Simple Flow Analysis of a NACA 23012 Airfoil Using SolidWorks FlowSim & Altair HyperWorks

Benjamin Horine

Mechanical Engineering and Material Science Dept.

Washington University in St. Louis

St. Louis, USA

bhorine@wustl.edu

Abstract—**Computational Fluid Dynamics (CFD) is used to analyze problems with fluid flow using numerical analysis and data structures. There are many types of software that offer simulation features involving interior and external CFD analysis. This experiment will use SOLIDWORKS Flow Simulation and Altair HyperWorks CFD to simulate and visualize airflow over a NACA 23012 airfoil. These models use speeds and other numbers from the standard NACA testing procedures. This report will compare pressure plots and drag to lift ratios recorded from each simulation.**

Index Terms—**Computational Fluid Dynamics (CFD), Solid-Works FlowSim, Altair HyperWorks, Flow Analysis, Lift, Drag**

I. INTRODUCTION

Computational Fluid Dynamics (CFD) is an useful tool for engineers that are trying to understand fluid flow through a surface or over a surface. Previously in the course an analysis is done to evaluate internal flow. The problem for this experiment is to provide an example of how to simulate and analyze external flow over a surface. External flow analysis will be useful for students working on projects such as WashU Racing, Design Build Fly, and WU Rocketry. These student groups work directly with models that are affected by moving through an environment. The ability to reliably predict how the models will behave improves safety and reduces the time in the design process.

The simulation software that this project will use is SOLID-WORKS Flow Simulation due to its ease of setup and availability, and Altair HyperWorks CFD due to the designated CFD application available and to broaden the scope of what software is available to students with CFD needs. This expirement will focus on obtaining pressure plots from each program and a drag to lift ratio of a NACA 23012 airfoil.

The drag to lift ratio is made of the following equation:

$$\text{Drag to Lift Ratio} = \frac{D}{L} = \frac{C_d}{C_l} \tag{1}$$

where C_d is the coefficient of lift, C_d is the coefficient of drag, L is lift, and D is drag. This equation also describes glide ratio.

II. BUILDING THE NACA 23012 AIRFOIL

This projects introduces the National Advisory Committee for Aeronautics (NACA) Airfoil data base. This is an useful tool for students building and designing with airfoils in mind.

The important modeling skill students learn from this segment is importing a curve file. From the data base, studetns will download a data file (*.dat) or a CSV and convert it to a tab-delimited text (*.txt) file. This file is imported to the model as a curve. Students will realize that these coordinates could be any sort of shape and could even be a 3D sketch. After importing the shape, building the geometry is pretty straight. The completed model is displayed below in Fig. 1 and Fig. 2.
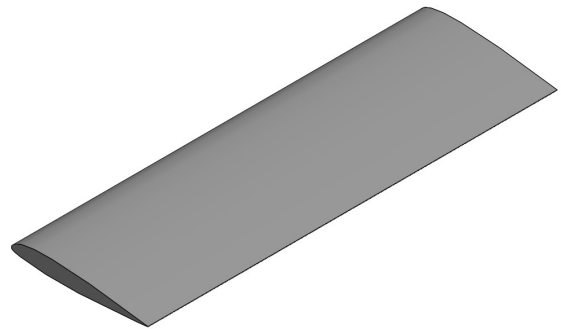

Fig. 1: Isometric view of the model

Fig. 2: Side view of the model

The specific airfoil used for this project is a NACA 23012 due to it's high drag to lift ratio and popularity in aircraft that fly at sub supersonic speeds. The NACA 23012 has a maximum thickness of 12% at 29.8%. The maximum chamber of this foil is 1.8% at 12.7% chord [1]. This foil is shown below in Fig. 3.
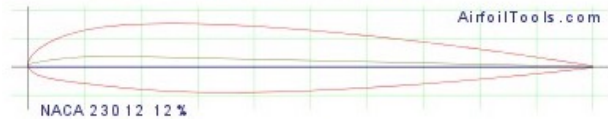
Fig. 3: NACA 23012 Airfoil description from the NACA Airfoil Data Base [1].

The only potential issue is that the downloaded curves are not completed entiteies, seen in Fig. 4. Students will need to take note of this before moving forward with the rest of the build.
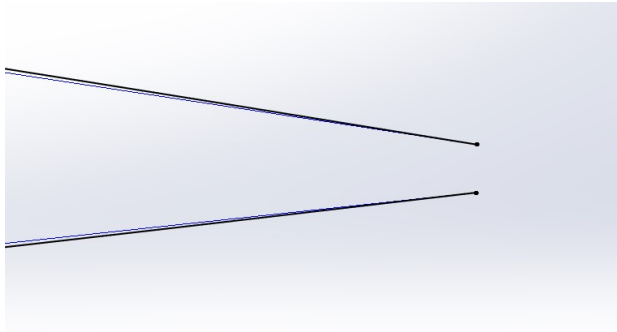

Fig. 4: Open geometry of the tailing edge

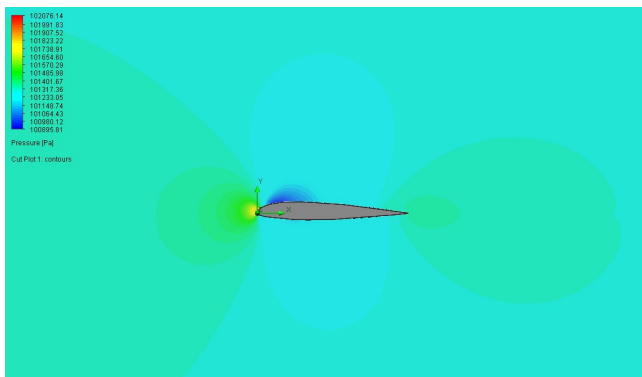## III. SOLIDWORKS FLOW SIMULATION RESULTS

Fig. 5: Pressure plot from SOLIDWORKS

SOLIDWORKs is the primary program used to model the geometry, so importing the model to the simlulation work space is almost effortless. The wizard function allows for an easy setup for the physics of the simulation. In comparison to the other skill demo on fluid flow, this part was very similar. The difference is in the staging of the design space.

Since the model is so simple, the simulation can be simplified to a 2D simulation. Thus, the design space is decreased to 0.02 m from the mid-plane. The tailing edge has 0.75 m. The top, bottom, and front of the design space account for 0.25 m of space.

The simulation did not take long to mesh or run. The results from the simulation are found in cut plots. The pressure plot from the simulation is shown in Fig. 5.

The drag and lift coefficients are disguised as an Force (X) and Force (Y) in the surface parameters. The maximum values will give the student the values they will need to calculate.
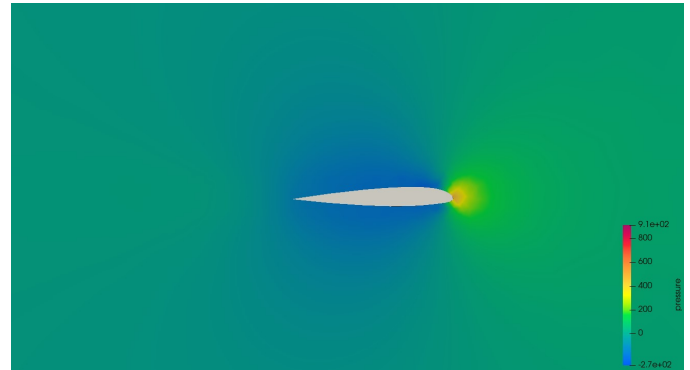
IV. ALTAIR HYPERWORKS CFD RESULTS

Fig. 6: Pressure Plot from HyperWorks CFD

This program required some external software to be downloaded on top of the HyperWorks Bundle. Beware of this before continuing with the simulation.

The design space setup in HyperWorks is a little more involved and less intuitive for a small simple model like this. For large assemblies, this software would work really well. The design space is set up by an arbitrarily located box that the user can then modify. For programs such as WashU Racing, this could be useful to simulate ground effect.

The physics setup is more involved and the options to the type of turbulence are also features that improve accuracy to real behavior in an air stream. HyperWorks uses a design space to set up the model before moving forward with meshing and simulation. When the mesh is created from the model and a results file is exported. This file is used for the simulation and analysis. The mesh model is shown below in Fig. **??**.
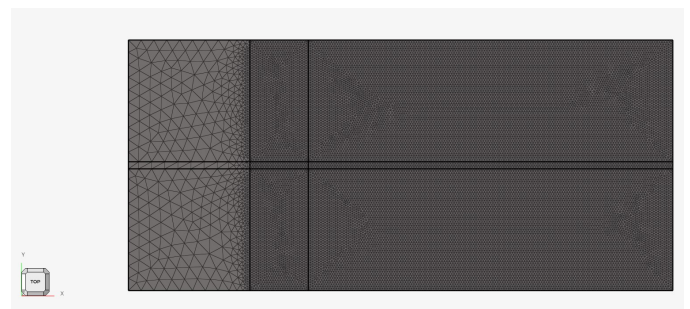

Fig. 7: Mesh of model from HyperWorks CFD

The pressure plot is found in the post simulation tab. The user needs to isolate the symmetric boundary to access the pressure plot seen in Fig. 6. The force of X and force of Y is also found in this window, for drag and lift, respectively.

V. RESULTS

As seen above in Fig 5 and Fig. 6, the pressure plots are very similar in contour over the model. Each plot has the rainbow color scheme selected for an easy comparison.

SOLIDWORKs Flow Simulation allows the user to choose how many contours to include in the plot. In the plot above, the number chosen was continuous at 40 contours. Even though its not a true continuous model, the SOLiDWORKs plot shows transition zones and separation regions. Similarly, HyperWorks displays a very continuous plot. At a closer inspection, the pressure regions vary in the low pressure zones. In the ambient pressure and high pressure regions the two models are very similar.

The low pressure regions between each model is similar where there is a more noticeable low pressure region above the foil. However, they differ in the HyperWorks model, as the low pressure region, and transition regions, are more pronounced. This difference is produced as the HyperWorks modeling space is able to define more surfaces for with specific element sizes in terms of the model and the design space.

The units between the two software are different. SOLID-WORKs uses SI and had standard MKS units set up. The HyperWorks manual says that it will only work with SI units; However, per the images above, there are no units listed anywhere to confirm this. This makes the comparison difficult. Please note that these ratios are not very good considering the fact that there is no angle of attack introduced. This analysis was done to understand the aerodynamics of a standalone foil In SOLIDWORKS, the ratio was 3.65 to 1. The ratio found in HyperWorks was 32.7 to 1. Assuming that there is a factor of ten off in the units. These are very close if that ends up being the case.

This shows that the two models are comparable in analysis. For simple bodied models or quick simulations, SOLID-WORKS Flow Simulation provides a better route to getting an analysis. The larger assemblies that have complex geometries and symmetrical bodies would benefit from an analysis from HyperWorks CFD.

VI. DISCUSSION

This experiment was also to include an analysis with AN-SYS Fluent, however, I quickly realized that this was too much to accomplish for a comparison skill demo. Which is why i reverted to comparing SOLIDWORKS Flow Simulation and Altair HyperWorks Bundle. The geometry in ANSYS Fluent began to get complex quick. The run times for this simple model also exceeded the HyperWorks run times. I think that in the future the use of ANSYS for a CFD analysis could be done as a small project option for a multiple element foil. Therefore, multiple angles of attack and distances can be introduced. This would be too much for a Skill Demo, but it would be just right for a Small Project. That being said, I would not include using HyperWorks or SOLIDWORKs with ANSYS unless it was to create a model for the simulation.

As far a HypeWorks goes, this software offers a lot. I really enjoyed learning it. The design space was challenging to pick up at first, but I was able to get the hang of it to get results. I am confused why they do not offer the program in any units other than SI. I also find it strange that they do not have any units posted in the software. This makes it difficult to verify what is being entered and could be a possible error to the difference in drag to lift ratios.

VII. CONCLUSION

This design study shows the comparison to external airflow analysis on a NACA 23012 airfoil using SOLIDWORKS Flow Simulation and Altair HyperWorks CFD. This experiment introduces users to a new type of CFD analysis from the others offered in this course. More preparation would be needed to provide a sufficient Skill Demo. This could be a feasible study for a small project.

REFERENCES

[1] Robert C. Platt and IRA H Abbot. "Aerodynamic Characteristics of N.A.C.A. 23012 and 23021 Airfoils With 20%-Chord External-Airfoil Flaps of a N.A.C.A 23012 Section". In: National Advisory Committee For Aeronautics 573 (1937), pp. 1–21.