Final Project Report
Aerodynamic characteristics of a real 3d flow around a finite wing

By

SUNIL KRISHNA SRIPERUMBUDUR, personnumber 841123-6139,
sksr@kth.se

Course Incharge
Lars Kjelldahl
INTRODUCTION

An airfoil is the shape of a wing or blade (of a propeller, rotor or turbine) as seen in cross-section.

The design and analysis of the wings of aircraft is one of the principal applications of the science of aerodynamics, which is a branch of fluid mechanics. Little modification in the airfoil has a direct impact on the performance of an aircraft.

In the present project, we design a 3D air wing and solve the flow equations in a CFD solver and study the characteristics features of the flow around a finite wing and the effect of the tip vortices that are caused by the difference of pressures between the lower and upper portion around the tips on an air airfoil. Better visualization of this vortex flow past an aircraft helps in optimizing the design of a wing.
DISTURBANCE OF FLOW PAST AIRFOIL

A lot of disturbance is created in the air when an aeroplane flies. It is through the study of these disturbance of the flow past the airfoil, lot of design considerations are done. Performance of the aeroplane is directly related to the size & shape of airfoil. A considerable difference is seen between the airfoil of the commercial airlines and the defense plane as most of the time better optimized airfoil leads to bad fuel consumption because of the huge drag and vice versa.

Design, Data Generation and Visualization

Some of the important works done on Airfoils have been studied before solving this problem. These papers served as a basis for understanding, designing and solving the flow problem however this my project is no way intended to validate my results with the results mentioned in the papers or to show discrepancy percentage of results with any of the research results. Details of the literature survey done has been included in the references list.

The Ansys workbench has a comprehensive list of software for doing various Structural and Fluid analysis and is also equipped with superior visualization capabilities which are in par with any other dedicated visualization software’s. This workbench has been used for designing, solving and visualizing the results.

Execution of the Project:

The project is executed in three phases.

- Pre-processing
  - Design of the model is done using Sumo-2.4.1 and Design Modeler
  - Meshing the model in Ansys workbench

- Solving
  - Solving using CFD package Ansys Fluent

- Post-Processing
  - Analyzing the results using various visualizations
  - Interpreting the results
PRE-PROCESSING

The design of the air wing is done using SUMO 2.4.1 and the wing is imported into Design Modeler to create far field.

A medium size mesh is used for meshing purpose. A refinement of the mesh is done near the wing region as it is the focus of our interest.
**SOLVING**

In the next phase, the various parameters of the flow is assigned to the model and also the boundary conditions are set and is solved using the CFD solver in Ansys Fluent.

The following are the details of the flow:
- Flow type: Inviscid Flow
- Solver Type: Pressure Based
- Discretization Method: Finite Volume Discretization
- P-V coupling: SIMPLE Algorithm
- Turbulence: Spalart-Allmaras (1 equation)

**POST-PROCESSING**

From the data that is generated, the flow is visualized using various visualization tools like contours, vector visualizations, particle tracing etc.

**Results**

![Fig: Static Pressure and Relative Pressure Contour of the entire domain](image)

From the above visualization, we cannot make any worthwhile inferences or analysis. This is a crude data result and we need more refine visualization techniques.
SLICING OF THE PRESSURE CONTOUR

We sliced (cut) the model at the appropriate portion of our interest in all the three planes and we can now clearly see the pressure distribution across the wing. From the XZ plan we can observe that the pressure difference is caused by the effect of tip vortex.
SLICING OF THE VECTORS

From the above Vector visualization in XY plane we observe the recirculation of the flow at the trailing end of the airfoil. At small angles of attack, air flows smoothly around an airfoil providing lifting force through the difference in pressure across the top and bottom of the airfoil. As the angle of attack increases, the lift produced by the airfoil increases as well but only to a point.
The tip vortices can be seen at the edge of the wing.

From the above figure, we can clearly see that there are tip vortices which influence the drag of the airfoil.
OTHER DISPLAY OPTIONS USED IN VISUALIZATION

Listed below are some of the other visualization tools used in post-processing phase.

Fig: 3D Wing Mesh in Ansys Fluent

Fig: 3D Wing vector of static pressure with mesh

Fig: 3D Wing vector of static pressure- Cone Options
By understanding the data through various techniques of visualizations, we can modify the shape and size of the air wing to prevent the tip vortices from influencing the drag which impacts the performance of the aeroplane.

From slicing and through the methods like volume rendering, we can have animation of the flow.
OTHER MODEL CONSIDERED- LIGHT JET EXECUTIVE

The Light Jet Executive has been modeled in SUMO 2.4.1 and the far field is modeled in Design Modeler.

Fig: Light Jet Executive CAD Model

Fig: Mesh of Light Jet Executive
Although the Light Jet Executive has been successfully modeled and meshed, there has been certain constraints due to which it could not be solved. Some of the major limitation were the problem with integrating the mesh of the far field with the Light Jet Executive.

Initial attempts to make the mesh very coarse were successful however solving the problem gave totally unrealistic results and was not stable.

CONCLUSIONS AND FUTURE WORK

• The air wing and domain is modeled, meshed and solved and various post processing visualization options has been used for better understanding & investigation of the flow.
• Visualization options such as slicing and volume rendering proved extremely useful for doing the investigation of flow pattern.
• A better refinement of the mesh and boundary mesh and implementing better solvers can give accurate results for the 3D wing which can be potentially be validated with experimental results.
• An adaptive mesh refinement near the high gradients especially close to the wing gives more accurate results.
• Refinement of mesh and better meshing options required for solving Light Jet Executive Model.

REFERENCES

5. Christopher P. Mellen, "Lessons from LESFOIL Project on Large Eddy Simulation of Flow around an Airfoil", University of Karlsruhe