Modeling and Aerodynamic Simulation of TU Delft's Flying-V Using *ANSYS Fluent* and *COMSOL Multiphysics*

SRUJAN PANDYA[1] **AND NAVNEET KAUR**[1]

[1] *B.Tech, Department of Mechanical Engineering, IIT Gandhinagar, Gandhinagar, Gujarat, India, 382355*

Compiled May 14, 2021

This paper focuses on the modeling and aerodynamic simulation of the novel airplane design Flying-V, originally designed by TU Delft and its comparison with the available literature on airplane simulations. The paper first introduces the simulation of a NACA 0012 airfoil and analyses the aerodynamic pattern around the airfoil. Using this fundamental object as a reference for understanding the velocity magnitude, the pressure contours and the lift and drag coefficients (with different angle of attacks) associated with an aerodynamic profile, the Flying-V model is compared to another commercially active airplane Air India F16 with respect to the parameters mentioned above.

1. INTRODUCTION

The simulation of various airfoils is common in literature. The paper discusses introduces some fundamental aerodynamics concepts to the reader, providing definitions to the ubiquitous terms of aerodynamics. The first major modeling is of the most common NACA 0012 airfoil which has been tested for the simulation with zero angle of attack. The angle of attack is entered as a parametric sweep and the streamlines will be different with different angles of attack. For the comparison with 3D simulations, the angle of attack is taken to be zero and the trends of the velocity magnitude and pressure are noted. This is followed by the simulation of Air India f16 model, which is taken as a reference 3D model for Flying-V as well as for verifying the results with the NACA 0012 airfoil before the main simulation on Flying-V. TU Delft's Flying-V is a novel approach to aircraft design and the model has been replicated in this paper to find approximate solutions to the simulation of Flying-V itself. The CAD model was designed on Autodesk Fusion 360 and then imported in COMSOL Multiphysics for meshing and simulations. RANS turbulence model is used for NACA 0012 Airfoil and Flying-V and both these simulations have been performed on COMSOL Multuphysics, while the simulation of Air India F16 is performed on ANSYS Fluent.

A. Definitions

Lift: It is a force that keeps the aircraft in motion, opposing the weight of the aircraft. Lift is majorly affected by the speed and inclination (angle of attack) of the aircraft with air flow. The pressure difference across the wings produce the lift. The air flowing over the wing will cross the wing at a faster velocity than the air flowing under the wing. Therefore, the pressure on top of the wing is greater than the pressure at the bottom of the wing, according to Bernoulli's principle. This is the difference that produces lift. The mathematical expression for lift is given by:

$$Lift = \frac{C_L \times \rho \times v^2 \times A}{2} \tag{1}$$

Where C_L is the coefficient of lift, ρ is the density of air (fluid in general), v represents the velocity of the aircraft and A represents the wing area. In practice, an aircraft producing lift is a complex process, and the calculation of coefficients of lift and drag depend on various design parameters like the

Angle of Attack: It is the angle at which the airfoil (or wing) is inclined with the air flow. A small angle of attack will not be sufficient to produce lift at a low speed. A high angle of attack will give a large lift when the speed of the object is constant. However, if the angle is too high, the ability to lift will decrease since the drag will increase with a larger angle. Thrust is provided to the aircraft in order to overcome the drag induced due to the speed of the aircraft.

Drag: It is the resistance felt by the aircraft, which depends on the size and shape of the aircraft. It opposes the forward motion of the aircraft, which reduces the overall efficiency of the aircraft in flight. It increases with increase in density and with the square of the velocity.

Turbulence: It is a state of disorganized chaotic fluid motion characterized by a hierarchy of length scales. It is always time-dependent and always 3-dimensional. It occurs at high Reynolds number ($Re = \rho v L/\mu$). The geometry influences the larger ed-

Fig. 1. Caption

dies, while the smaller eddies are more universal. The smallest eddies are influenced by the viscosity of the fluid.
Airfoil: The cross-sectional shape of a wing. **Leading Edge:** The foremost edge of an airfoil.
Trailing Edge: The rear edge of an airfoil.
Camber: The convexity of the curve of an airfoil from the leading edge to the trailing edge.
Chord: An imaginary straight line connecting the leading edge to the trailing edge.
Pressure Contours: A line joining points on a diagram that have the same pressure value.

In any of the wing designs, the major goal is to increase the Lift to Drag Ratio (L/D) which is indicative of lower resistance to the aircraft body.

2. MODELING AND SIMULATION OF NACA 0012 AIRFOIL

The NACA 0012 design represents a symmetrical airfoil, where the 00 indicates no camber and the 12 indicates the airfoil has 12% thickness to chord length ratio.
For $0 \le x \le 1$, the *ideal* NACA 0012 airfoil is given by the mathematical expression:

$$y = \pm\, a_1 [a_2 \sqrt{x} - a_3 x - a_4 x^2 + a_5 x^3 - a_6 x^4] \tag{2}$$

Where $a_1 = 0.594689$, $a_2 = 0.298223$, $a_3 = 0.127125$, $a_4 = 357908$, $a_5 = 0.291985$ and $a_6 = 0.105175$.

COMSOL Multiphysics was used to obtain the results for NACA 0012 airfoil, which are further explained and illustrated in the following sections.

A. Reynolds-Averaged Navier-Stokes (RANS) Turbulent Model

It is the most commonly used turbulence model in engineering. At large Reynolds numbers when the range of vortices is high and there is a dynamic chaotic motion, the RANS model is used to calculate an ensemble average both in space and time, where the intricate flow structures becomes clear with zooming in space and the chaotic behaviour becomes clear with moving forward in time.
The numerical simulation in this model finds the effect of turbulence on the mean fluid flow of the system. There are some additional terms added in the original Navier Stokes equation in this model, which are called the Reynolds's stress terms.
All RANS models have some limitations with respect due to the assumptions taken while modeling them, however the RANS models are the most common and most exploited models in industry.

B. Simulation Results

Fig. 1,2 and 3 show the plots for velocity magnitude, streamlines and pressure contours for a zero angle of attack on the airfoil.
The plots show that the pressure is highest and the velocity is lowest at the leading edge of the airfoil. The flow then symmetrically divides and meets again at the trailing edge where the magnitude of velocity is again lower than the inflow velocity.

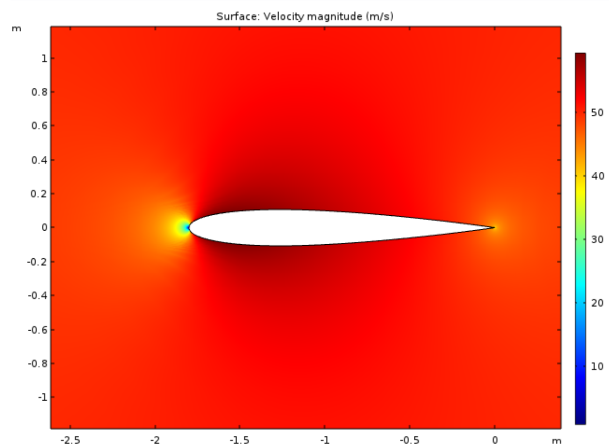

Fig. 2. Velocity Magnitude Around NACA 0012 Airfoil

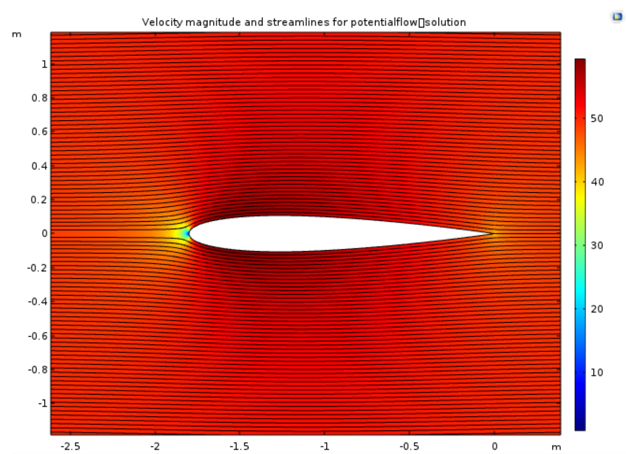

Fig. 3. Streamlines Around NACA 0012 Airfoil

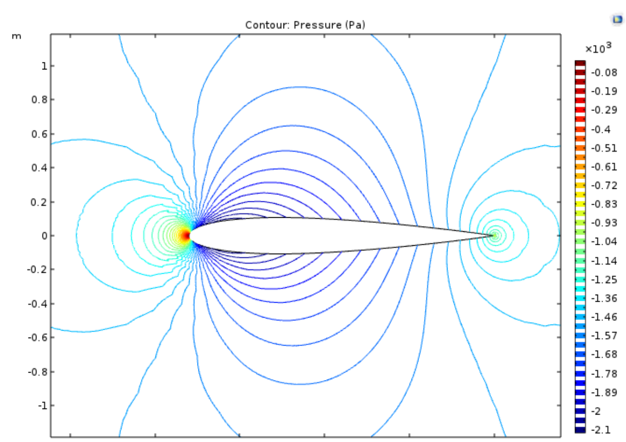

Fig. 4. Pressure Contour Around NACA 0012 Airfoil

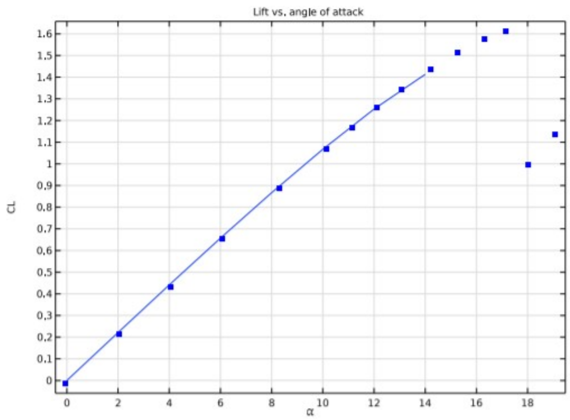

Fig. 5. Coefficient of Lift v/s Angle of Attack for NACA 0012 Airfoil

3. SIMULATION OF AIR INDIA F16 MODEL USING ANSYS FLUENT

After the 2D simulation of NACA 0012 airfoil, the Air India f16 airplane model was taken for 3D surface simulation. The model is commercially available and only the simulation part has been worked upon on . The objective is to set a comparison between the 2D results for the airfoil and the 3D results for model f16, as well as setting the 3D results as a reference for the simulation results of Flying-V which are followed up in the next few sections.
For this particular simulation, we have taken a viscous, Reynolds Stress model, with low *Re* and shear flow corrections.
The parameters included in this simulation are listed below.

Name	Value
C1-PS	1.8
C2-PS	0.52
Alpha*_inf	1
Alpha_inf	0.52
Alpha_0	0.105
Beta*_inf	0.09
Beta_i	0.072
R_beta	12
R_k	12
R_w	6.2
TKE Prandtl Number	2
SDR Prandtl Number	2

Table 1. Model Constants

The meshed model of Air India F16 is shown below:

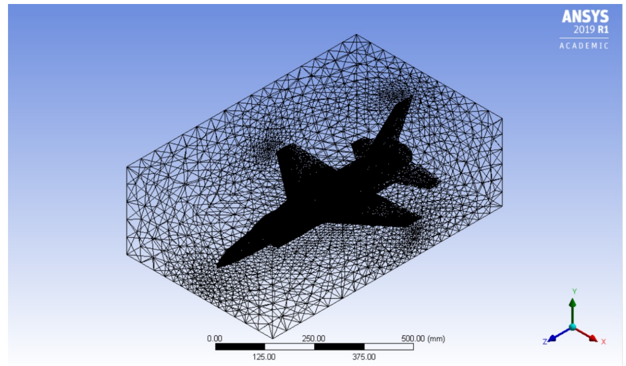

Fig. 6. Meshed Model of Air India F16

The results of the simulation (velocity magnitude and pressure magnitude) on F16 model are shown in the following images:

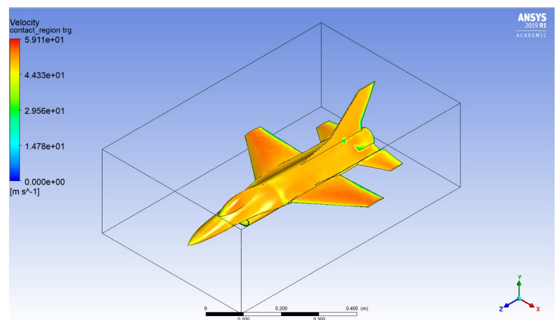

Fig. 7. Velocity Magnitude plotted over the Surface or f16

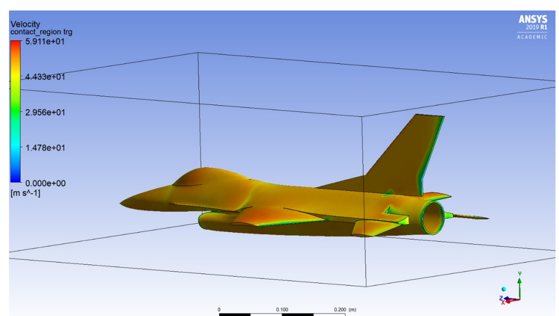

Fig. 8. Real View of the Surface Simulation of f16

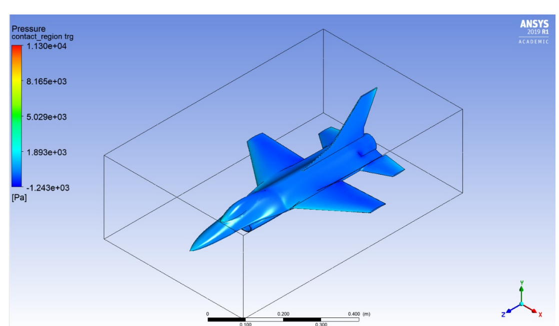

Fig. 9. Surface Simulation of Pressure over f16

The results show that the magnitude of velocity is minimum when the flow encounters the streamline shaped parts of the aircraft. The similar trend was shown in the 2D simulation where the flow (and the streamline) breaks at the streamline shape where the magnitude of velocity is lowest and consequently, the magnitude of pressure is highest.

4. MODELING AND SIMULATION OF FLYING-V

A. Modeling - CAD Model on Autodesk Fusion 360

The CAD software used for the design replication of Flying-V was Autodesk Fusion 360. Note that the design is not an exact replica but an approximate model of Flying-V. The images below show the CAD model used for simulations.

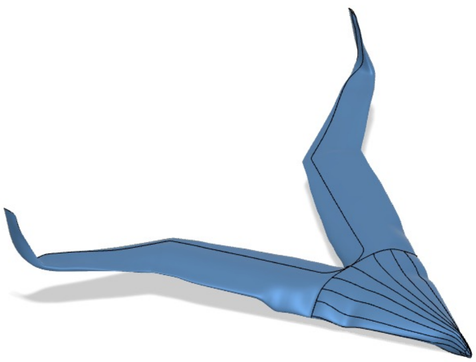

Fig. 10. Isometric View of Flying-V CAD Model

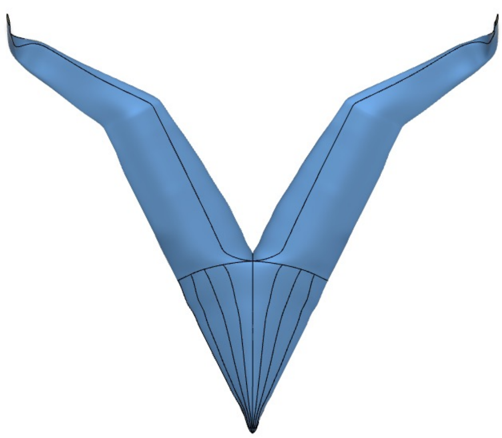

Fig. 11. Top View of Flying-V CAD Model

With COMSOL Multiphysics and RANS turbulence model, the objective was to get the results for surface simulation of velocity magnitude and pressure magnitude. The results for velocity magnitude were plotted; however the pressure magnitude plot did not have any physical meaning, a result of some error in the software.
The figure below shows the meshed model of Flying-V:

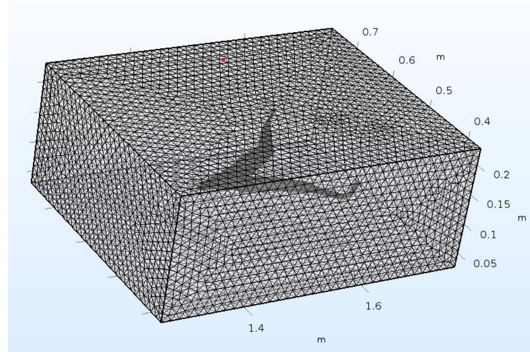

Fig. 12. Meshed Model of Flying-V

The result for velocity magnitude over the surface of Flying-V are shown below:

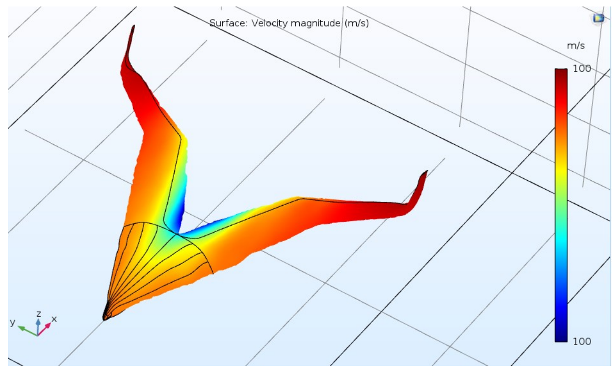

Fig. 13. Velocity Magnitude Over the Surface of Flying-V

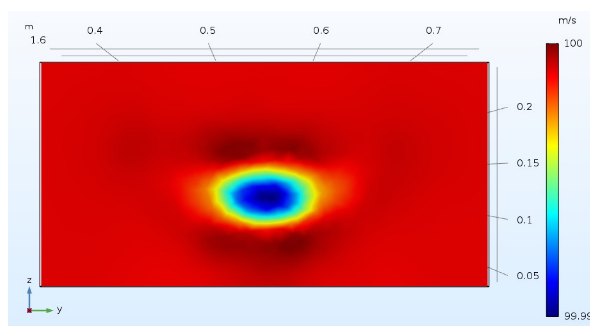

Fig. 14. Cross-Sectional View of Velocity Magnitude Over the Surface of Flying-V

From the results above, the velocity magnitude increases outwards radially and since the pressure is high where there is low velocity, the pressure on the rear bottom would be the one with the greatest pressure, hence the lift would be possible even with a zero angle of attack. Also, the lower magnitude of velocity is located near the front and middle part of the model, and the higher magnitudes of velocity are found near the two ends of the airplane. This means that the design naturally helps the aircraft to lift the nose region more easily and therefore, help steer the aircraft in an upward motion.

5. LIMITATIONS AND FUTURE WORK

The Flying-V model, although meshed properly, showed erroneous results in pressure magnitude and wall resolution of the model.
Thus, the future work includes the plots of pressure magnitude (majorly) and wall resolution.
The lift and drag coefficients in case of the two 3D models are very difficult to extract from the simulation results, since the calculation has to be manually done and there are a lot of variables/parameters involved in the model, unlike the 2D case of NACA 0012 airfoil. This can also be considered as part of the future work.
Thus, there are certain limitations to the implementation of this model and the results needs to be furnished before presenting as a concrete model.

6. REFERENCES

[1] Immonen, Eero, "2D shape optimization under proximity constraints by CFD and response surface methodology", Institute of Technology, Environment and Business at Turku University of Applied Sciences, Elsevier, September 21, 2016, Finland.