Global Journal of Researches in Engineering: D
Chemical Engineering
Volume 14 Issue 1 Version 1.0 Year 2014
Type: Double Blind Peer Reviewed International Research Journal
Publisher: Global Journals Inc. (USA)
Online ISSN: 2249-4596 & Print ISSN: 0975-5861

Aerodynamic Characteristics of a Real 3D Flow around a Finite Wing

By Sanjay Kumar Sardiwal, Md. Abdul Sami, B. V. Sai Anoop, Syed Arshad Uddin, Gudipudi Susmita & Lahari Vooturi

MLR Institute of Technology, India

Abstract- This paper presents a new method of solution for the aerodynamics of finite-span wings, which overcomes the difficulties of the previous methods. A lot of disturbance is created in the air when an aeroplane flies. It is through the study of these disturbances of the flow past the airfoil, lots of design considerations are done. In this paper, 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.

Keywords: finite wing, real 3d flow, vortices, pressure differences, vortex flow.

GJRE-D Classification : FOR Code: 090101

AERODYNAMICCHARACTERISTICSOFAREAL3DFLOWAROUNDAFINITEWING

Strictly as per the compliance and regulations of :

Aerodynamic Characteristics of a Real 3D Flow around a Finite Wing

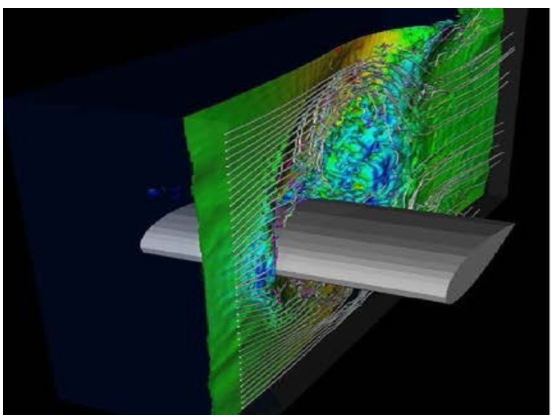
Sanjay Kumar Sardiwal [α], Md. Abdul Sami [σ], B. V. Sai Anoop [ρ], Syed Arshad Uddin [Ѡ], Gudipudi Susmita [¥] & Lahari Vooturi [§]

Abstract- This paper presents a new method of solution for the aerodynamics of finite-span wings, which overcomes the difficulties of the previous methods. A lot of disturbance is created in the air when an aeroplane flies. It is through the study of these disturbances of the flow past the airfoil, lots of design considerations are done. In this paper, 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.

Keywords: finite wing, real 3d flow, vortices, pressure differences, vortex flow.

I. Introduction

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



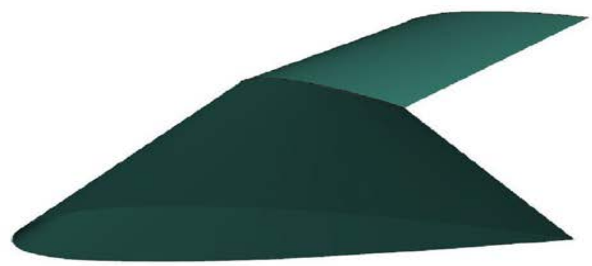


Figure 1 : Cross-section of a wing

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.

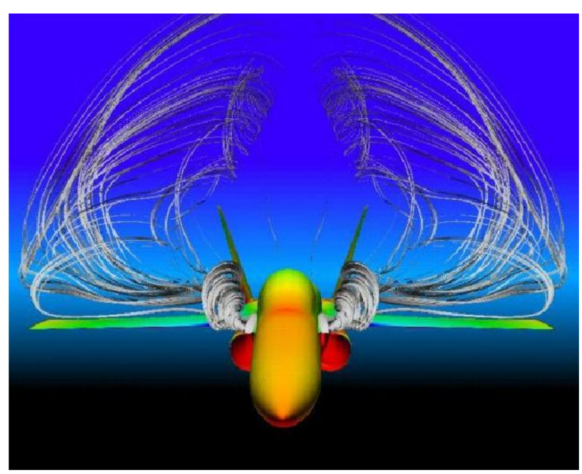


Figure 2 : Visualization of vortex flow past an aircraft

Author α σ ρ ¥ § : Department of Aeronautical Engineering, MLR Institute of Technology, Hyderabad. e-mail: sanjay.sardiwal33@gmail.com
Author Ѡ : Department of Aeronautical Engineering, Malla Reddy College of Engineering and Technology, Hyderabad.

Here, 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.

II. Disturbance of Flow Past Airfoil

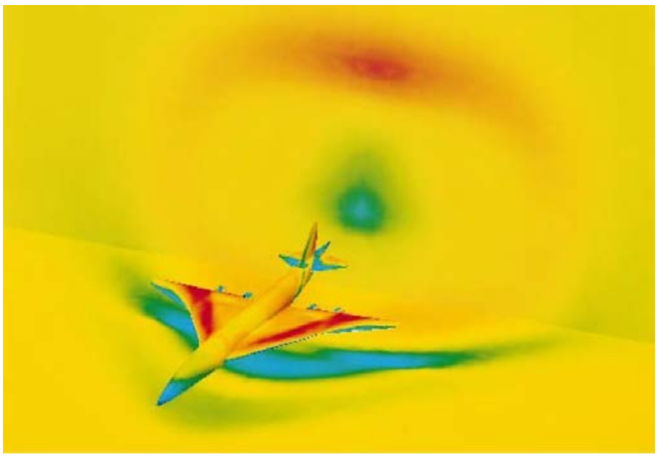


Figure 3 : Disturbances created on an aircraft

A lot of disturbance is created in the air when an aeroplane flies. It is through the study of these disturbances of the flow past the airfoil, lots 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.

III. Design, Data Generation and Visualization

Some of the important works done on Airfoils have been studied before solving this problem. This paper serves as a basis for understanding, designing and solving the flow problem.

A detail 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

a) 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.

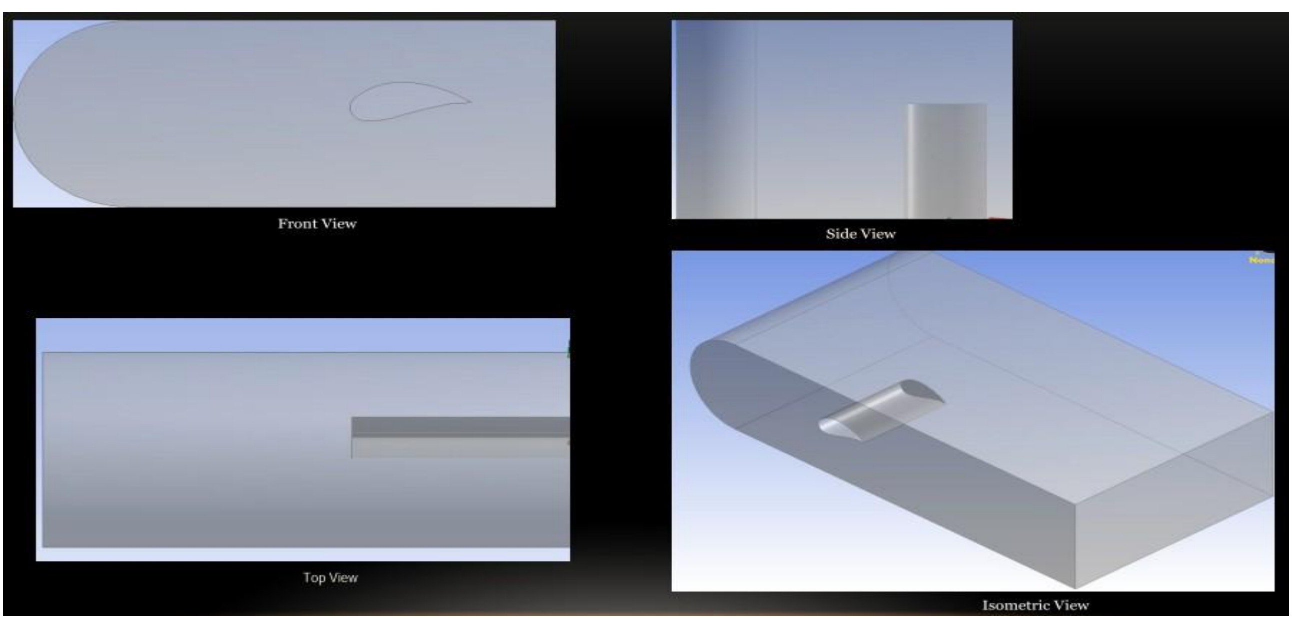


Figure 4 : CAD model of the 3d Air wing with the far field-

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.

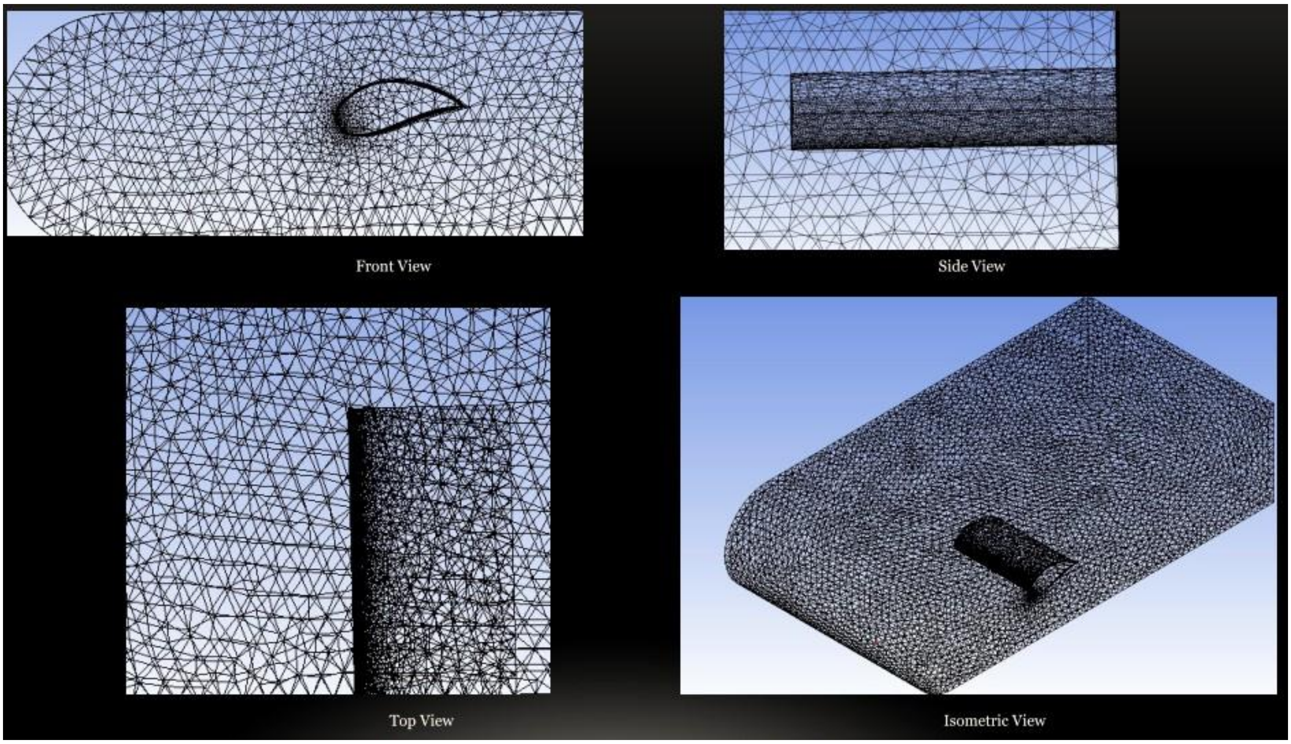


Figure 5 : Mesh of the entire domain

b) 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)

c) Post-Processing

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

IV. Results

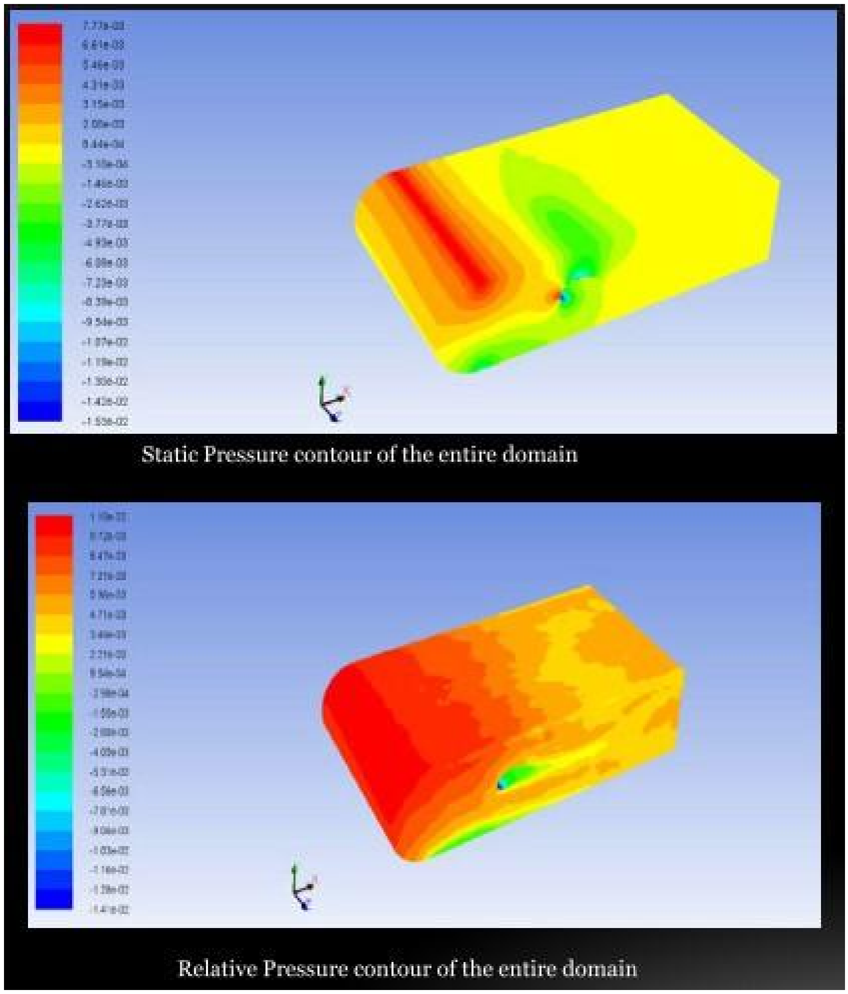


Figure 6 : Static Pressure and Relative Pressure Contour of the entire domain

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.

a) Slicing of the Pressure Contour

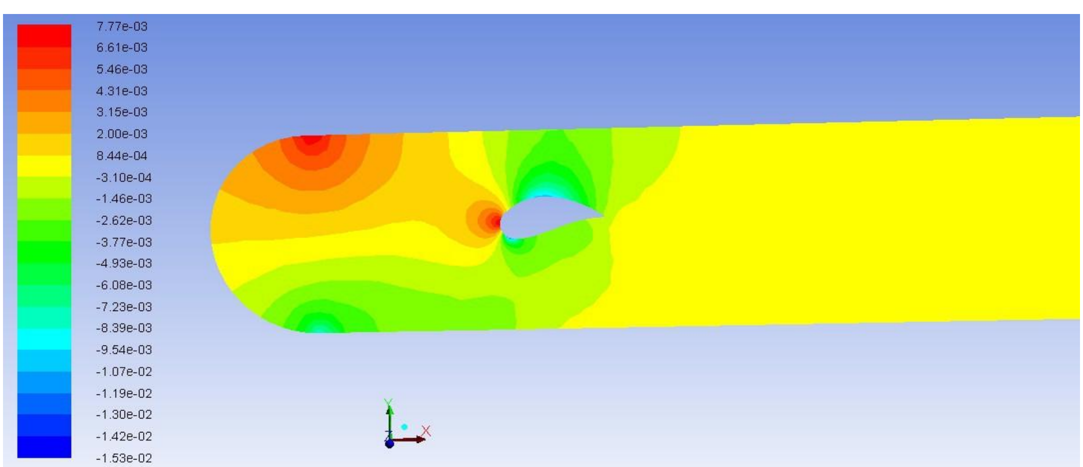


Figure 7 : Static Pressure contour from the XY Plane

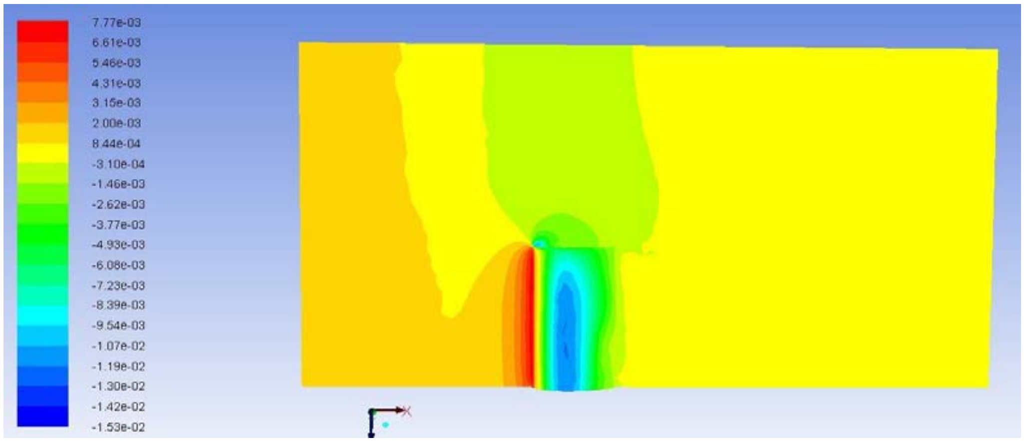


Figure 8 : Static Pressure contour from the XZ Plane

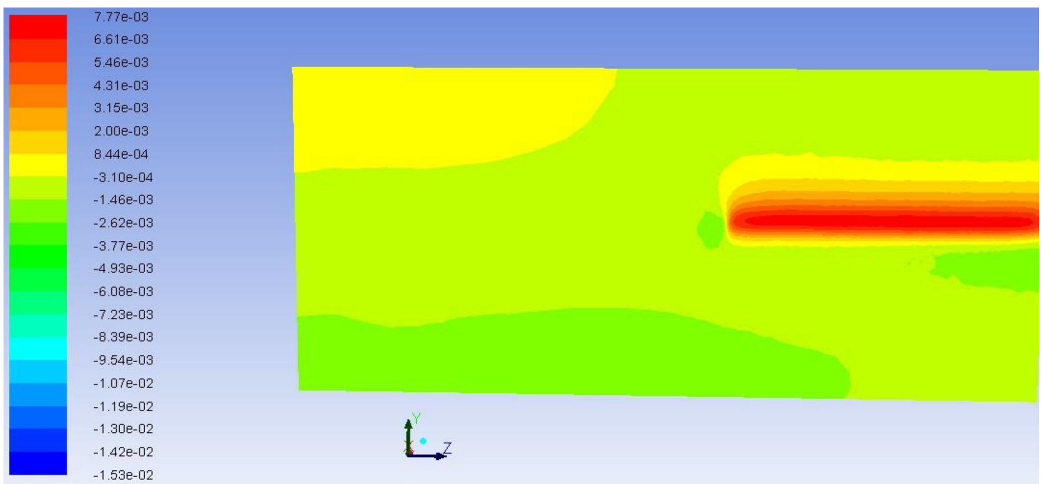


Figure 9 : Static Pressure contour from the YZ Plane

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.

b) Slicing of the Vectors

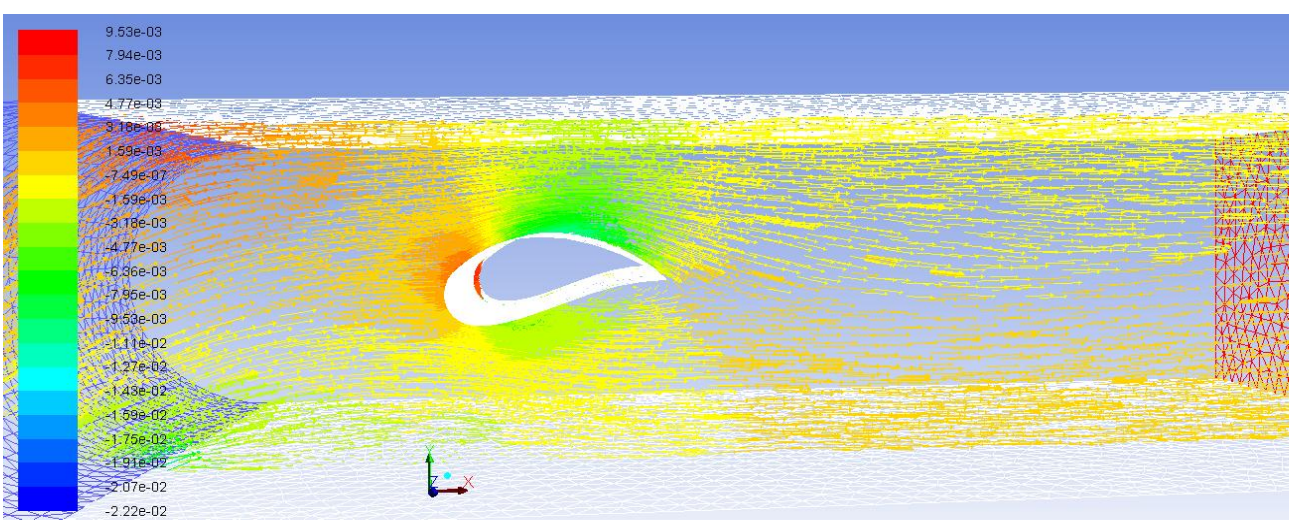


Figure 10 : Vector of the Pressure contour in XY Plane

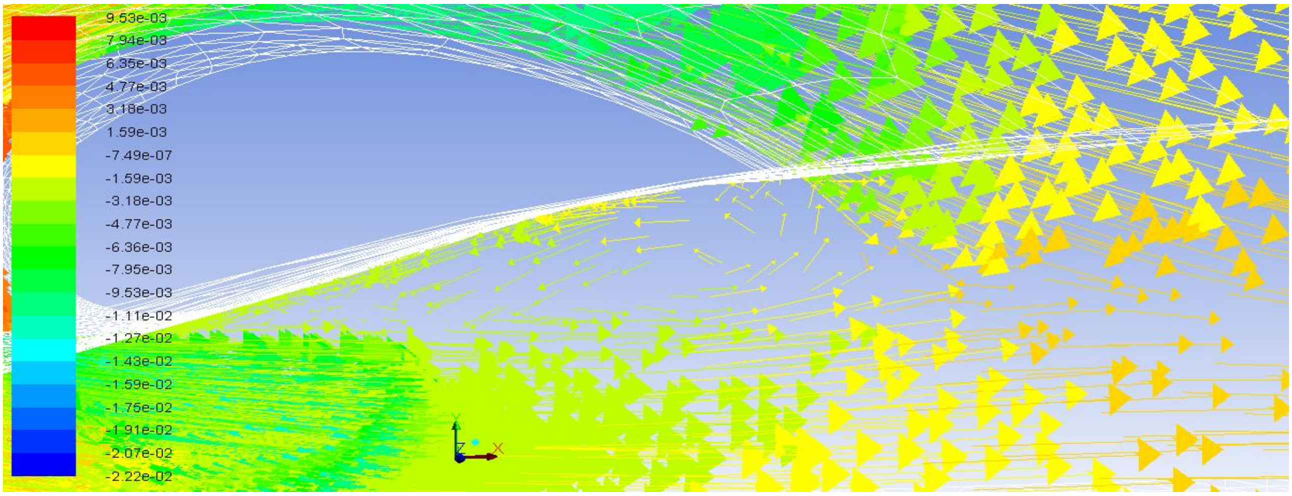


Figure 11 : Recirculation of the flow- XY Plane

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.

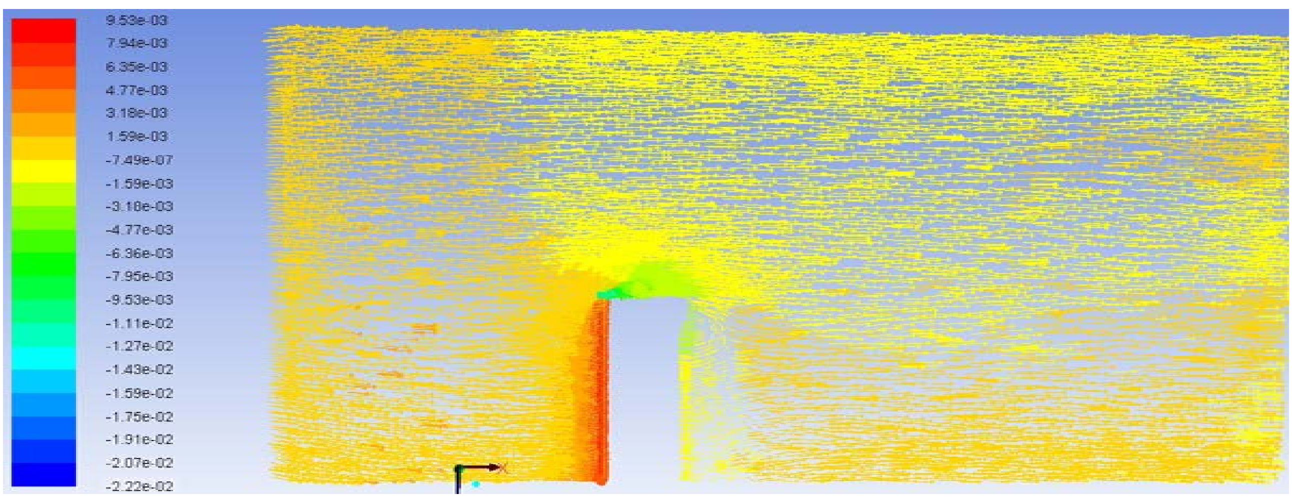


Figure 12 : Vector of the Pressure contour in XZ Plane

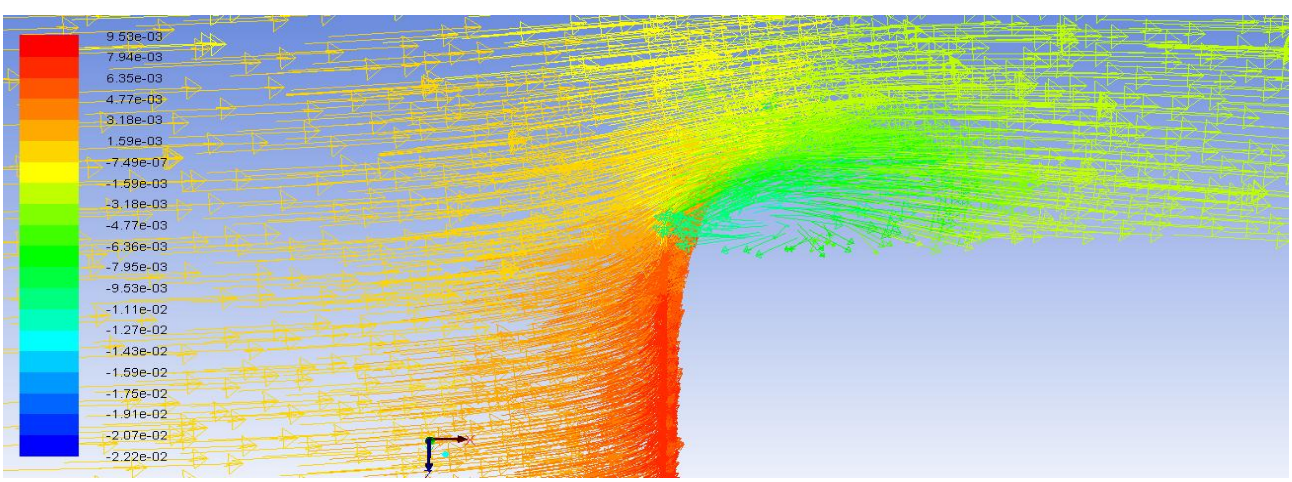


Figure 13 : Vector of the Pressure contour in XZ Plane

The tip vortices can be seen at the edge of the wing.

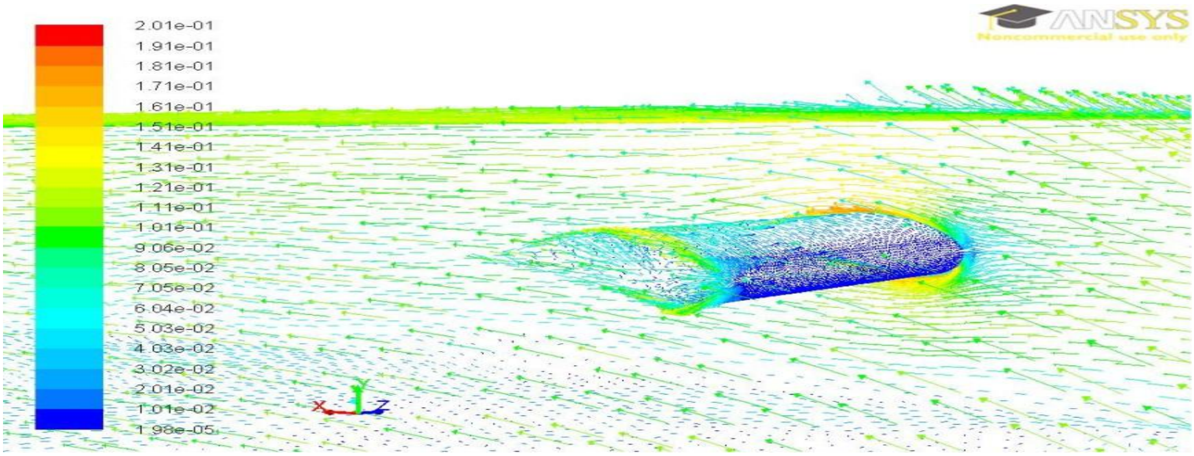


Figure 14 : Vector of the pressure contour of the 3d wing in Isometric view.

From the above figure we can clearly see that there are tip vortices which influence the drag of the airfoil

c) Other Display Options Used In Visualization

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

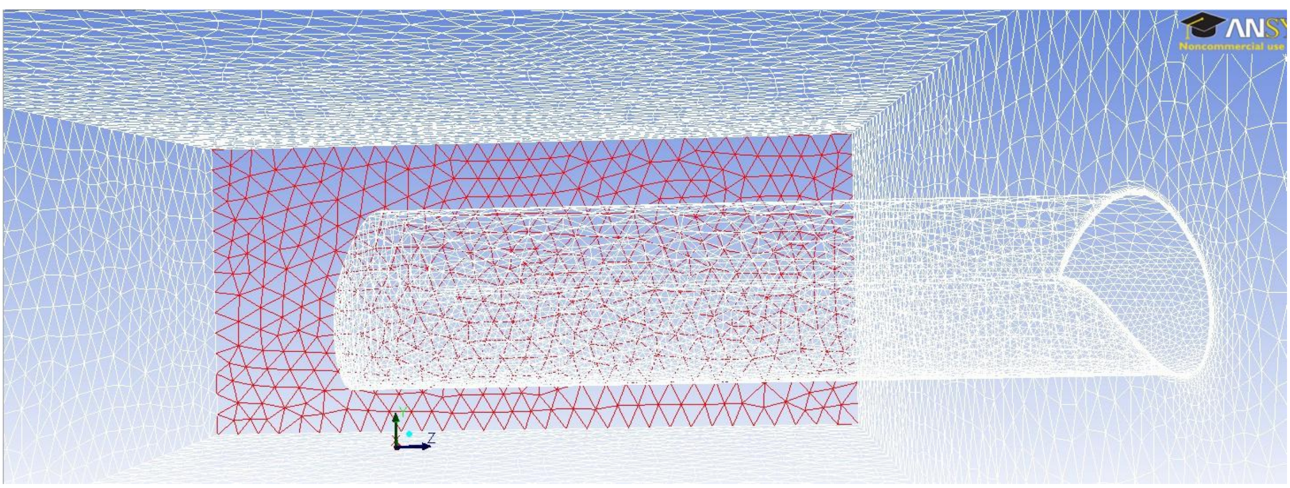


Figure 15 : 3D Wing Mesh in Ansys Fluent

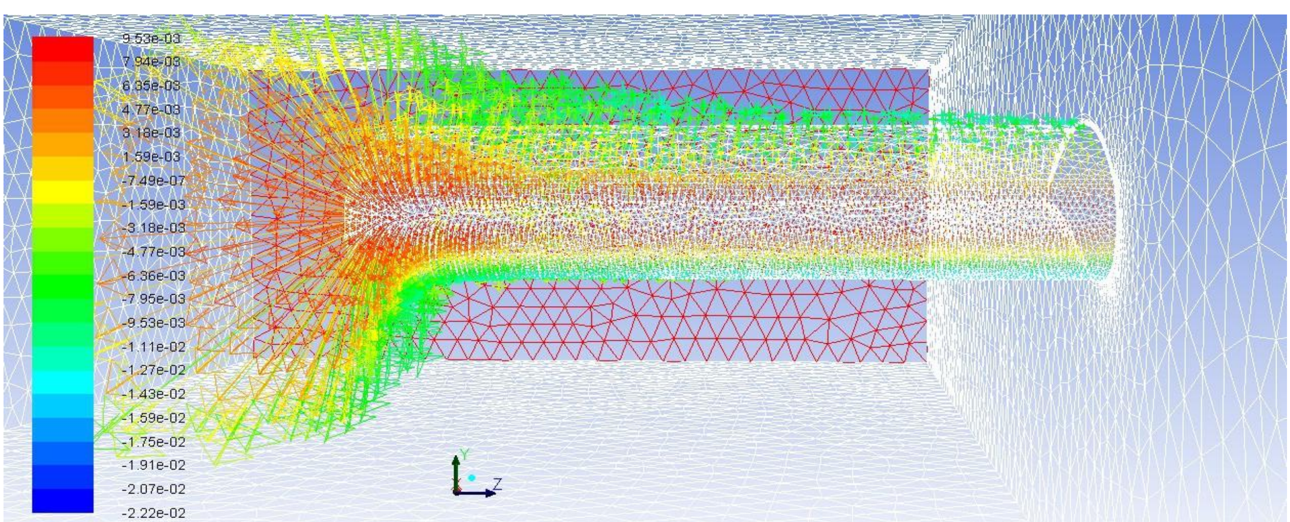


Figure 16 : 3D Wing vector of static pressure with mesh

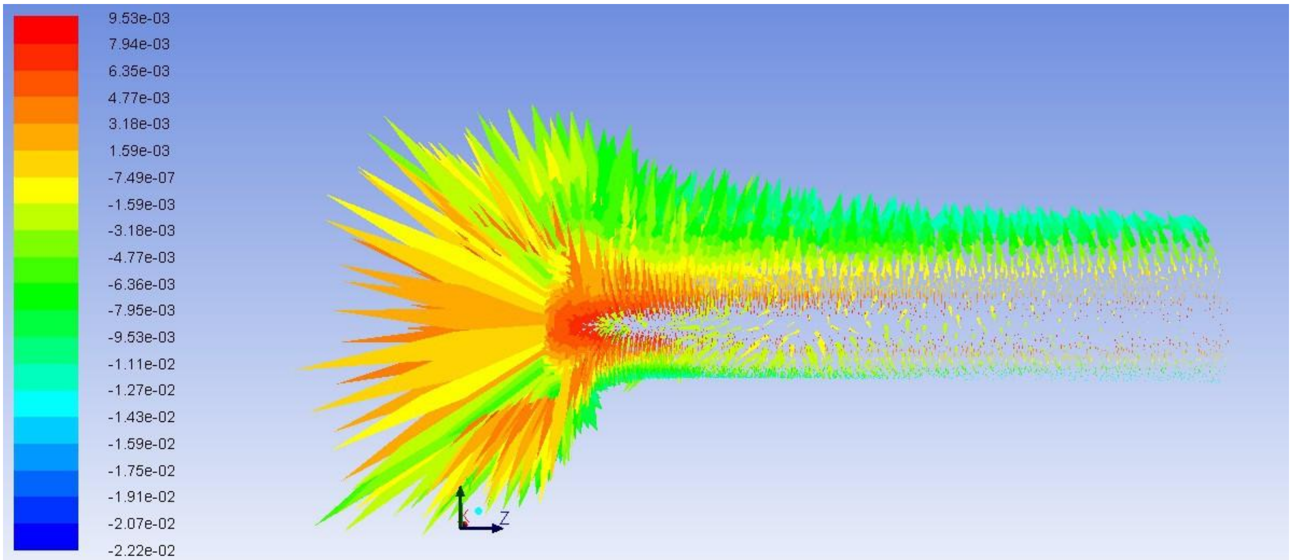


Figure 17 : 3D Wing vector of static pressure- Cone Options

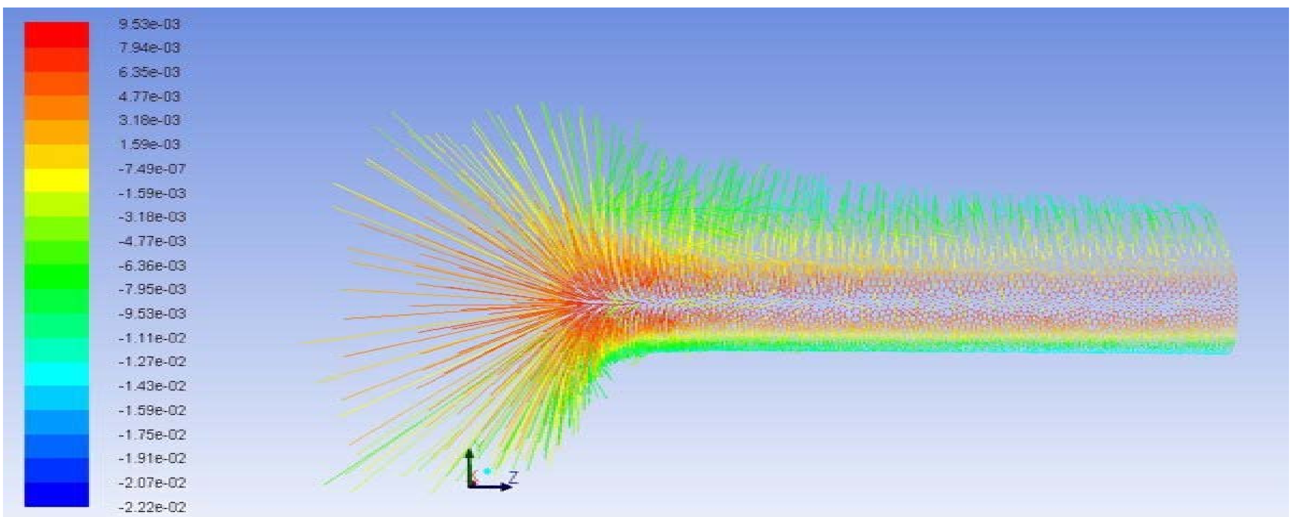


Figure 18 : 3D Wing vector of static pressure- headless option instead of arrows

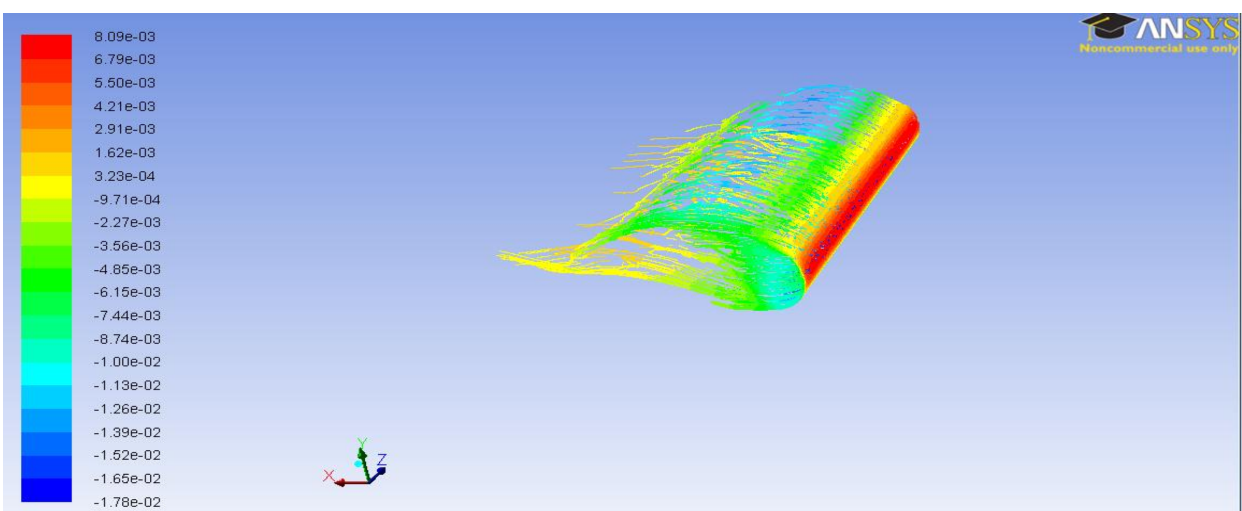


Figure 19 : 3D Wing Static pressure path lines

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.

d) 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.

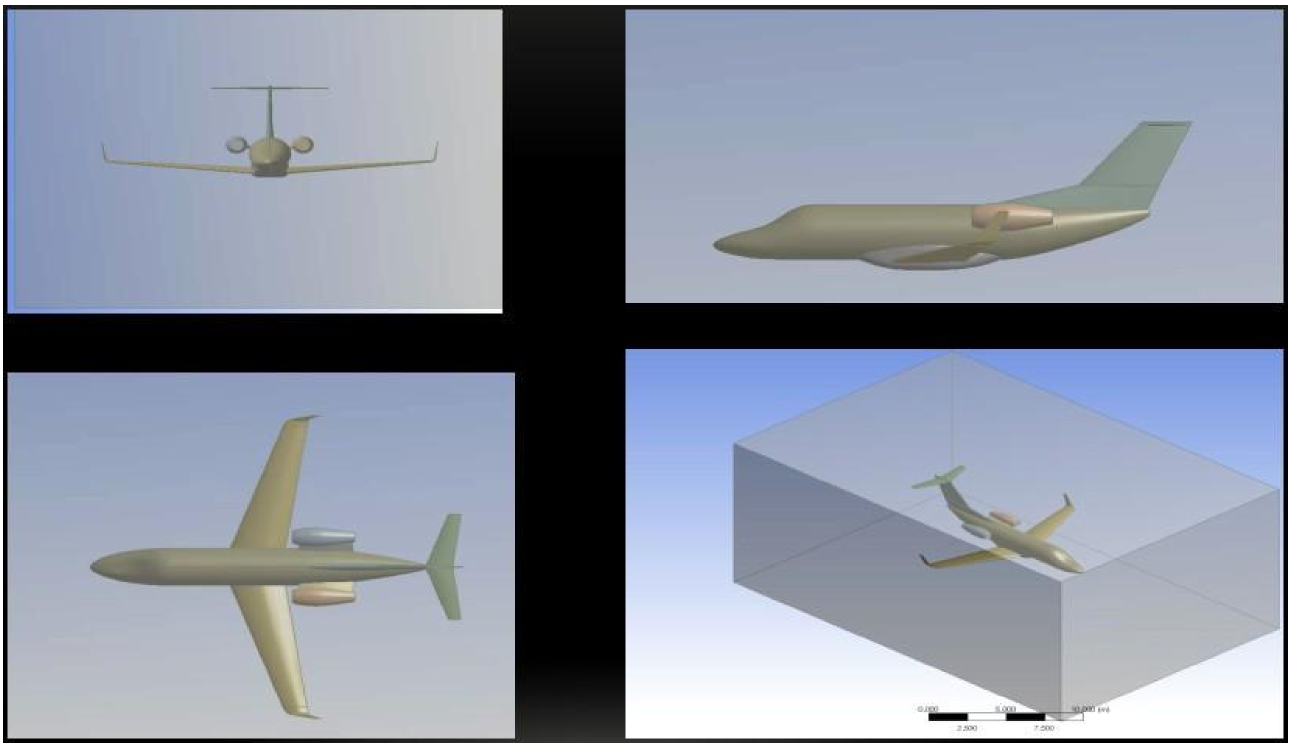


Figure 20 : Light Jet Executive CAD Model

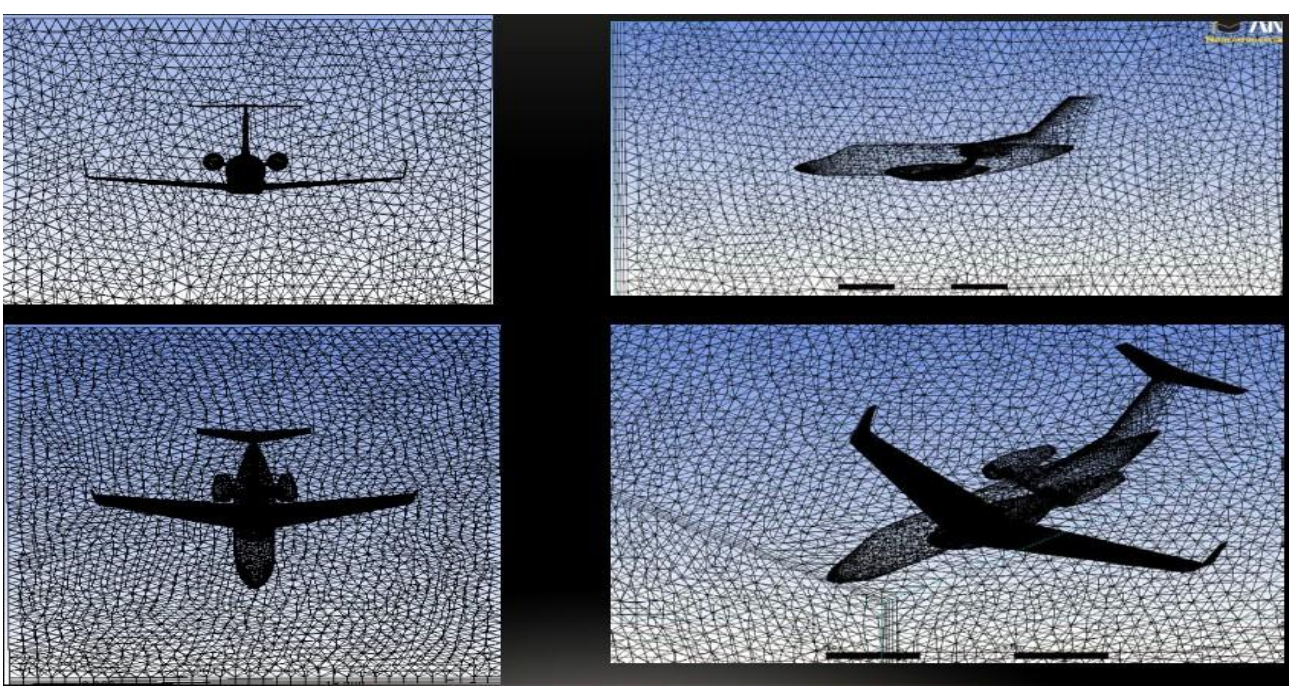


Figure 21 : Mesh of Light Jet Executive

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.

V. 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 Références Referencias

1. [Cessna] L.L Leisher et al, Stability derivatives of Cessna aircraft, Cessna Aircraft company 1957.
2. [Melin] Melin.T, Tornado, a vortex lattice MATLAB implementation for Linear Aerodynamic Wing applications", Masters thesis, Royal Institute of Technology (KTH), Sweden, December 2000.
3. [Moran] Moran, J "Computational Fluid Dynamics", Weily & Sons, 1984.
4. W. J. McCroskey, "A critical assessment of wind tunnel results for the NACA 0012 Airfoil", AMES Research centre.
5. Christopher P. Mellen, "Lessons from LESFOIL Project on Large Eddy Simulation of Flow around an Airfoil", University of Karlsruhe.
6. G. Martinat, "Numerical Simulation of the Dynamic Stall of a NACA 0012 Airfoil using DES and Advanced OES/URANS Modelling".