Notes on Importing Airfoil Data Points into Ansys Fluent - Design Modeler
MEEG 332
Armani Batista
Ver. 1.0 - 3/14/2019

0. Go to airfoiltools.com and select an airfoil you want to do.

1. Click "Selig format dat file" to load the airfoil data points. You'll see a 2 columns of data corresponding to x and y values.

2. You'll need to change this format to something Fluent's Geometry tool can recognize. Create a text file using: notepad, notepad++, or any other plain ascii text file editor to create a .txt file. Another option is to use excel and save-as a .txt file.

3. If using excel, put the below at the top of the file for the first row. These are just comments, hence #
#group #point   #x   #y   #z

4. Next, input the airfoil dimensions in the #x and #y columns.

5. Populate the first column #group with all 1.

6. Populate the second column #point starting with 1, then increasing by 1 until the row just before the last x, y data values.
ie. If there are 100 data points, the #point column should be: 1,2,3, ..., 97, 98, 99

7. The very last row in the #point column should be 0. ie. The 100th data point should be 0.

8. Populate the #z column with all 0.

9. Save the file as a text file: ie. naca_airfoil.txt

10. Follow the steps on Cornell's CFD airfoil tutorial. Below are excerpts from their tutorial (R1).
10.1 Change the geometry properties from 3D to 2D
10.2 Right-click geometry and click Design Modeler
10.3 In the top menu bar, click Concept -> 3D Curve. In the Details View window, click Coordinates File and click the ellipsis to browse to a file. Browse to and select the airfoil file you just created. Proceed as usual.

References:
R1. Flow over an Airfoil - Geometry, Benjamin Mullen, Cornell University, 2019,
https://confluence.cornell.edu/display/SIMULATION/Flow+over+an+Airfoil+-+Geometry