International Journal for Research in Engineering Application & Management (IJREAM)
Volume XX, Issue XX, MMM YYYY.

CFD Analysis of Typical Civil Aircraft to Study Drag Variation.

Vinay P. Chavhan, Student, Aeronautical Engineering, AnnaSaheb Dange College of Engineering & Technology Ashta, Sangli, India, vinaychavhan99@gmail.com

Dr. Devabrata Sahoo, Assistant Professor, Aerospace Engineering Department, MIT School of Engineering, MIT ADT University, Pune, India, Devabrata.sahoo@mituniversity.edu.in

Abstract: To determine and analyze the different aircraft performance parameter. Major studies have happened so far, but still to increase the efficiency of aircrafts many different techniques and research are going on. One of such study and research area is drag reduction. Drag is one of the important parameter which effects efficiency of aircraft. To study the drag reduction of aircraft a computational comparison study of a typical civil aircraft model is done at 0 and 3 degree angle of attacks at a velocity of 200m/s. It is known that, as angle of attack of the aircraft increases the flow over the surface of the fuselage starts to separate resulting in larger wake zone, which eventually increases the drag of aircraft. In present study the flow over the clean aircraft at 0 and 3 degree is first analyzed and after that the roughness is added to the surface, to see the effect of adding roughness on aircraft drag

Keywords — Angle of Attack, Drag Reduction, Fuselage, Roughness, Separation Point, Wake Zone,.

I. INTRODUCTION

Presently in today's aviation industry there is greater need of more efficient aircrafts. Hence in order to design and implement such economically favorable aircrafts, advancement of aircraft surfaces, drag reductions technologies has been paving their way in the aviation sector. And in designing of such aircrafts, different performance parameters has to be analyzed. One of which is the Drag of aircraft. According to J. Reneaux skin friction drag and lift induced drag of any typical civil aircraft contributes around half and one third of total drag value, and different drag reduction technologies with surface modifications are used on today's commercial aircrafts [1].

There are different types of drag that act on the aircraft as resistance to its motion. In following figure 1 different drag types are mentioned. It can be understood that around 45%

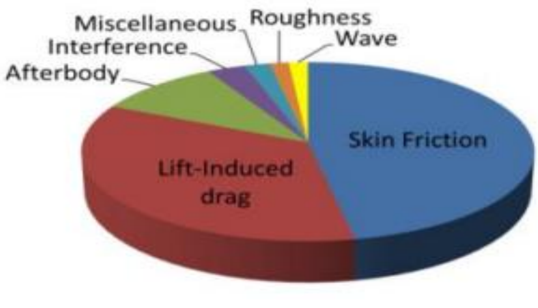

Figure 1 Drag Breakdown of a Typical Civil Transport Aircraft [7].

of total drag is contributed by Skin friction drag which is followed by lift induced drag. This drag types can be minimized by adapting surface modifications. The afterbody drag is generated due to wake zone formation behind the body, and this can be reduced by keeping flow intact over the surface. Figure 2 illustrates the similar scenario in case of flow around an Airfoil. Where, on smooth surface, flow separates at a point, called separation point. And by adding dimples to smooth sphere, the flow turbulence delays separation of flow, thus resulting in low drag value.

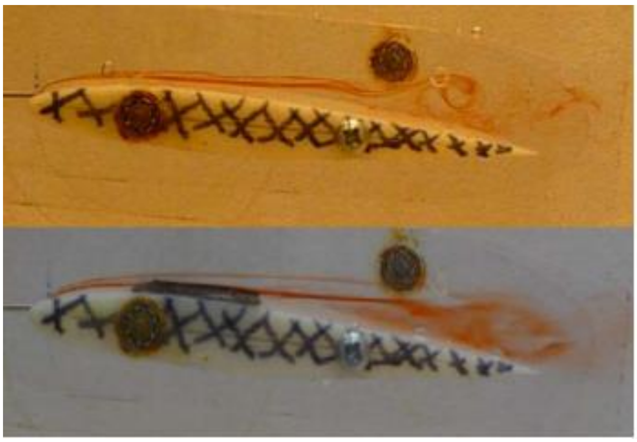

Figure 2 Comparison of flow over Smooth (Up) and Dimpled (Down) Airfoil [6].

The location of dimples is also important to study the flow behavior [6]. Different Dimple shapes can also be used to see the changes in flow behavior [5].

II. RESEARCH METHODOLOGY

In order to understand the real time flow behavior or performance of any device, vehicle or aircraft in a fluid flow with more approximate results computational fluid dynamics approach is the reliable testing platform for the parametric and comparison study.

A. Aircraft Model

Typical aircraft model used for the CFD simulation, along with the dimensions is shown below in figure 4

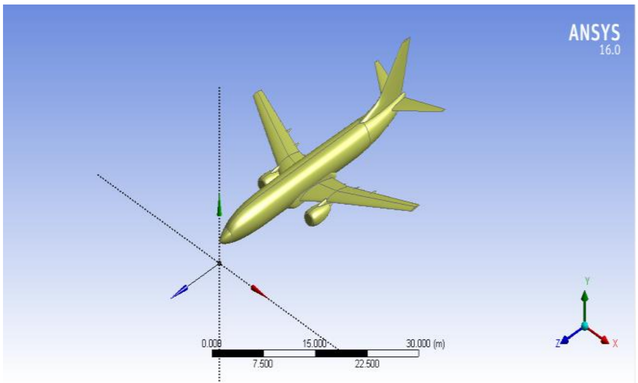

Figure 3 Typical aircraft model used for CFD simulation.

Dimensions:
Span (b) = 32.4m
Diameter (D) = 3.90m
Length (l) = 38 m

B. Validation

The software validation is done by comparing the values of Coefficient of lift and drag for 0 and 6 degree angle of attack, from paper entitled as, “CFD Analysis of Airfoil”, [9]. The pressure contours for both cases is shown below with comparison of results.

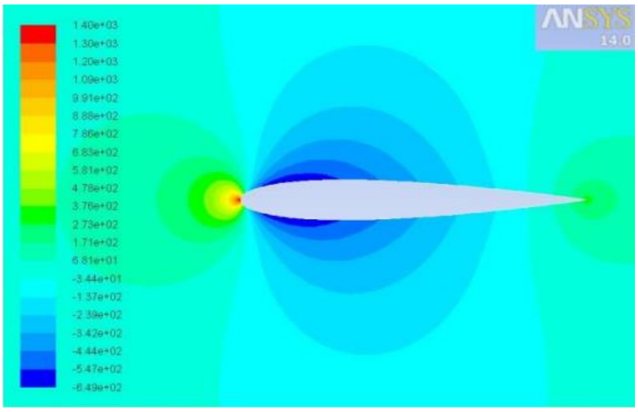

Figure 4 Static Pressure contour at 0 degree AOA for NACA0012 [8]

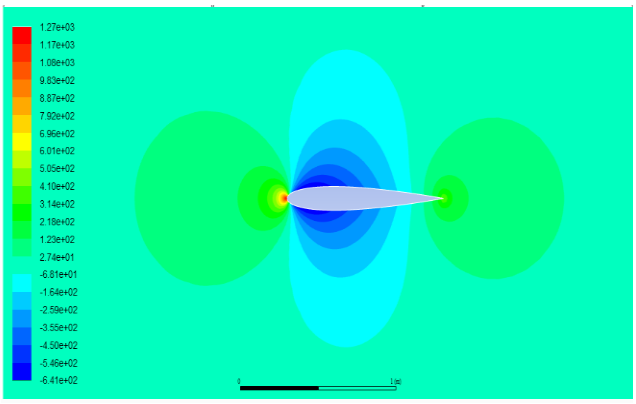

Figure 5 Static Pressure contour at 0 degree AOA for NACA0012

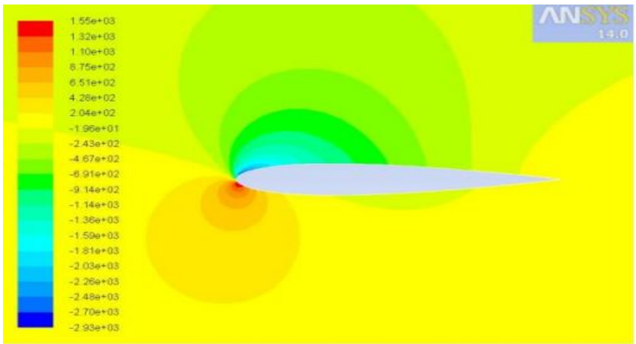

Figure 6 Static Pressure contour at 6 degree AOA for NACA0012 [8]

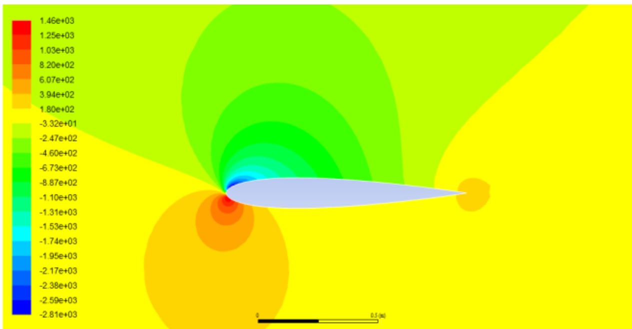

Figure 7 Static Pressure contour at 6 degree AOA for NACA0012

Following table shows the result comparison for reference paper result and results obtained by performing the simulation over NACA 0012

	Reference Paper Values	CFD Analysis Values
Cd at 0°	0.01373	0.01411
Cd at 6°	0.02566	0.02616
Cl at 0°	0.00015	0.00019
Cl at 6°	0.56947	0.57124

Table 1 Comparison of Cl, Cd values for reference paper reading and CFD Analysis.

C. Simulation

The CFD analysis of aircraft is done using Ansys Fluent. As the aircraft model is axisymmetric in nature, for simulation purpose half part of aircraft, along the axis of symmetry is taken and meshed. Figure 4 shows the meshing of the aircraft model.

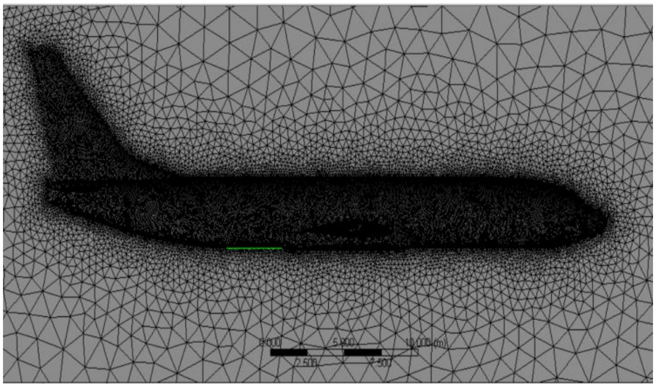

Figure 8 Meshing of typical aircraft model along symmetry.

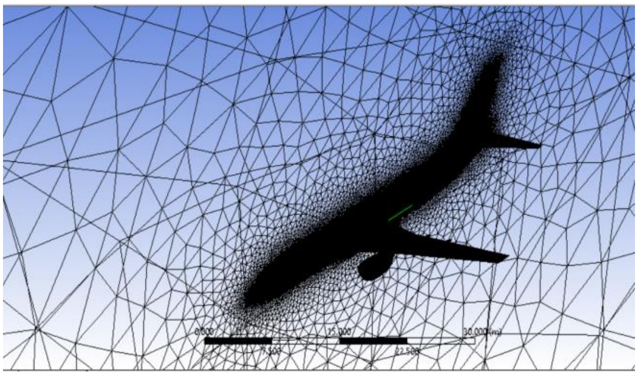

Figure 9 3-D view of half meshed typical aircraft model.

All the simulations are performed using sea level conditions. Boundary conditions along with model selection for 0° and 3° angle of attack are as follows;

Solver type: Pressure based
Model: K-ε, Standard Wall Function
Inlet type: Velocity Inlet (components)
Outlet: Pressure Outlet (atmospheric condition)
Aircraft: It is defined as wall, where the roughness height is given as 0.01m with roughness constant 0.5
Symmetry: boundary wall for analysis of flow behavior.

Reference values are given from inlet with Area = 160 m^2 and, Pressure (P) = 101325 Pa. and fluid characteristics of air as Density (ρ) = 1.225 kg/m^3 with Dynamic viscosity (μ) = 1.78e-5.

III. RESULTS

After performing the simulation on the aircraft model at 0 and 3 degree at 200 m/s, with and without adding the roughness following data and contour plots are obtained.

1. For 0° Angle of Attack

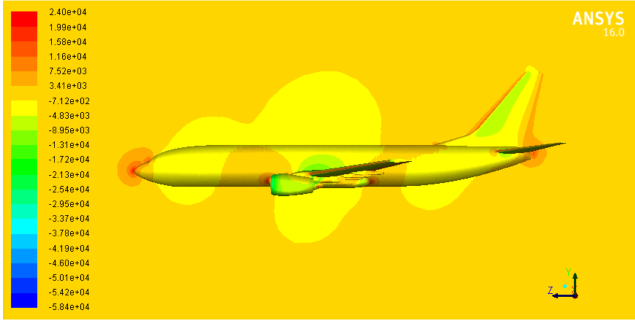

Figure 10 Static Pressure contour at 0 degree AOA

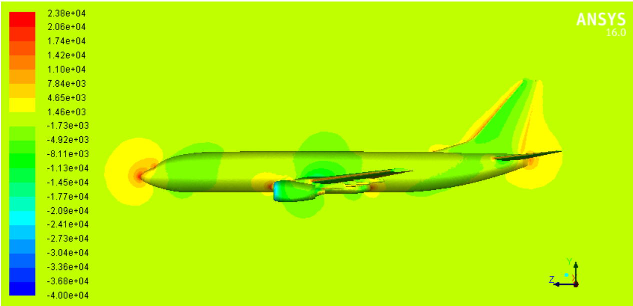

Figure 11 Static Pressure contour at 0 degree AOA with roughness

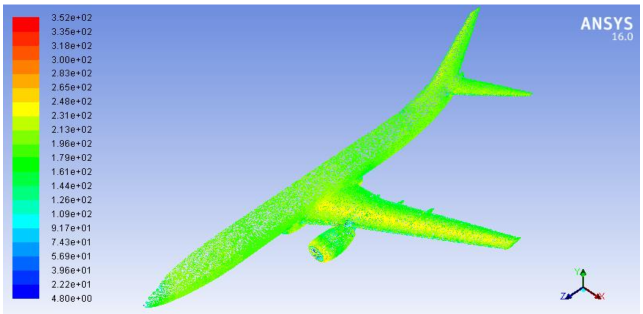

Figure 12 Velocity vectors colored with magnitude at 0 degree AOA

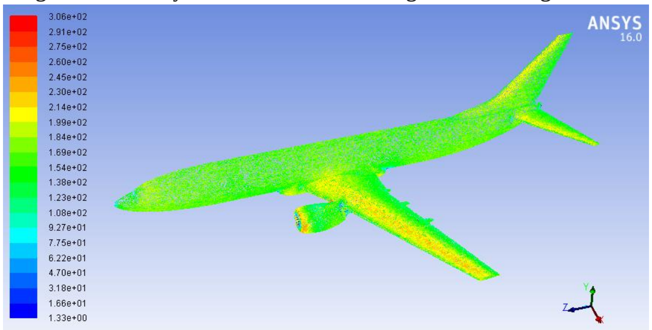

Figure 13 Velocity vectors colored with magnitude at 0 degree AOA with roughness

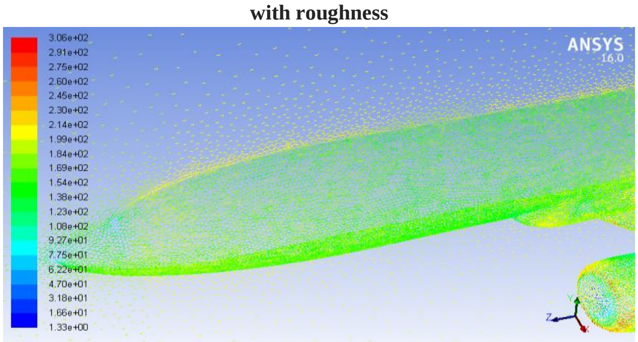

Figure 14 velocity vectors near boundary at 0 AOA with roughness

2. For 3° Angle of Attack

International Journal for Research in Engineering Application & Management (IJREAM)
Volume XX, Issue XX, MMM YYYY.

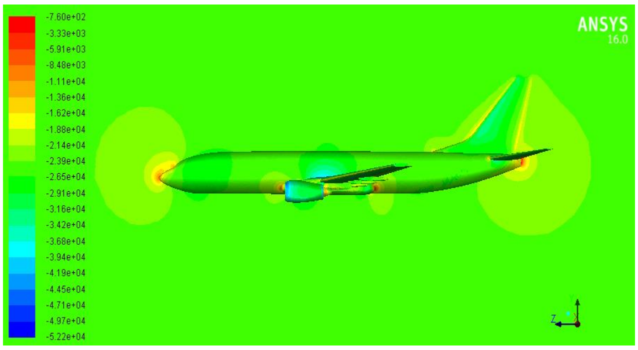

Figure 15 Static Pressure contour at 3 degree AOA

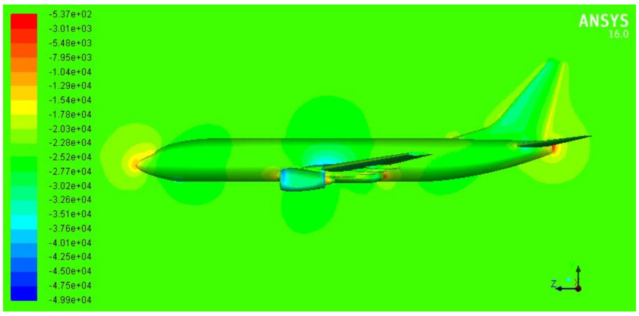

Figure 16 Static Pressure contour at 3 degree AOA with roughness

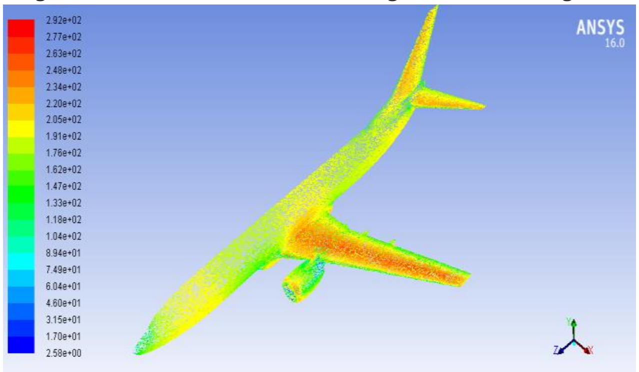

Figure 17 Velocity vectors colored with magnitude at 3 degree AOA

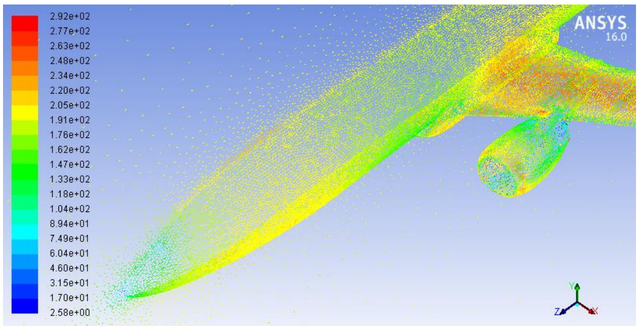

Figure 18 velocity vectors near boundary at 3 AOA

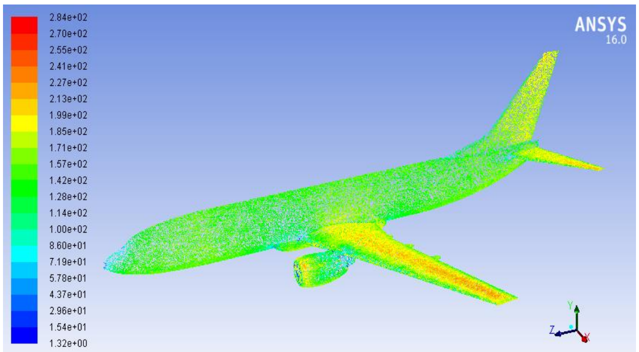

Figure 19 Velocity vectors colored with magnitude at 3 degree AOA

with roughness

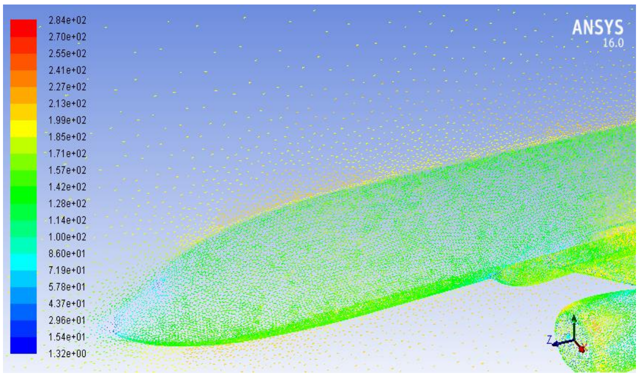

Figure 20 velocity vectors near boundary at 3 AOA with roughness

From the figure 14, 18 and 20 it can be observed that, flow velocity vector magnitudes shows boundary layer effects near the surface of fuselage

Coefficient of lift and coefficient of drag of the aircraft model for 0° and 3° angle of attack, with and without the roughness addition is listed in table 2 and table 3

For 0° AOA

	Without Roughness	With Roughness
Cl	0.07420	0.07095
Cd	0.01372	0.03475

Table 2 Cl &Cd values comparison of aircraft model for 0 degree AOA

For 3° AOA

	Without Roughness	With Roughness
Cl	0.082041	0.06029
Cd	0.01415	0.03168

Table 3 Cl &Cd values comparison of aircraft model for 3 degree AOA

IV. CONCLUSION

From the obtained values of Coefficient of Lift and Drag it can be observed that

1. In case on aircraft model without roughness the coefficient of lift and drag increases as the angle is increased from 0° to 3°.
2. As the roughness is added to aircraft surface at 0°, the coefficient of lift is found to be slightly decreasing, whereas Cd value is increased which may have

happened due to friction between flow and aircraft surface, resulting in additional drag addition.

3. At 3° similar results are obtained as for 0° as discussed above.
4. In case of 0° and 3° when the roughness is added, for 3° angle of attack the Cd value is found to be less by 0.00307 as compared to Cd value for 0° angle of attack.
5. Ultimately it can be concluded that at 0° angle of attack due to addition of roughness the total drag increases due to skin friction between fluid flow and aircraft surface. Hence to obtain favorable results the roughness should be added to a specified location over the surface where flow separates, so that flow will remain attached to surface.

REFERENCES

[1] J. Reneaux, “Overview: drag reduction technologies for civil transport aircraft”, *European Congress on Computational Methods in Applied Sciences and Engineering.* ECCOMAS, 2004.

[2] B. Rajasai, R. Tej, and S. Shrinath, “Aerodynamic effects of dimples on aircraft wings,” *IJAMAE*, vol. 2, Issue 2, October 2015.

[3] E. Livya, G. Anitha, P. Valli, “Aerodynamic Analysis of Dimple Effect on Aircraft Wing,” *International Journal of Aerospace and Mechanical Engineering,* vol. 9, no. 2, 2015.

[4] S. Hiremath, A. S. Malipatil, “CFD Simulation of Aircraft Body with Different Angle of Attack and Velocity,” *International Journal of Innovative Research in Sciences, Engineering and Technology*, vol. 3, Issue 10, October 2014

[5] S. Paul, A. Paul, F. A. Koly, “A Review of Different Shaped Dimple Effects on Aerofoil Surfaces,” *International Conference on Mechanical Energy and Renewable Energy*, Chittagong, Bangladesh, December 2019

[6] A. Kapoor, Jaykrishnan R., “Study & Design Golf Ball like Dimpled Aircraft 2-D Wing and Effects on Aerodynamic Efficiency,” *Proc. International Conference on Modern Research in Aerospace Engineering,* 2018

[7] Fabrizio De Gregorio, Daniel Steiling, Ernesto Benini, Rita Ponza, “ERICA Tiltotor airframe wake characterization,” *41st European Rotorcraft Forum*, 2015.

[8] Karna S. Patel, Saumil B. Patel, Utsav B. Patel, Ankit P. Ahuja,”CFD Analysis of an Airfoil”, *International Journal of Engineering Research* , volume 3, Issue 3, pp: 154-158, March 2014.