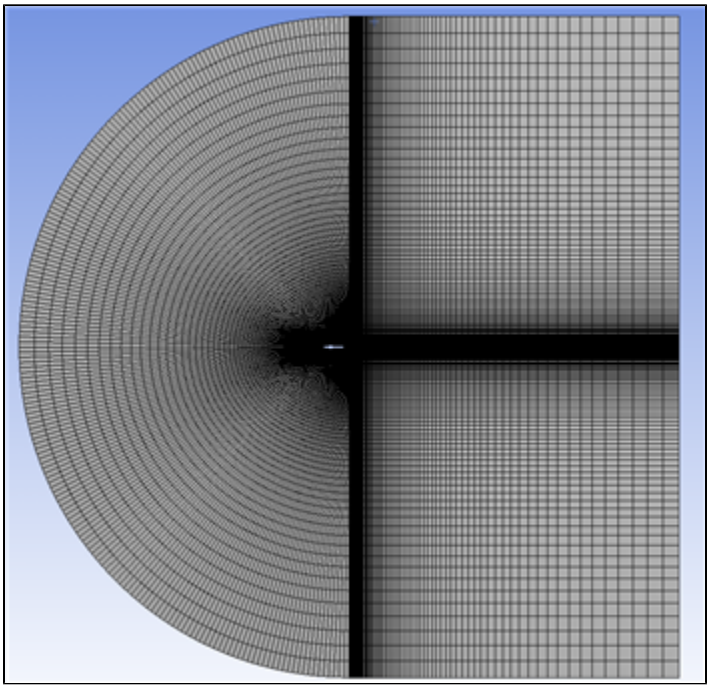
Flow over an Airfoil - Verification & Validation

Author: Benjamin Mullen, Cornell University

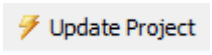
Verification & Validation

Verification

One of the ways we can verify our data is by refining the mesh. Open up the mesh, and increase the ***Number of Divisions*** for *Edge Sizing* and *Edge Sizing 2* to `100`. Click ***Mesh*** in the *Outline* window, and in the *Details* window, expand ***statistics***. The number of elements should now be 40000.



Click here to enlarge

Exit out of the mesher. First, right click ***Setup*** and select ***Reset***. Then click 

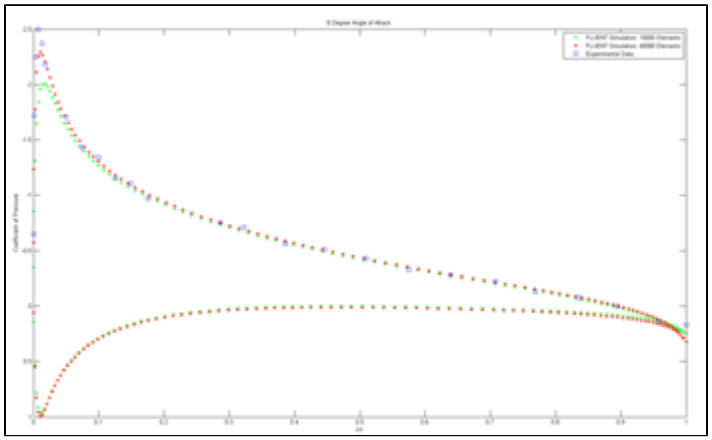
in the project schematic. Open up the solver, and solve the simulation using the same solver and boundary conditions (you'll have to input them again), but this time change the number of iterations to `5000`. Again, calculate the force coefficients and graph the pressure coefficient.

Validation

To validate our data, we can compare values to actual experiment. The drag coefficient coming from our inviscid model, however, cannot be compared to actual data. The drag coefficient in our model is theoretically zero.

	Unrefined Mesh	Refined Mesh	Experimental Data
Lift Coefficient	0.6315	0.6670	0.6630
Drag Coefficient	0.0122	0.0063	n/a

Below is a graph displaying the comparing Coefficient of Pressure along the airfoil for the experimental data and the CFD simulation. The data is from Gregory & O'Reilly, NASA R&M 3726, Jan 1970.



Click here to see an enlarged image

As we can see from the table and the graph, the CFD matches the data fairly well. There are inaccuracies from factors such as our assumption that the flow is inviscid, but we still managed to extract some meaningful information from the simulation. Click here to go to an exercise which demonstrates a more thorough approach to modeling this flow problem, which involves turning on the turbulence model equations and using a more refined mesh provided by NASA.

Go to Exercises

Go to all FLUENT Learning Modules