

## Exercise: Exterior Flow Analysis

This exercise is to show you how to run an exterior flow analysis model using SolidWorks Flow Simulation, Altair HyperWorks, and ANSYS Fluent. The model that you will create will be of an airfoil using the NACA Airfoil Database. Each program will be able to give you information about the model and simulation you run. In this exercises you will compare a lift to drag ratio between each program to the NACA Airfoil Data Base. Additionally, you will be able to explore the different types of plots and features each program has to offer. **Deliverables to submit: A pressure plot from each program, and a lift to drag ratio from each simulation.**

Use the following equation to calculate drag to lift ratio:

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

## Build the Model

1. On your computer, designate a **folder** for this exercise.

Note: Be sure to be mindful of file naming convention.

2. Open a **web browser** (Google Chrome is recommended). Navigate to [airfoiltools.com](http://airfoiltools.com). This website is a database full with NACA airfoils of four and five digits. There are other airfoils available. If time allows, please browse around.
3. Go to Airfoil database search. In the text search box, please type in the following number: 23012
4. The following result should appear:  
\* Insert Photo\*
5. Click on this image and it will take you to a page with more information about the NACA 23012 airfoil. This airfoil is very popular as it has the a good lift to drag ratio [1]. See appendix for more NACA Airfoil information.
6. On the right hand side of the screen click on the link titled “[Send to airfoil plotter](#)”.
7. You will want to download this file as an **Excel** file with an .csv extension.
8. Open the file in **Excel** and save as a tab delimited text file with a .txt extension. We will be importing these coordinates into SOLIDWORKS, which will only accept SOLIDWORKS Curve files with a .sldcrv extension or tab-delimited text files with a .txt extension.
9. Launch **Excel** and open the CSV file you just saved.
10. Save this file as a **tab-delimited text** file with a .txt extension.

11. Launch **SOLIDWORKS**.

12. Open a new part file and ensure that the units are in **MKS**.

Note: This is found in the bottom left corner or in settings.

13. Save this file to your designated folder created in Step 1.
14. Under the SOLIDWORKS Add-Ins tab, please ensure that **ScanTo3D** option is **selected**.
15. Under the Features Bar, select **Curve Through XYZ Points** in the **Curves** tab.  
Note: For reference, this button tab is located in right next to the Reference Geometry tab and should be in the middle or right side of your screen.
16. Select the **Browse** button. At the lower right corner change file type from Curves (\*.sldcrv) to **Text Files (\*.txt)**. Navigate to the .txt file you created and select **Open**.
17. Select **Save** and then **Insert**.  
This has saved a Curve (\*.sldcrv) file and inserted the Text File (\*.txt). If this set has not worked, try saving your CSV file again as a tab-delimited text file. There are many different times of \*.txt files available, it needs to be tab-delimited. If you have more issues please reach out to the course assistants or the instructor.
18. The curved outline of NACA 23012 should appear in the work space. Once this has happened, select **OK**.
19. Create a new sketch on the front plane. Convert entities from the curve to this sketch. On the trailing edge, create a line to complete the sketch. Note: This will create a new face when extruded. Do not forget to include this face when running simulations.
20. **Extrude** this sketch **0.300m** from the midplane.
21. The model is created, now would be a good time to save it again as a SOLIDWORKS Part File (\*.sldprt) and a save as a Parasolid file (\*.x\_t).

## SolidWorks Flow Simulation

1. Launch **SOLIDWORKS** if you have not already done so.
2. Open the part file of the model created in [Build the Model](#).
3. Similar to step 14, under the SOLIDWORKS Add-Ins tab, please ensure that **Flow Simulation** option is **selected**. This may take some time to load.  
Note: This is different from the simulation tab, which may or may not be selected already, which will not affect is exercise either way.
4. At the upper left hand corner, select the **Wizard** button.
5. Same the project with your last name using the following format: LASTNAME\_NACA23012. Use the current configuration. Select **Next**→.
6. Ensure that SI units are being used. Select **Next**→.
7. Select external. The other options can be unselected. Select **Next**→.

8. Select (+) on the Gases section. Double click Air, or single click and select the Add button. Ensure Laminar and Turbulent option is selected for Flow Type. The other options can be unslected. Select **Next**→.
9. Ensure Adiabatic wall with 10 micron roughness (RA). Select **Next**→.
10. Ensure temperature is at 293.2 K and pressure is at 101325 Pa. Check to make sure the x-axis goes from the leading edge of the foil to the trailing edge of the foil. Change the velocity in X direction to 35 m/s. If the orientation of the foil is not as pictured, or described above, please adjust your model or parameters above to match this. Select **Finish**.
11. Under the project manager tree, please select the Flow Simulation tab if it is not already selected.
12. Right-click on Computational Domain and select **Edit Definition**. Change the following options to setup the design space:
  - ▶ Change type to **2D Simulation** in the XY plane.
  - ▶ Change X max to **0.75 m** and X min to **-0.25 m**.
  - ▶ Change Y max to **0.25 m** and Y min to **-0.25 m**.
  - ▶ Change Z max to **0.01 m** and Z min to **-0.01 m**.

Select **OK**.

13. Right-click Boundary Conditions and select **Insert Boundary Conditions...**. Select the following options:
  - ▶ Under Type, select the **Wall** button, and choose **Real Wall**.  
Note: Leave all other settings unchanged.
  - ▶ Under selection, select the **main face** of the airfoil and select the **trailing edge face**.  
NOTE: THIS IS IMPORTANT, DO NOT FORGET TO SELECT BOTH

Select **OK**.

14. Right-click Goal, select **Insert Global Goals...**. Select under the Max column: Force (X) and Force (Y). These will drive the necessary results for finding lift and drag.

Select **OK**.

15. If you would like to make a mesh, select **Create Mesh** under global mesh. Otherwise, one will automatically be generated when the simulation runs.
16. Under the Flow Simulation tab, click **Run**.
17. In the Run window select the following:
  - ▶ Select Mesh if you have not already created a mesh for the model.
  - ▶ Select Solve and run a **New calculation**

- ▶ Run at: **This Computer**
- ▶ Use all of the cores **minus one**.

Note: This will be done by you manually, select the second to last option. For example, if you have 8 cores on your computer use 7, if you have 4 cores use 3.

- ▶ Select **Load results**.

18. Select the **Run** button.

Note: The time to run this simulation will depend on your computers abilities. If this step takes more than ten minutes abort and try a different set up.

19. Expand the Results branch in the Project Manager Tree.

20. Insert a **Mesh** from the Front Plane and select Refinement Level, this shows how refined the mesh is inside of the design domain. Right-click this feature and hide it.

21. Insert a Cut Plot with **Pressure** selected.

22. To find lift and drag, select the following:

- ▶ Right-click Surface Parameters and select **Insert**.
- ▶ Under selection, select the **main face** of the airfoil and select the **trailing edge face**.  
NOTE: THIS IS IMPORTANT, DO NOT FORGET TO SELECT BOTH
- ▶ Under Parameters, select **Force**.  
Note: If this option is not listed, select More Parameters and find it under Loads.
- ▶ Select the Show button.
- ▶ On the lower right-hand side of the screen record the X-component and Y-component, which are drag and lift, respectively.

Select **OK**.

23. Calculate the drag to lift equation using Eq. [1](#).

24. Take a picture of the model from a side view. Include the pressure plot and the scale.

25. Use this image and the drag to lift coefficient as your deliverable for this part of the exercise.

## Altair HyperWorks

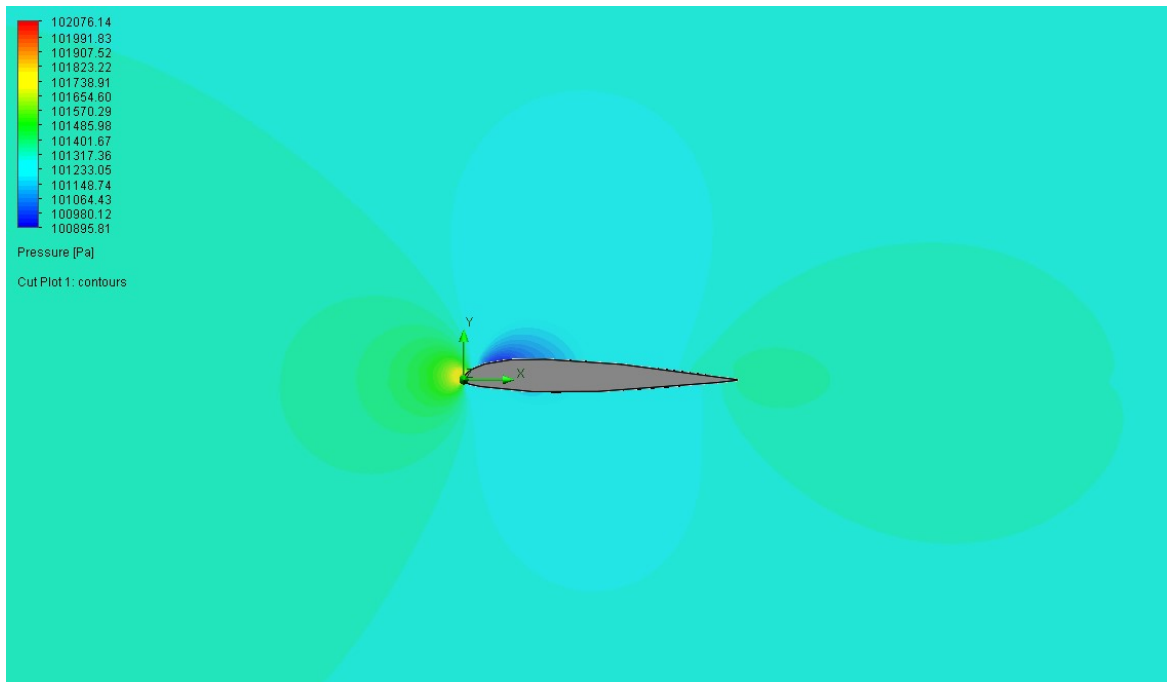
1. **Launch** Altair Inspire 2021.1.
2. **Open** the STEP (\*.step) of your model by selecting File → Import and navigating to the file location. Select [width=].
3. Ensure the part is in meters and that the dimensions imported correctly use the **measuring tool** at the top left.
4. Under the Geometry tab, select the **Validate** button on the left hand side. This will run SurfaceChecks. For larger assemblies this is a valuable tool to understand if there is a surface or connection between surfaces that would need to be cleaned up prior to meshing.
5. Under the same Tab, select **Split**. Then select Plane, choose Solids as the Target and use Plane as the Tool.
6. Select any arbitrary point on the leading edge surface, this will make a box appear. Select the XYZ option, and enter zero for X and Y. For the Z coordinate select 0.01 m. Select **Split**.
7. Repeat the step above, this time select the Z coordinate as -0.01 m.
8. Under the Parts Browser, **delete** the two larger bodies, leaving the smaller +/-0.01 part.
9. Create a design space by selecting the Solids button. Under this, choose the Box option, select For to, and select the part.
10. Use the drag tool to extrude the following:
  - ▶ Inlet **0.25 m**
  - ▶ Outlet **0.75 m**
  - ▶ Top **0.25 m**
  - ▶ Bottom **0.25 m**
11. Use the Boolean feature to **subtract** the airfoil from the design space by selecting Subtract. Use the airfoil as the tool and the design space as the target. The airfoil should disappear and there should only be one Solid Body in the work space.
12. Run another **Validation**, and ensure that there are not checks. If there is a check, run another Boolean **subtract** to get only one solid body.
13. Select Physics Tab, and it will show a setup window shown below. Select **Mildly compressible** single phase flow. Select **Air-Ideal** Gas for Ideal gas. Select Steady for Time Marching. Select **Spalart-Allmaras** for the Turbulence model. Ensure that the pressure for the pressure gauge is set at 101325 Pa.
14. Select **Profiled** and select the three surfaces of the box to be the inlet. Then change the temperature and average velocity to 293.2 K and 35 m/s respectively. If temperature is not an available option, ignore it.

15. Create an **Outlet** on the opposite side of the inlet and set the pressure and loss factor to zero.
16. Create a **Slip** condition on all surfaces of the upper, lower, and opposite side of the box (bottom plane view).
17. Create a **No Slip** boundary condition by selecting the leading edge surface and the trailing edge surface.
18. Create a **Symmetry** boundary on the faces closest from the top plane view.
19. Under the Mesh tab, select surfaces and click on the three dots. Select the **o Slip boundary** condition of the two faces. Change the element size to 0.001 m, and the Geometric feature angle to 15 degrees. Ensure that the Minimum size factor is at 0.1 and that the mesh growth angle is at 1.3.
20. When the mesh is completed, select the **Run** under the Solution tab.
21. Ensure the correct directories and file naming. Allow all processes to run, and use Intel MPI for the Parallel processing. Use one processor or less than half the processors you had available in the SolidWorks simulation, and use that same number for the number of threads if you have an Intel i7 on your computer due to the ability to HyperThread. If you have anything less than that, you will need to run one less than the number available on SOLIDWORKS.
22. Select **Run**.
23. Use the Post Tab to get the plots. Isolate the main body and then right-click and select edit. Change the type to **pressure**, and you can see the plot.
24. Similarly, plot **force (X)** and **force (Y)** to get your drag and lift.
25. Calculate the drag to lift equation using Eq. [1](#).
26. Take a picture of the model from a side view. Include the pressure plot and the scale.
27. Use this image and the drag to lift coefficient as your deliverable for this part of the exercise.

## Answer Key: Exterior Flow Analysis

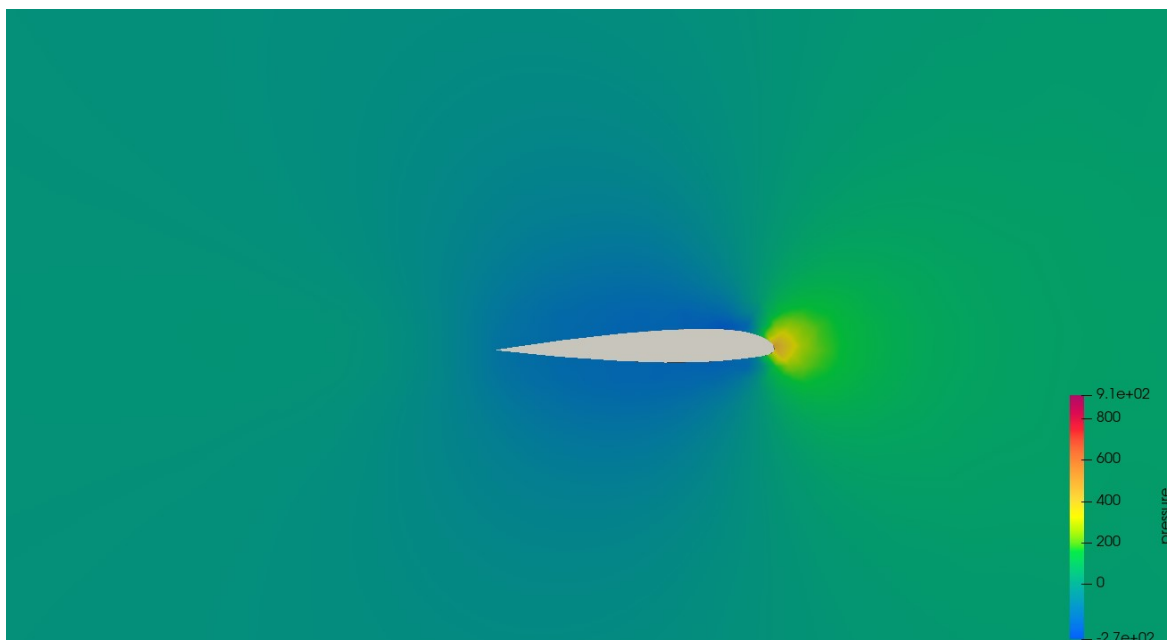
### SOLIDWORKS Flow Simulation

Drag to lift coefficient = 3.65 to 1



### Altair HyperWorks CFD

Drag to lift coefficient = 32.7 to 1



## 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.