

REVIEW ARTICLE



ISSN: 2321-7758

FLOW ANALYSIS AROUND THE DIMPLE WING ON AIRCRAFT

MOHANASARAVANAN P S

Final Year PG Student, TRP Engineering College (SRM Group), Trichy

Article Received: 11/04/2015

Article Revised on:15/04/2015

Article Accepted on:18/04/2015



MOHANASARAVANAN P S

ABSTRACT

The present work describes change in aerodynamic characteristics of an airfoil by applying certain surface modifications in form of dimples. Our aim of the project is to design a wing with dimples on the upper surface of the wing and analyse the results using design software ICEM CFD and CFX in ansys. We found they results such as increasing the angle of stall, increase the flow separation from laminar to turbulent, pressure drag of the wing will decrease but skin friction point drag will increase due to attached flow over the surface of the wing. A comparative study showing variance in lift and drag of modified airfoil models at different angle of attack with and without dimple shape of wing is done. In olden days people were thinking that only smooth surface is efficient one but not a ruff surface Now, research has made it as false and ruff surface can also increase the efficiency.

Keywords: Golf ball, Dimples, Wing etc.

©KY Publications

I. INTRODUCTION

Wing design is constantly evolving. If you compare the wing of Wright flyer with that of a modern aircraft such as Boeing, airbus etc. the difference is remarkable. Since 1930's, NASA and its predecessor NACA have been on the forefront of wing design, developing the basic airfoil shapes for airplane manufactures. Freedom in the air is still not complete. Continuous attempts are being made to increase freedom in air, be it speed, size or maneuverability. From commercial jetliners to supersonic fighters, there has been an exponential growth in the aviation industry. Still there is vast

scope for further improvements. Here is a study that makes one such attempt.

II. FLOW SEPARATION OF GOLF BALL

Reynolds number (Re) is an important non-dimensional parameter that is used to relate the size of an object to the flow conditions it experiences, and is defined by the equation

$$Re = \rho V_{\infty} l / \mu$$

Where

ρ = Atmospheric density

V_{∞} = Velocity

l = Reference length (in the case of a sphere, this

variable is defined as the diameter)

μ = Viscosity (or friction)

From the survey of literature, any two spheres that experience the same Reynolds number should exhibit the same aerodynamic characteristics even if the spheres are of different sizes or flying at different speeds. The above figure indicates that there is a significant change in the drag on a smooth sphere at a Reynolds number of about 3×10^5 . Below this Re, the drag coefficient is roughly constant at 0.5. Above this Re, the drag coefficient again becomes nearly constant at about 0.1.

We have already discussed the concept of flow separation. At low Reynolds numbers, the boundary layer remains very smooth and is called laminar. Laminar boundary layers are normally very desirable because they reduce drag on most shapes. This separated flow is what causes the drag to remain so high below the critical Reynolds number.

At that Reynolds number, however, the boundary layer switches from being laminar to turbulent. The location at which this change in the boundary layer occurs is called the transition point. A turbulent boundary layer causes mixing of the air near the surface that normally results in higher drag

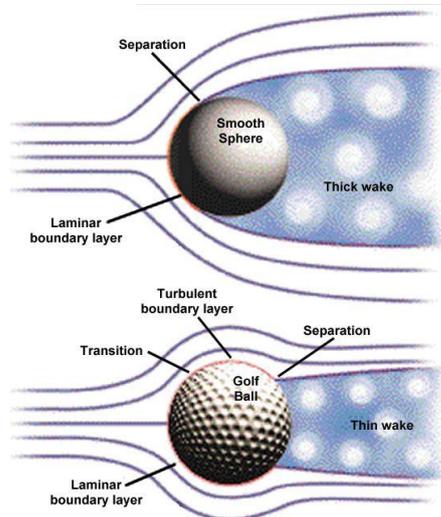


Figure 1: Flow separation of between smooth ball and golf ball

The difference in the flow fields around a smooth sphere and a rough, or dimpled, sphere can be seen above. Since the laminar boundary layer around the smooth sphere separates so rapidly, it creates a very large wake over the entire rear face. This large wake maximizes the region of low pressure and, therefore, results in the maximum difference in pressure between the front and rear faces. As we

have seen, this difference creates a large drag like that seen below the transition Reynolds number.

The transition to a turbulent boundary layer, on the other hand, adds energy to the flow allowing it to remain attached to the surface of the sphere further aft. Since separation is delayed, the resulting wake is much narrower. This thin wake reduces the low-pressure region on the rear face and reduces the difference in pressure between the front and back of the sphere. This smaller difference in pressure creates a smaller drag force comparable to that seen above the transition Reynolds number.

The purpose of the dimples is to do just that--to create a rough surface that promotes an early transition to a turbulent boundary layer. This turbulence helps the flow remain attached to the surface of the ball and reduces the size of the separated wake so as to reduce the drag it generates in flight. When the drag is reduced, the ball flies farther. Some golf ball manufacturers have even started including dimples with sharp corners rather than circular dimples since research indicates that these polygonal shapes reduce drag even more.

III. MODELLING

The wing model is generated in CATIA and imported in ICEM CFD using unstructured hexahedral elements. Critical to the success of every three dimensional CFD analysis is the mesh that defines the computational domain.

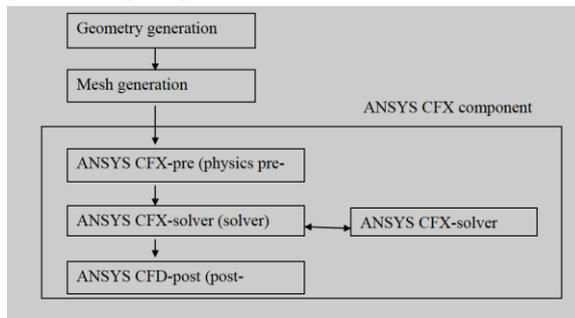
Primary objective for the designers is to balance mesh size with computational time to achieve a mesh independent converged solution in a reasonable amount of time. Here the symmetrical airfoil is used to find if any lift is created by dimples or not in zero angle of attack. But cambered airfoil difficult to analyze the result at zero angle of attack. The specifications of the wing are illustrated in the table below

Wing series	NACA 0014
Wing span	6.650 m
Chord length	1m
Maximum camber	14%
Type of dimple	Sphere
Number of dimple	147
Diameter of dimple	0.25m
Depth of dimple	0.072m

Table 1: Specification of wing

III. ANALYSIS SETUP

ANSYS CFX contains three division such as pre-processor, post processor, and solver.



A. ICEM CFD

First we need to import the wing from CATIA V5 model, then after that following procedure is to be done

File > Import Geometry>CATIA>Choose the File >Import>Apply

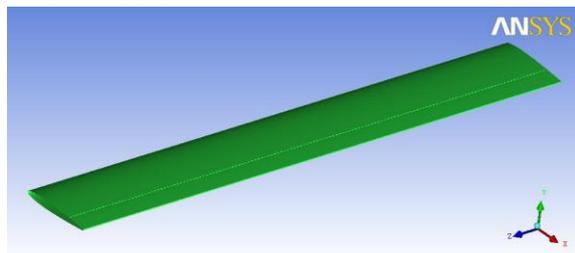


Figure 2: Imported wing from CATIA

Here create a control volume using appropriate points, curves and surfaces

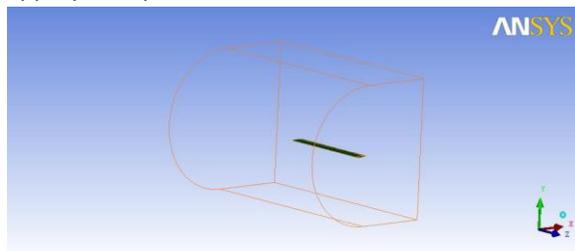


Figure 3: Mesh generation of dimple wing

Parts are to be created such as wing, inlet, outlet, walls, and symmetry. The following steps are to be followed to create parts.

Right Click on Parts > Create Parts > Name the Parts > Select the Respective Surfaces > Apply

Bodies are to be created to specify the fluid which is used. To create body the following procedure is to be done.

Geometry > create body > name the body > centroid of two points > select two points in the screen > apply > dismiss

To create mesh on the geometry follows the procedure as below

Mesh > global mesh setup > set the size of the meshing element > apply > dismiss

Part mesh setup>set mesh size to get fine mesh>apply>dismiss

Compute mesh > mesh type-tetra/mixed > mesh method-robust > compute > dismiss

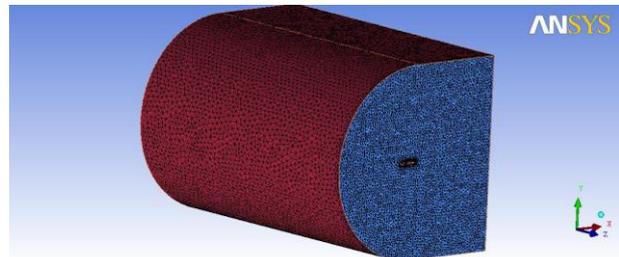


Figure 4: Mesh generation of dimple wing

B. CFX- Pre

In CFX-pre we need to set the boundary conditions for inlet, outlet, and symmetry and set the working directory. For that follow the procedure below:

- ✓ File > New Case > General > ok
- ✓ Mesh > Right Click > Import mesh > ICEM CFD > Select the mesh file (.cfx5) > Open
- ✓ Insert > Domain > Name (Air) – ok
 - Basic Setting
 - Location > Air (Body)
 - Fluid 1 > Material Library
 - Material > Air Ideal Gas
 - Pressure Reference > 1 atm
 - Fluid Models
 - Heat Transfer > Total Energy
 - Turbulence > k-epsilon
 - ✓ Insert > Boundary > Name (Inlet) – Ok
 - Basic Settings
 - Boundary Type > Inlet
 - Location > Inlet
 - Boundary Details
 - Flow Regime > Subsonic
 - Mass and Momentum > Cart. Vel. Components (u-180m/S v-0 w-0)
 - Heat Transfer > Static Temperature > 288.15 K
 - ✓ Insert > Boundary > Name (Outlet)
 - Basic Settings
 - Boundary Type > Outlet
 - Location > Outlet
 - Boundary Details
 - Flow Regime > Subsonic
 - Mass and Momentum >Average Static Pressure > 0 Pa
 - Pres. Profile Blend >0.05

- ✓ Insert > Boundary > Name (Symmetry)
- Basic Settings
- Boundary Type > Symmetry
- Location > Symmetry
- ✓ Insert > Boundary > Name (Wing) – Ok
- Basic Settings
- Boundary Type > Wall
- Location > Wing
- Boundary Details
- Mass and Momentum > No Slip Wall
- Wall Roughness > Smooth Wall
- Heat Transfer > Adiabatic
- ✓ Insert > Solver > Solver Control
- Basic Settings
- Maximum Iteration > 1000
- Residual Target > 1.E-4

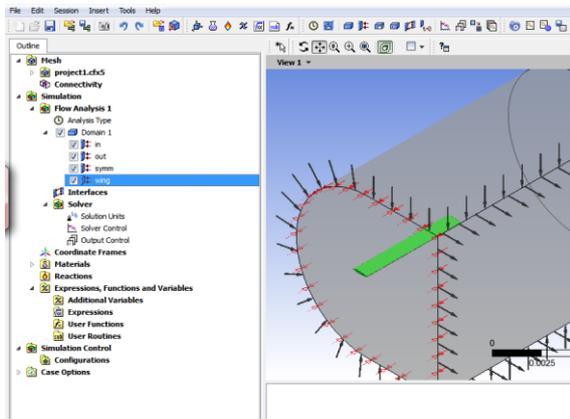


Figure 5: Creation of boundary condition in CFX
 Tools > Solver > Start Solver > Define Run > Save (name.def) Solver run finished normally dialog box appears click on post process result and then click ok .

C. CFX Post

To visualize the flow over the model

- Insert > Location > Plane > ok
- Select domains > Select the Plane
- > Select the location of plane
- Insert > Contour > ok
- Domain: All Domains
- Location: Plane
- Variable: Pressure
- Color Scale: Liner
- Color Map: rainbow
- Tools > Function Calculator
- Function: Force
- Location: Wing
- Direction: X (for drag)
- Y (for lift)

IV. RESULT

Form the postprocessor I found the result for both with and without dimple wing and finally that result to be compared for different angle of attack.

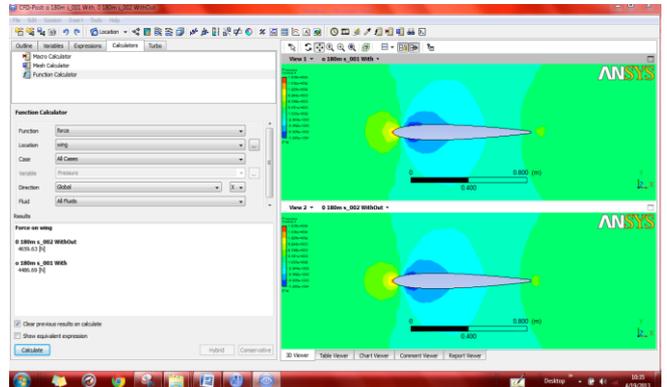


Figure 6: At angle of attack 0 deg

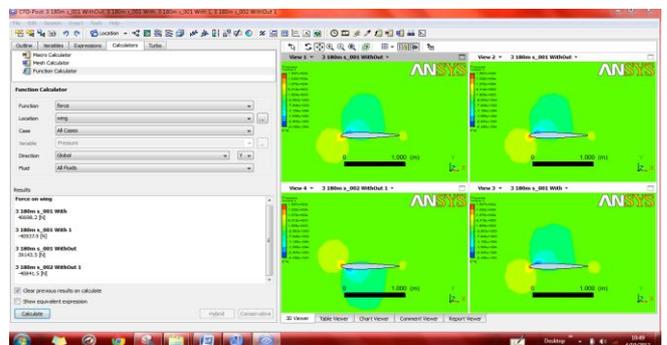


Figure 7: At angle of attack 3 deg and -3 deg

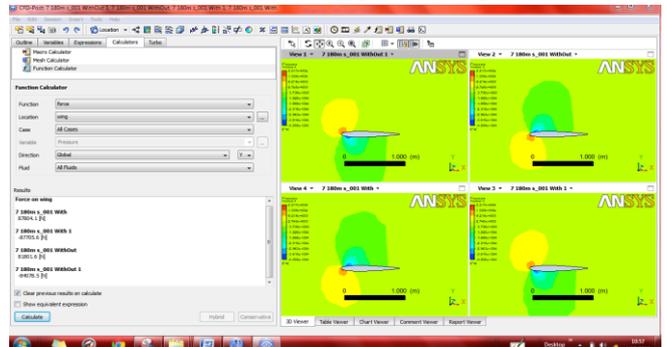


Figure 8: At angle of attack 7 deg and -7 deg

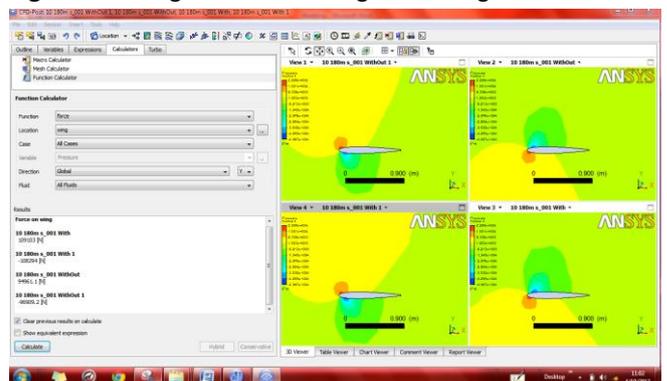


Figure 9: At angle of attack 10 deg and -10 deg

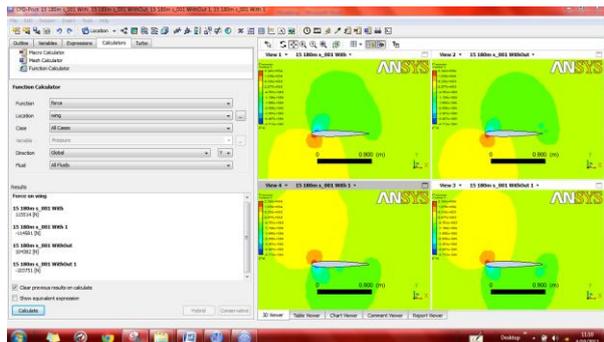


Figure 10: At angle of attack 15 deg and -15 deg

V. CONCLUSION

By conclude the work based on result obtained from the analysis part. Create a chart from the value obtained which is listed below.

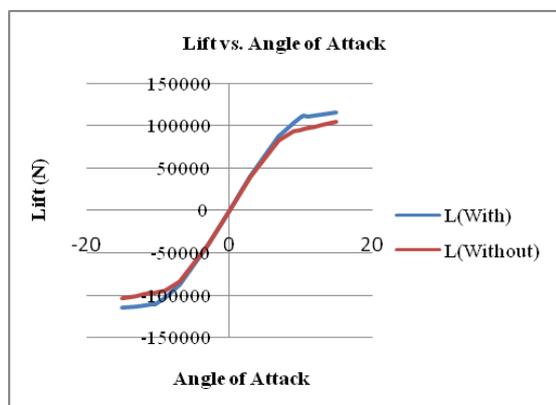


Figure 11: Lift vs. Angle of attack

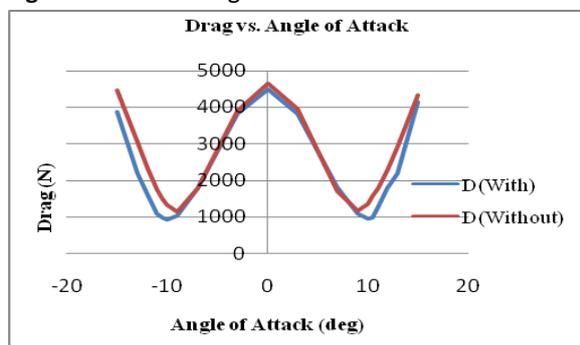


Figure 12: Drag vs. Angle of attack

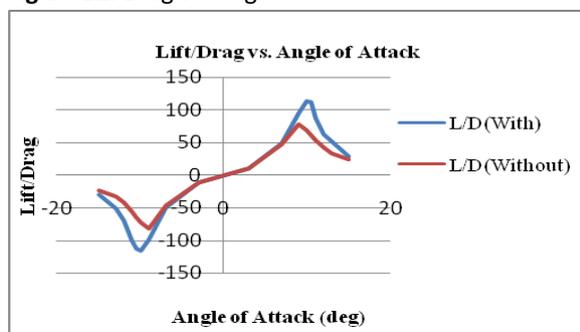


Figure 13: Lift / Drag vs. Angle of attack

From the analysis of the software designed model with dimple wing and normal wing. It is clear that critical angle of stall increases in dimple wing. There

is much variation in lift and drag. In future, the prototype of this software model should be designed and tested in the practical environment to get the actual result.

ACKNOWLEDGEMENT

The author is very much thankful to the management of jet wings technology Bangalore for granting permission to do analysis in the institute and also thankful to my college.

REFERENCES

1. Anthony C.Occhipinti (1998), "More speed with less noise by" January 17 in Aviation Sports.
2. Brett Burglund and Ryan Street, (May 2011) "Golf Ball Flight Dynamics".
3. Bruce D. Kothmann, (January 2007) "Aerodynamics of Sports Balls".
4. Chowdhury.H, (2010) "A comparative study of golf ball aerodynamics".
5. Deepanshu Srivastav (2012), "Flow control over airfoil using different shaped dimples" IPCSIT vol.33.
6. Jhon Louis Vento, (2011) "Analysis of Surface Augmentation of Airfoil Sections via Flow Visualization Techniques".
7. Jon Fichtelman, "The Fluid Mechanics of Sports Balls".
8. Katsumi Aoki, (August 2009) "Aerodynamic Characteristic and Flow Pattern on Dimples Structure of a Sphere".
9. Michael B. Patacsil, (2010) "Dimples on Wings", California Science Fair 2010.
10. Moscow Russia (2009), "Aerodynamic characteristic and flow pattern on dimples structure of a sphere".
11. Riley J.Norman (2010), "Don't be a drag: The effect of dimples on an airplane wing" california state science fair 2010 project number J0121.
12. <http://www.aerospaceweb.org/question/aerodynamics/q0215.shtml>
13. <http://www.articlesbase.com/science-articles/golf-ball-dimples-drag-3649702.html?en>
14. <http://qm-aerospace.blogspot.com/>