Inverse Airfoil Design

Utkarsh Chauhan
Dept. of Aerospace Engineering
Indian Institute of Technology, Bombay
Mumbai 400 076

Supervised Learning Project

under the guidance of

Prof. A.M. Pradeep

Abstract

In this report, optimal airfoil shapes are found through manipulation of the pressure distribution using XFOIL as solver and an optimization algorithm. The Airfoil geometries generated by this method is then compared to standard geometries from UIUC Database to measure the performance of algorithm and solver. In the report following method is used to generate NACA 0009 Airfoil from NACA 0015 Airfoil. Results indicate that using the design variables defining the pressure distribution in the inverse method has great potential for increasing the efficiency of airfoil shape optimization. High Fidelity solvers can be further used to solve for flow with high Mach number.

Contents

1 Introduction

Inverse methods of aerodynamics determine the geometry of the aircraft elements for a given pressure distribution and are a powerful tool of aerodynamic design. An Inverse Airfoil Design method includes a solver and an optimization algorithm.

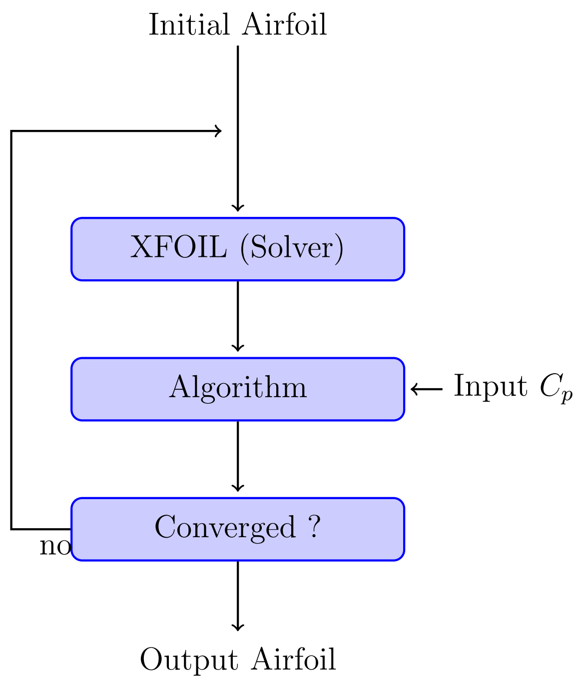

Figure 1: Inverse Airfoil Design

An Input Reference Airfoil is provided along with the C_p distribution of the desired airfoil. Solver is used to generate the C_p distribution of the input airfoil which is further compared with the algorithm to modify the coordinates of initial airfoil. Output comes out once the solution converges (error is less than desired).

2 Available Solvers

The following Solvers were tested to get the best performance keeping in mind the computation time and power.

2.1 Ansys Fluent

Ansys is one of the most powerful CFD solver commercially available. It allows an in-depth analysis of flow over an airfoil. On the other hand it needs a lot of computational power for one single iteration of Inverse design which makes it very hard to use. Also coupling Ansys with our own algorithm in Python is very hard.

2.2 OpenFOAM

OpenFOAM is also a high fidelity, Open Source CFD solver. It also comes with a python library PyFOAM to control OpenFOAM-runs and manipulate OpenFOAM-data. Problem in working with OpenFOAM is that it requires a lot of time to get familiar with and requires prior CFD knowledge for meshing etc. Due to time constraint we couldn't implement OpenFOAM with our optimization algorithm.

2.3 Xfoil

Xfoil is an Open Source low fidelity, fast and reliable solver using Panel method to solve flow over an Airfoil. XFOIL significantly reduces the computational requirements while retaining the ability to predict low Reynolds number flow. Being open source it is easy to couple XFOIL with our algorithm using python scripts.

Hence XFOIL was used as the final solver for the Design Method as we were dealing with fairly low Re for which the result calculated from both XFOIL and ANSYS was comparable.

3 Final Design

3.1 Step 1: Initial Airfoil

A custom python script was written to get NACA 0015 airfoil coordinates from UIUC Coordinate file and extract X-Y coordinates to feed XFOIL.

3.2 Step 2: Running XFOIL

A detailed python code is written to run XFOIL specify the flow conditions and extract the C_p distribution in a separate file to be further used.

3.3 Step 3: Interpolation

Since both the C_p distribution have different $\frac{x}{c}$, a separate linear interpolation function is ran to get C_p at same $\frac{x}{c}$ and compare both C_p distributions.

3.4 Step 4: Algorithm

Optimization algorithm is applied to the C_p distributions. Based on the L2 norm and flow properties coordinates of the given airfoil are modified and saved as a new airfoil which is again fed to XFOIL and the procedure is again applied.

On comparison once the $error \leq error_{desired}$ further iterations are stopped and output airfoil and its C_p ditribution is saved.

All the codes and result can be found here

4 Result

The code was tested to obtain NACA 0009 Airfoil where initial airfoil chosen was NACA 0015. The following Result was obtained:

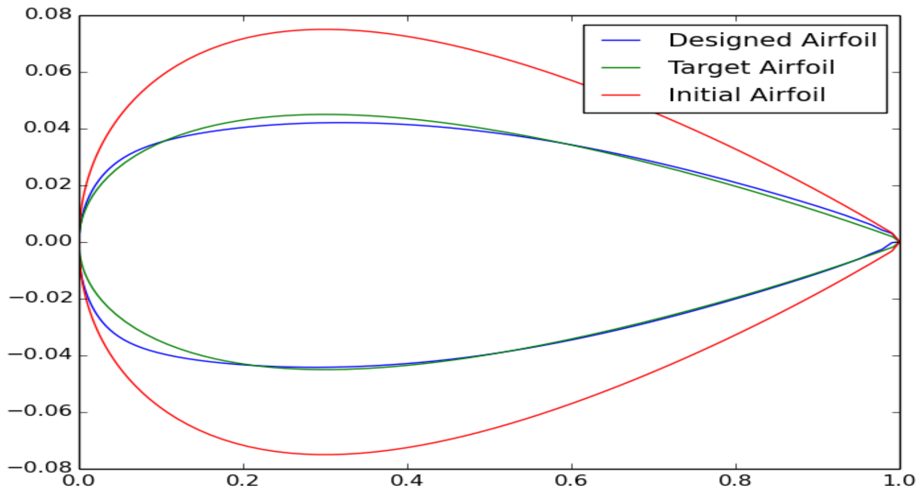

Figure 2: Airfoil Coordinates

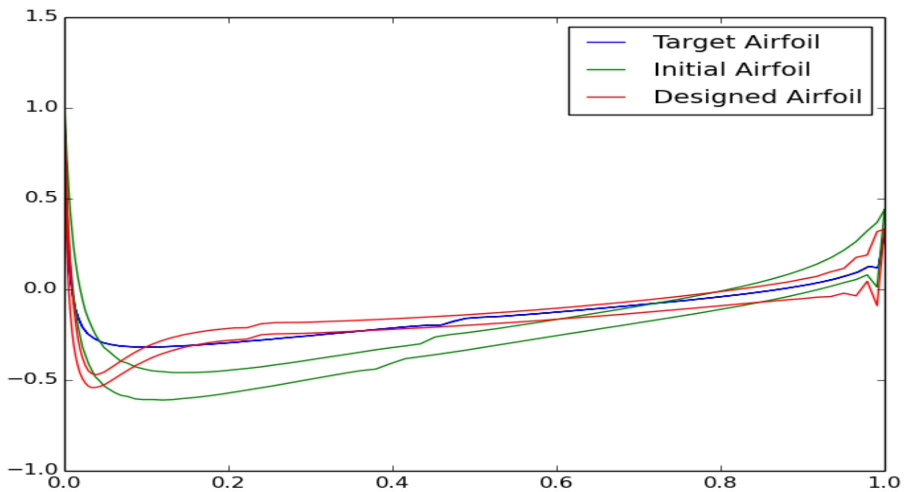

Figure 3: C_p Distribution

5 Conclusion and Future Work

As seen from the plots the algorithm work fairly well in case of symmetric airfoils in low Reynolds number. There are some anomalies on the leading and trailing edge of the the designed and target airfoil which can be improved in the algorithm but could not be done due to time constraint. The algorithm can also further be improved to generate cambered airfoils. Standard Optimization Algorithm like Genetic or PSO can be implemented to improve performance. High fidelity solvers like OpenFOAM or MSES can be implemented to perform well for high Mach number for Transonic Design.

6 Bibliography

- "A VISCOUS INVERSE DESIGN METHOD FOR INTERNAL AND EXTERNAL FLOW OVER AIRFOILS USING CFD TECHNIQUES" by Raja Ramamurthy, Benedikt Roidl and Wahid Ghaly
- XFOIL by Mark Drela, MIT
- Ansys by ANSYS, Inc.
- OpenFOAM by CFD Direct