the stall condition, and that there is a NACA Handbook case for $R_e = 6.0 \times 10^6$ close to or near the same value. Like laboratory experiments the students are asked to write up their results in a standard technical report format while providing some further data reduction.

The students are asked to plot the experimental data and the JavaFoil data on the same plot and comment on the comparison by exporting data from the program. In addition, they are asked to plot the lift to drag ratio as a function of angle of attack and determine the angle of attack that is most likely to have the highest lift to drag ratio. They are asked about whether they find a stall condition in any of their calculations. They are also asked to comment on inviscid flow/potential flow as a valid assumption. They are asked what happens to the boundary layer at higher Reynolds numbers, and what was the effect of turning off the "stall model" on the C_l.

The results requested above correspond to several of the topics that were addressed in the incompressible flow over airfoils part of the lecture course and enable the student to synthesize the ideas [10], [11]. Examples of major results the students produce are shown in the results section. Though JavaFoil has some methods for adding compressibility effects, they are limited, so the students were then asked to use Ansys Fluent to solve a compressible flow problem for the NACA 0012.

Ansys Fluent tutorial: 2D airfoil compressible flow

For the Ansys Fluent analysis the students were given the additional prompt which leads them to the free student version [6] and to the relevant airfoil tutorial [12], [13]:

"Your branch of the 'Aerodynamic Store' typically also sells to companies using airfoils which encounter compressible flow. The tool of choice at your branch for this analysis is Computational Fluid Dynamics (CFD) using Fluent. The software is available on the "Engineering Server" or you can choose to download a newer student version https://www.ansys.com/academic/free-student-products and install Ansys Workbench 2019 R3 on your personal computer (it requires about 20 GB). Your branch manager suggests that you first complete Tutorial 3 in Ansys Fluent (CFD) (the older version) or Tutorial 6 in the newer version. This tutorial involves compressible flow modeled over an airfoil and will give you the necessary experience to do this type of analysis. Provide the plots that are produced in the tutorial as proof that you completed the tutorial/training (Appendix of your Analysis Report).

Once you have completed your mandatory training tutorial, analyze the NACA 0012 Airfoil under compressible flow conditions. The market segment that would like to use this Airfoil would like to be able to use the airfoil at varying angles of attack ranging from 0.0 to 10.0 degrees and Mach numbers from 0.6 to 0.8. You can use the same geometry/grid as before (turns out the tutorial had the NACA 0012 already in it, bonus!). Hint: Boundary condition changes are needed for the different angle of attacks (the angle affects the velocity components and the force vectors that provide lift and drag). Typical test cases that have been used for other airfoils for similar clients is given below: run these Fluent Cases by changing the boundary conditions."

Table 1. New Conditions the students are asked to evaluate with Ansys Fluent.

Case 1:	Case 2:	Case 3:
M = 0.6 α = 0.0, 2.0, 4.0, 6.0, 8.0, 10.0	M = 0.7 α = 0.0, 2.0, 4.0, 6.0, 8.0, 10.0	M = 0.8 α = 0.0, 2.0, 4.0, 6.0, 8.0, 10.0

For these cases the students are asked to provide the following analysis:

1. Plot of C_l as a function of angle of attack for the three cases
2. Plot of C_d as a function of the angle of attack for the three cases
3. Plots of the C_p for each angle of attack for the three cases.
4. Contour plots of the C_p for the cases at 4.0 degrees

In addition to the analysis, the students are asked to consider the following when completing their discussion of the results in the report:

- How does C_l change with angle of attack?
- How does C_d change with angle of attack? How do the quantities change with Mach numbers?
- What would you predict for the location of separation for each angle of attack along the chord length?
- How do these results compare to the results obtained for the NACA 0012 airfoil for low speeds from both JavaFoil and the NACA Handbook.
- Is this a conventional airfoil or a supercritical airfoil? Explain based on your analysis.
- What is the approximate chord length at which a shock is obtained on this airfoil for the given angle of attack and what is the corresponding Coefficient of Pressure?
- What is your estimated value of the critical Mach number, M_{crit}. How does this value correspond to the freestream value?
- What is the corresponding wall shear stress before and after the critical chord length where a shock has been achieved? Explain these results.
- If you look at flow vectors near the boundary layer, is there flow reversal anywhere in the boundary layer? If so, where, and why does this occur?

The results requested above correspond to several topics that were addressed in the compressible flow over airfoils part of the lecture course and enable the student to synthesize the ideas [11]. Typical major results the students produce are shown in the next section.

results

JavaFoil: incompressible flow

The airfoil that the students put into JavaFoil has more x-y points than the default NACA 0012 airfoil as shown below in Figure 1. Using input from a text file requires them to interact more with the actual geometry and they provide verification that they put in the correct airfoil.

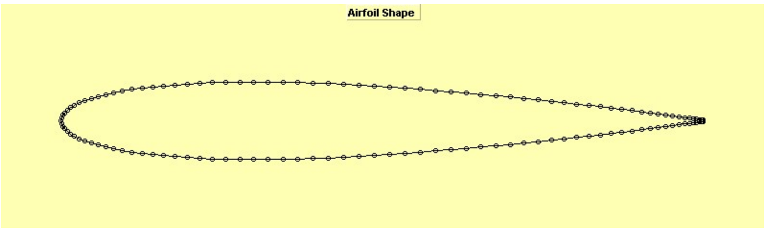


Figure 2. NACA 0012 airfoil data as input to JavaFoil.

The students then go analysis tab-by-tab with JavaFoil and perform the analysis as prescribed. All data is available for export so that they can produce the requested plots. Likewise, they are asked to digitize the relevant NACA 0012 handbook data and plot the data on the same plot. The NACA 0012 handbook has typical values of C_l and C_d and the types of plots that students produced are shown in Figure 3 and 4 respectively.

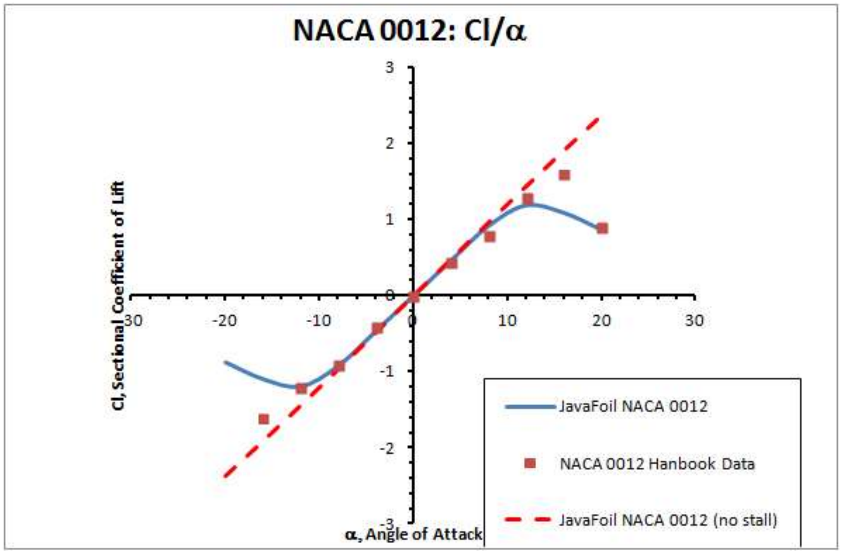


Figure 3. C_l, Coefficient of Lift for NACA 0012 data comparisons between JavaFoil and NACA Handbook.

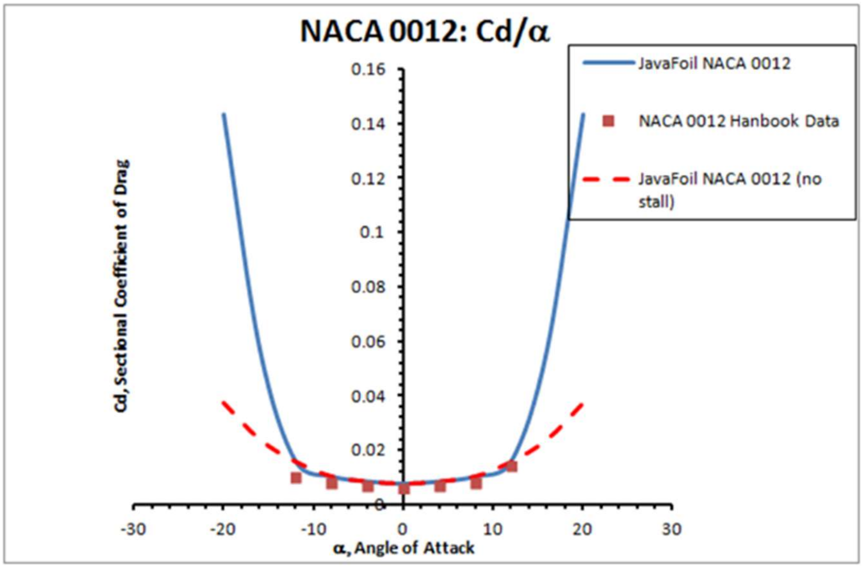


Figure 4. C_d, Coefficient of Lift for NACA 0012 data comparisons between JavaFoil and NACA Handbook.

The students can detect an airfoil stall condition as seen in these plots. The students can perceive through this analysis that the JavaFoil simulation is a realistic simulation when the stall model is turned on for incompressible flow when determining C_l and is good up to stall condition when only considering nearly inviscid flow which predicts no flow separation effect. The values of C_d as predicted entirely by skin friction drag for the no stall condition are also good up to stall and both stall and no stall models do a relatively good job of matching the NACA 0012 handbook data. Having realistic solutions of both of C_l and C_d allow the students to find the lift to drag ratio as shown in Figure 5 and predict the angle of attack that provides the maximum value of lift to drag. In the second half of the course, maximizing the lift to drag ratio is imperative for determining various aspects of flight performance and the students make this connection.

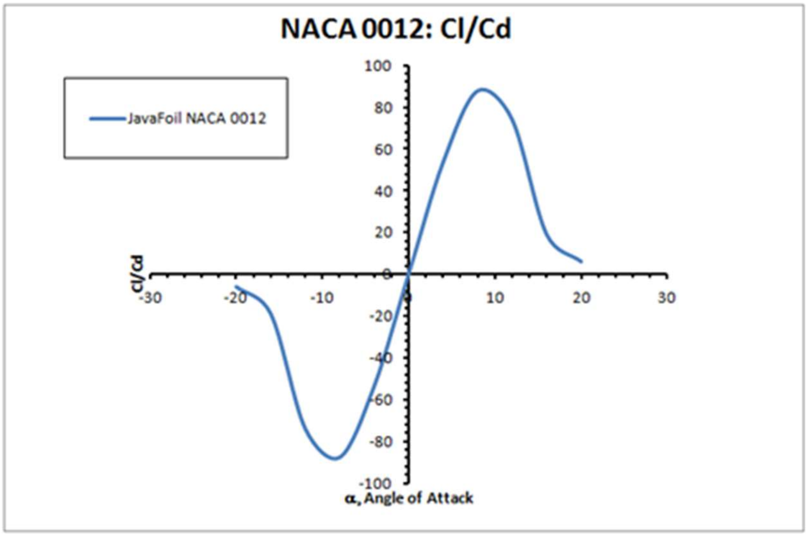
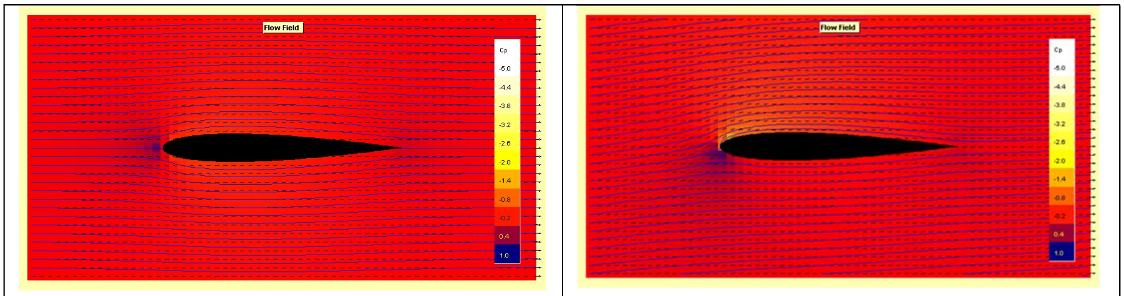
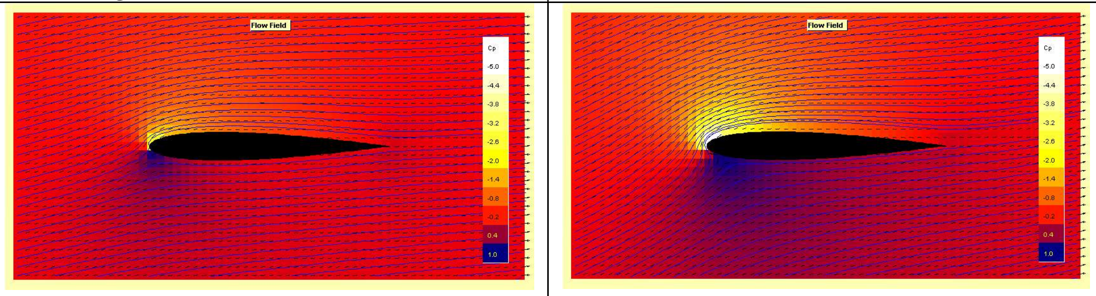


Figure 5. C_l/C_d, Lift to Drag ratio for the NACA 0012 using JavaFoil.

Through this analysis the students are exposed to the type of analysis done in a low-speed wind tunnel. In addition, they are exposed to gaining insight through simulation of some of the failings of over-simplified simulation models as well as the power of prediction if the right physics are incorporated into the model. In addition, by using simulation they can visualize streamlines, flow vectors and pressure contours as shown in Figure 6. The streamline and flow vector visualization are akin to using a smoke generator or a Particle Image Velocimeter (PIV) for visualization in the wind tunnel. The pressure contours are a bonus.



α = 0 deg. α = 4 deg.



α = 8 deg. α = 16 deg.

Figure 6. Streamlines, flow vectors and pressure contours produced with JavaFoil for four angles of attack.

Likewise, not all wind tunnels are equipped with airfoils with pressure taps and boundary layer rakes. and JavaFoil is able to provide additional insight into these types of experiments through simulation. For example, JavaFoil results for the C_p is plotted below for two angles of attack in Figure 7. For α = 0° the students observe for themselves that the symmetric airfoil is unable to produce any lift since both the top and bottom have identical pressure profiles. Likewise, plotting any of the other angles of attack such as α = 8°, the students observe that there is a pressure differential that should result in lift. Now, in Figure 8, the student has predicted the boundary layer height using JavaFoil for two different Reynolds numbers R_e =100,000 and R_e = 6,000,000. In studying viscous flow in the course, the students do homework and listen to lectures on turbulent and boundary layer flows. Inevitably these lectures are on flat plate theories. In class, the students do calculations to predict the boundary layer height on both a laminar and turbulent flow for a flat plate and conclude that the laminar boundary layer is considerably thicker than the turbulent one. The boundary layer result from JavaFoil shown in Figure 8 for an airfoil for α = 0° also indicates a similar trend and helps synthesize the flat plate result as being applicable to airfoils as well.

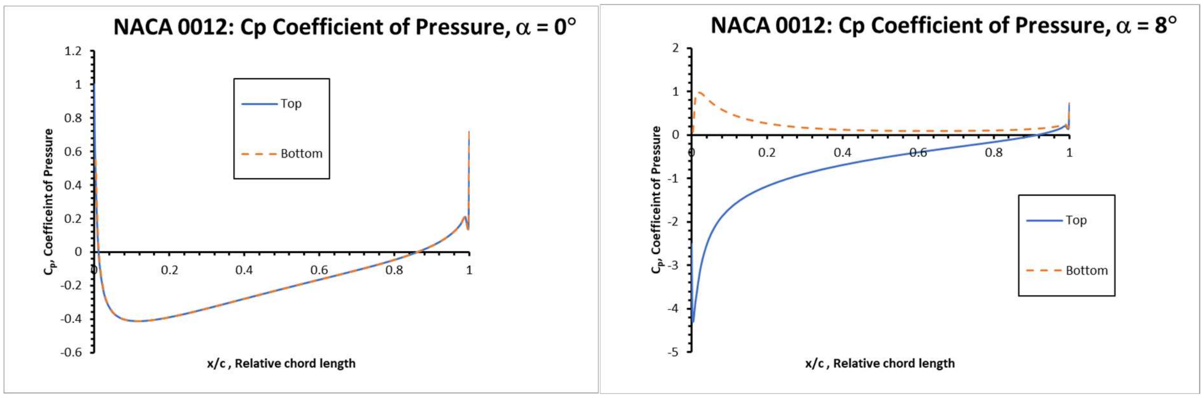


Figure 7. C_p, Coefficient of Pressure predicted for two different angles of attack with JavaFoil.

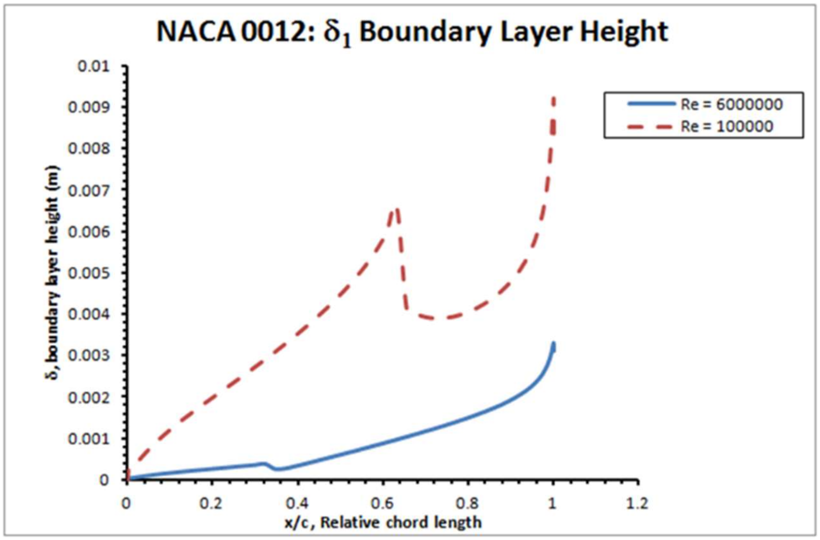


Figure 8. The boundary layer predicted for an airfoil for α = 0° for a laminar and turbulent boundary layer.

All these results for incompressible flow help synthesize the homework and lectures the students have been introduced to in “Introduction to Aerospace Engineering I”. The scaffolding for this part of the project included sections on low-speed wind tunnels and measurements, viscous flow (laminar and turbulent), and 2D airfoils [10], [11]. But the students have also been given scaffolding on compressible flow, high-speed wind tunnels, and compressible effects on 2D airfoils [10], [11], and the next part of the project results helps them synthesize their understanding on how this new flow regime might affect their results.

Ansys Fluent tutorial: 2D airfoil compressible flow

The students complete the Ansys Fluent tutorial entitled “Modeling External Compressible Flow” [12], [13] and then treat the methodology as a way of having a transonic wind tunnel at their disposal to investigate additional flow conditions. The main analysis are plots of C_l and C_d as function of angles of attacks from 0 to 10°. Each one of these angles of attack is a separate Fluent simulation and then they are repeated for different Mach numbers (M). At the higher angles of attack above 10°, the CFD results start to become unstable. In this paper, some illuminating results that the students find are shown in Figures 9 and 10 when plotting these results for the M = 0.6, and M = 0.8, respectively with the incompressible data for the same angles of attack.

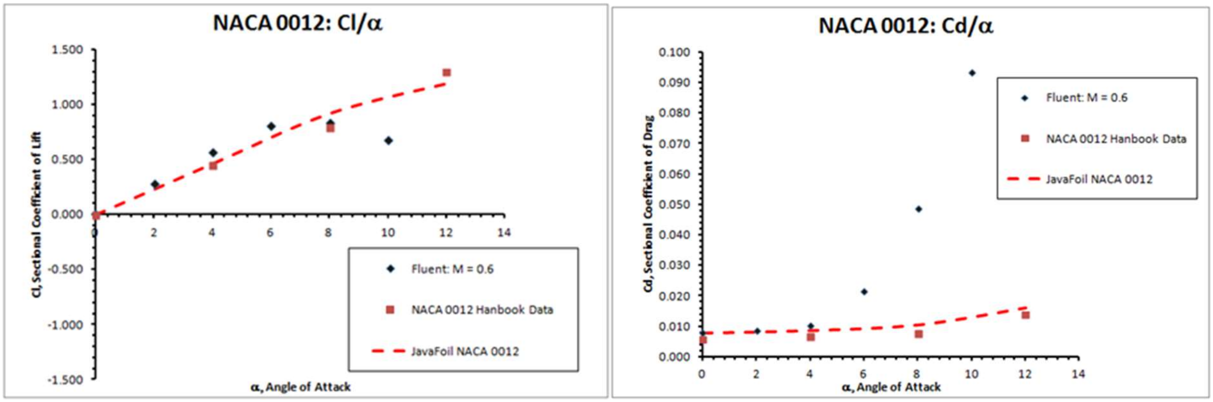


Figure 9. At M = 0.6, the plots for plots of C_l and C_d as function of angles of attacks from 0 to 10°.

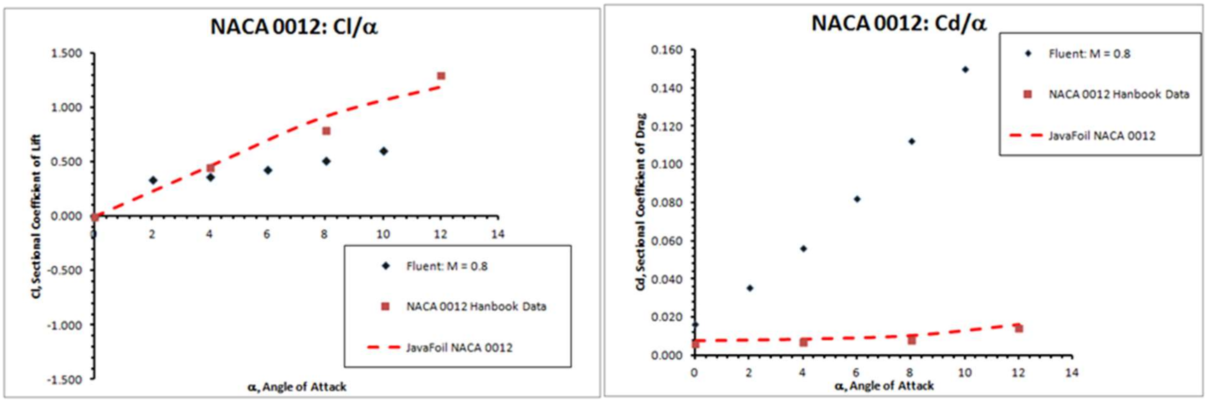


Figure 10. At M = 0.8, the plots for plots of C_l and C_d as function of angles of attacks from 0 to 10°.

In Figures 9 and 10, C_l increases with angle of attack for both cases until about 8 degrees as is typically expected up to stall; however, for M = 0.6, there is significant advantage in increasing the speed over the low-speed airfoil. This phenomenon is in accordance with the Prandtl-Glauert correction equation that the students learned about in lecture. However, at M= 0.8, the C_l actually is predicted to be lower by CFD, likely due to shock and wave effects where the Prandtl-Glauert correction is no longer valid as discussed in class. Over this range of angles of attacks, the C_d rises rapidly for compressible flow, even faster for M = 0.8. These effects are likely due to wave drag-type effects and shock induced separation that was also discussed in lecture on values above the critical Mach number M_{crit}.

The next CFD prediction of interest that the students make are various plots of C_p over the chord length of the airfoil. For their reports, they are asked to report all of them, but here only a couple representative plots are shown at M = 0.6 and M = 0.8 at two different angles of attack, $\alpha = 0°$ and 4° as shown in Figures 11 and 12, respectively. Like the incompressible case, the student observes that at $\alpha = 0°$ that there is no lift due to the fact the NACA 0012 is symmetric. Likewise, at $\alpha = 4°$ there is obviously lift due to the pressure differential between the top and bottom. However, now the shape of the curve indicates that there is likely flow separation occurring due to a shock on the airfoil for the M = 0.8 case at $\alpha = 0°$ and 4°. To further investigate whether a shock forms on the airfoil, students can plot pressure contours at M = 0.6 and M = 0.8 as shown in Figures 13 and 14, respectively for $\alpha = 0°$ and 4°.

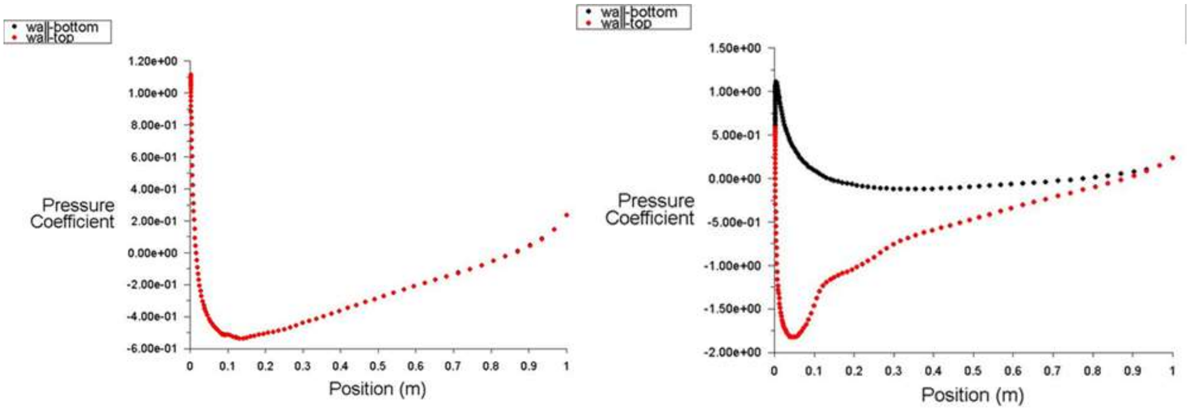


Figure 11. Plots of C_p for M = 0.6 at $\alpha = 0°$ and 4°.

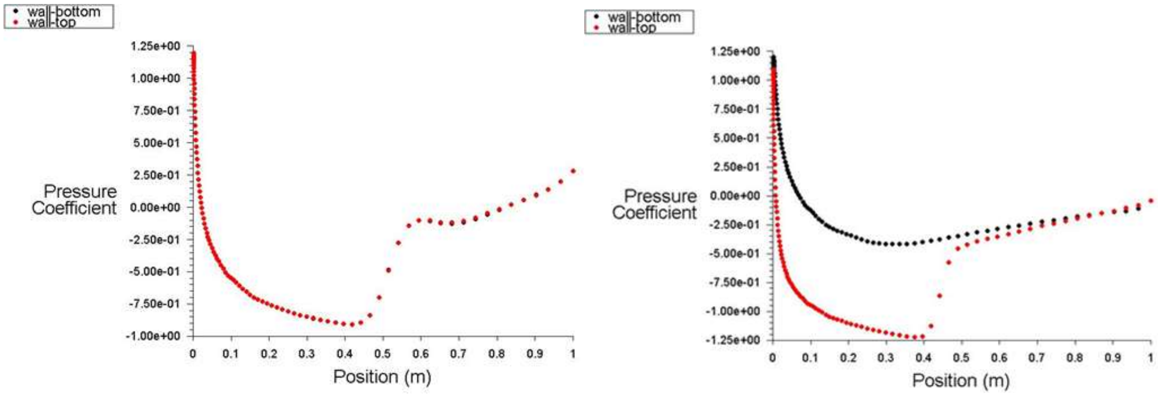


Figure 12. Plots of C_p for M = 0.8 at $\alpha = 0°$ and 4°.

From the C_p contours the students are able to further identify that there is no shock at M = 0.6, but there is shock behavior at M = 0.8 due to flow over the wing speeding up to M = 1.0 conditions. These results correspond well with what is also seen in the C_p contours.

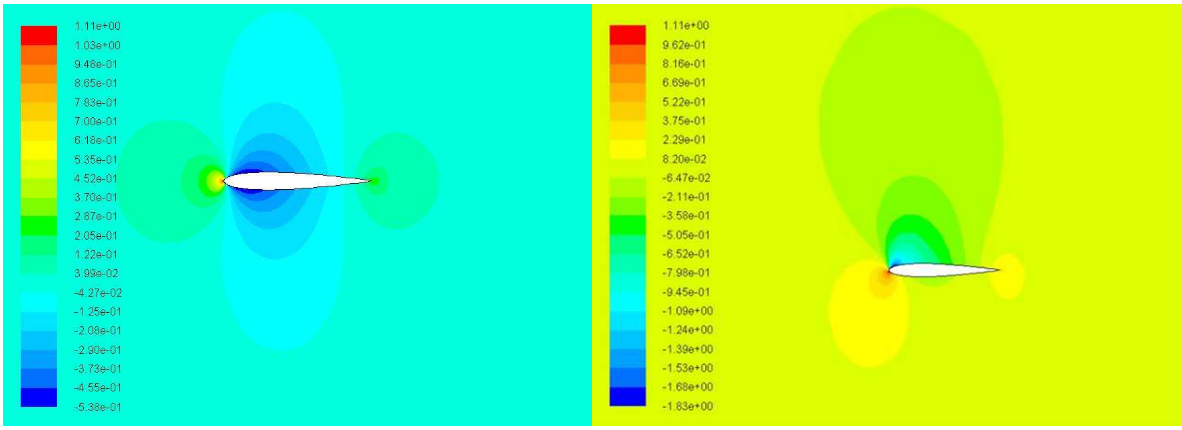


Figure 13. Plots of C_p contours for M = 0.6 at $\alpha = 0°$ and 4°.

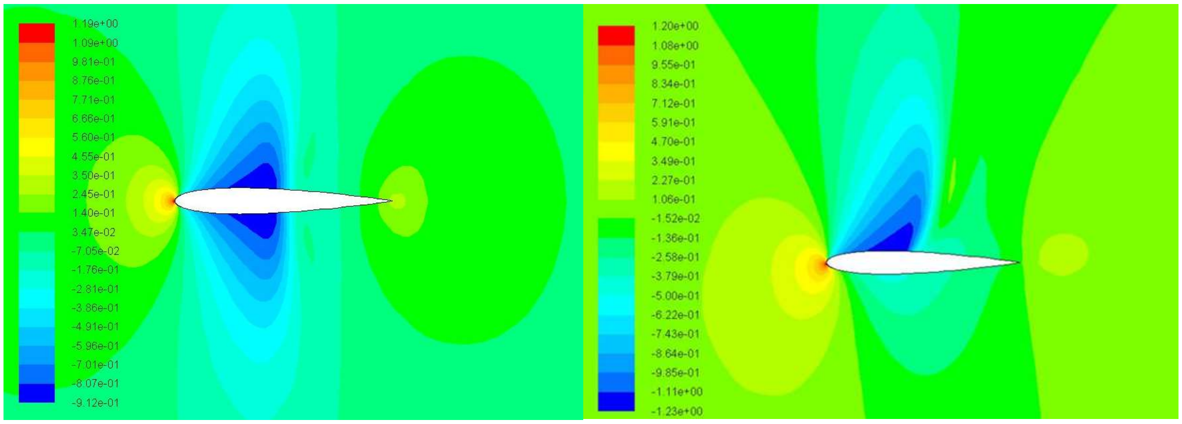


Figure 14. Plots of C_p contours for M = 0.8 at $\alpha = 0°$ and 4°.

The students are also asked to calculate the critical Mach number M_{crit} at a 4° angle of attack for the tutorial conditions at M = 0.8 for the NACA 0012. The theory for performing this calculation was presented in lectures, and in doing this calculation they find that $M_{crit} \approx 0.65$ for this condition at approximately x/c = 0.45 as shown in Figure 15 and as indicated by the intersections of the two lines representing the minimum coefficeint of pressure $C_{p,0}$ and the critical coefficient of pressure $C_{p,crit}$. From this theory the students should glean that for freestream values of $M_{crit} > 0.65$ that there will be a local shock on the NACA 0012 for a 4° angle of attack. This result also corresponds to their C_p contour CFD results which shows that there are no shocks at M = 0.6 but shocks form for M = 0.8 at a 4° angle of attack. Other items students can further investigate in this problem include finding flow reversal due to flow separation and wall shear stress going to zero due to separation as shown in Figures 16 and 17, respectively for M = 0.8 at a 4° angle of attack.

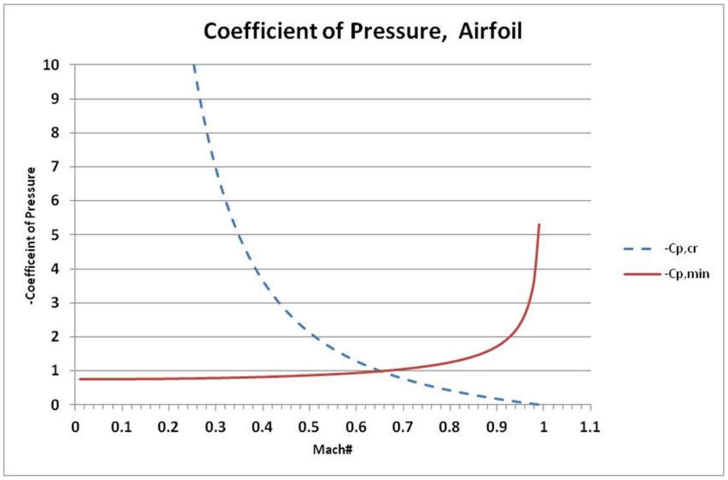


Figure 15. Plots for determining the critical Mach number for the NACA 0012 at a 4° angle of attack.

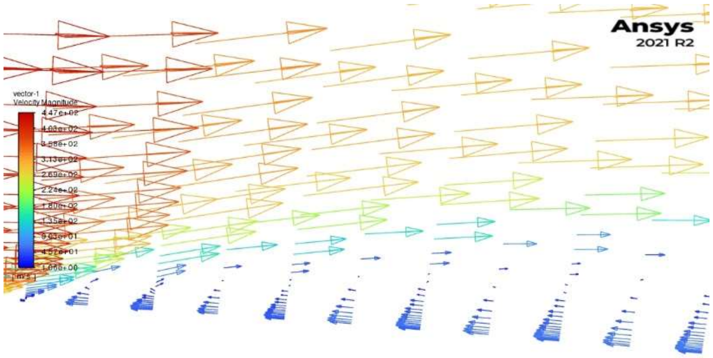


Figure 16. Vector flow field shows flow reversal at the separation location.

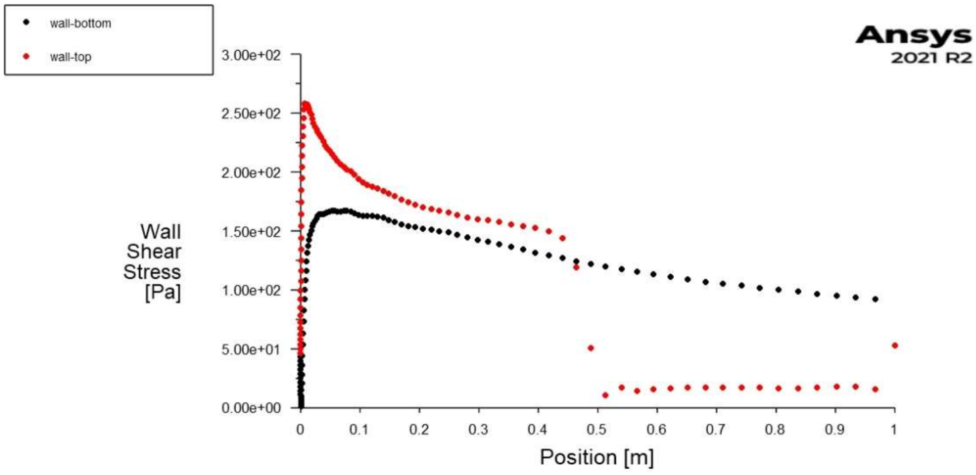


Figure 17. Wall shear stress shows flow separation location as it goes to zero and reverses on the top of the airfoil.

All these results for compressible flow using CFD as a wind tunnel to study compressible flow effects before and after the critical Mach number help synthesize the homework and lectures the students have been introduced to in “Introduction to Aerospace Engineering I”. The scaffolding for this part of the project were lectures on compressible flow, high-speed wind tunnels, and compressible effects on 2D airfoils [10], [11].

discussion

The above results are simply highlights of some of the results students can obtain by using JavaFoil and the free Ansys Fluent Student Version (CFD). As shown, meaningful results for airfoils can be obtained by using these tools as computational wind tunnels. The results compare quite favorably to the NACA Handbook data for incompressible flow calculations for the NACA 0012 using JavaFoil, and conceptually well with expected results for compressible flow using Ansys Fluent CFD.

Additional add-ons and variations to the computational experiments that continue to make this project fresh each year are analyzing different NACA airfoils with both JavaFoil and Ansys Fluent. In addition, the computational tools allow additional or different analysis each year. Since I also teach two CFD courses at the University of Denver, it is easy to create new geometries for different airfoils meaning that student projects will have slightly different results from year to year depending on the chosen airfoil.

In additional to using a computational wind tunnel, in recent years, the students are additionally exposed to a low-speed wind tunnel experiments with an airfoil either at the end of “Introduction to Aerospace Engineering I” or in “Mechanical Engineering Capstone Lab”, both of which are taught by the me. In the future, students will use our makerspace to 3D print airfoils for use in the wind tunnel while performing computational wind-tunnel analysis on those same airfoils.

Since developing this course with a PBL and EL slant this technical elective has become quite popular which may be indicative of students appreciating the engaging content. The initial course which was offered in Fall 2014 had 8 students and peaked at 42 students in Winter 2020. It was just recently offered again in Winter 2022, and 26 students took the course. Our typical graduating classes are on the order of 30 to 40 students and only Juniors and Senior students are eligible to take the course. In a typical quarter, they have five other offerings other than this course, so around 50% or more choose to take this course before graduation.

conclusions

Through incorporating PBL, EL, and EM methods in projects using free software such as JavaFoil and Ansys Fluent as computational wind tunnels, students synthesized their understanding of fundamental concepts in aerodynamics and 2D airfoils. The main aim of the project described in this paper was to introduce the students to similar findings that one may find for airfoils in compressible and incompressible flow regimes with experimental wind tunnels. Highlights of students synthesizing ideas related to incompressible flow airfoils included creating plots of C_d and C_l that compare well with the NACA Handbook data for the same airfoils. At the same time these methods aided them in further understanding concepts of viscous flow, inviscid flow, and laminar and turbulent boundary layers, and stall. Highlights of students synthesizing ideas related to compressible flow airfoils included determining how plots of C_d and C_l change as the Mach number increases. This analysis furthered their understanding of such concepts as the Prandtl-Glauert correction, the critical Mach number, and how local shock waves create flow separation.

Lastly, the EM prompt of working for the Aerodystore (role-playing) provided a framework for the students to first become curious about the performance of airfoils and what NACA was and how wind tunnels had been used in the past. Though they did not use an experimental wind tunnel it allowed them to become curious about how wind-tunnel data is reported and how a computational tool like JavaFoil or Ansys Fluent could provide similar data. Through the prompts, students began to see the connections between airfoil analysis and the theory that was presented in class and how ultimately it is related to airfoil and aircraft performance. Finally, by role playing that this analysis was for a business, the students saw that this work could create value for clients who uses airfoils in their industry.

References

[1] M. Frank, I. Lavy, and D. Elat, "Implementing the project-based learning approach in an academic engineering course," *International Journal of Technology and Design Education*, vol. 13, pp, 273–288, 2003.

[2] R. J. Durkin, "Experiential learning in engineering technology: a case study on problem solving in project-based learning at the undergraduate level," *Journal of Engineering Technology*, Spring, pp, 22-29, 2016.

[3] The Mathworks, MATLAB. [Download]. Natick, MA: The MathWorks, Inc., 2023.

[4] I. H. Abbott, A. E. von Doenhoff and L. S. Stivers, Jr., "Summary of airfoil data," National Advisory Committee for Aeronautics Langley Memorial Aeronautical Laboratory, Langley Field, VA, USA. Report No. 824, June,1948.

[5] L. K. Loftin, Jr. and K. S. Cohen, "Aerodynamic characteristics of a number of modified four-digit series airfoil sections," National Advisory Committee for Aeronautics Langley Memorial Aeronautical Laboratory, Langley Field, VA, USA. Technical Note, No. 1591, 1945.

[6] Ansys, "Ansys Student - Free Software Download" February, 2022. [Online]. Available: https://www.ansys.com/academic/students/ansys-student. [Accessed Feb. 27, 2023].

[7] M. Hepperle, "JavaFoil — Analysis of Airfoils," February, 2022. [Online]. Available: https://www.mh-aerotools.de/airfoils/javafoil.htm . [Accessed Feb. 27, 2023].

[8] M. Hepperle, "JAVAFOIL User's Guide," December 22, 2017. [Online]. Available: https://www.mh-aerotools.de/airfoils/javafoil.htm . [Accessed Feb. 27, 2023].

[9] KEEN, "The Entrepreneurial Mindset" February, 2022. [Online]. Available: https://engineeringunleashed.com/mindset . [Accessed Feb. 27, 2023].

[10] J. D. Anderson, Jr., “Chapter 4, Basic Aerodynamics,” in Introduction to Flight, 8th Edition. New York: McGraw Hill Education, 2012, pp-134-287.

[11] J. D. Anderson, Jr., “Chapter 5, Airfoils, Wings, and Other Aerodynamic Shapes,” in Introduction to Flight, 8th Edition. New York: McGraw Hill Education, 2012, pp-288-440.

[12] Ansys, *Tutorial 3. Modeling External Compressible Flow,* Release 12.0, ANSYS, Inc. March 12, 2009.

[13] Ansys, *Chapter 6: Modeling External Compressible Flow*, Release 2019 R2 - © ANSYS, Inc. All rights reserved.