ISSN (Print) : 0974-6846
Indian Journal of Science and Technology, Vol 9(45), DOI :10.17485/ijst/2016/v9i45/104585, December 2016 ISSN (Online) : 0974-5645

CFD Study of Solid Wind Tunnel Wall Effects on Wing Characteristics

Hanumanthu[1*] and K. V. Sreenivas Rao[2]

[1]Thermal Power Engineering, SIT, Tumkur – 572103, Karnataka, India; hanumanthbalgeri@gmail.com
[2]Department of Mechanical Engineering, SIT, Tumkur – 572103, Karnataka, India

Abstract

Most of airplane structures include the components such as Fuselage, Wings, Empennage, Landing gear and Power plant etc. Wings are the important part of aircraft and these are airfoils attached to each side of the fuselage and are the main lifting surfaces that support the airplane in flight. SSD-2 airfoil, a modified version of NACA23015 airfoil is used to study the aerodynamic characteristics such as lift, drag, moment and wall effects with different wind tunnel blockage ratios. Generally, any new airfoil characteristics are derived from the extensive wind tunnel tests, however experiments are costly. Hence, to get the idea of aerodynamic characteristics and wall effects over airfoil with different wind tunnel blockage ratios, CFD tools are used such as ICEM CFD and ANSYS FLUENT.

This paper presents the results of simulation of flow past SSD-2 airfoil. The study is carried out on a 2D airfoil section for free stream condition. Further the computations are made on 3D rectangular wing of solid wind tunnel simulation by varying wind tunnel height to get the different wind tunnel blockage ratios, keeping the other parameters constant. In solid wind tunnel simulation the lift and moment are less compared to free stream simulation results due to blockage effects. Finally, it is observed that increase in the wind tunnel blockage ratio, lift and moment decreases, drag increases with less difference in pressure; whereas decrease in wind tunnel blockage ratio, the lift and moment increases, drag decreases with more difference in pressure.

Keywords: Drag, Fluent, ICEM CFD, Lift

1. Introduction

It is a common experience that a body in motion through a fluid experiences a resultant force which, in most cases is mainly a resistance to the motion. A class of body exists, however for which the component of the resultant force normal to the direction to the motion is many times greater than the component resisting the motion and possibility of the flight of an airplane depends on the use of the body of this class for wing structure. Airfoil is such an aerodynamic shape that when it moves through air, the air is split and passes above and below surface of the wing. The wing's upper surface is shaped so the air rushing over the top surface speeds up and stretches out. This decreases the air pressure above the wing. The air flowing below the wing moves in a comparatively straighter line, so its speed and pressure remains the same. Since high air pressure always moves toward low air pressure, the air below the wing pushes upward toward the air above the wing. The wing is in the middle and the whole wing is lifted. The faster an airplane moves, more lift there is. From the Bernoulli's principle if the air speed is more on the upper surface of wing, the pressure decrease on it and ultimately it is more over the lower surface of wing since the air speed on

*Author for correspondence

it is low. This pressure difference causes the lifting of aero plane. The lift force is more when the pressure difference is large. For flying of aero plane the lift force should also more than the center of gravity.

2. Modeling and Grid Generation

The commercial available software such as ANSYS ICEM CFD is used for developing three different cases of models. The co-ordinates of SSD-2 aerofoil which was developed by Dr. S. S. Desai into ANSYS ICEM CFM are imported. Then using available option in the software it is transferred to 2D aerofoil, then extruded it into 3D model, which is a wing of 4 m wing span and 1 m chord length. After this, the domain for 2D model was developed as shown in Figure 7, for 3D H-grid as shown in Figures 1 and 2 and for 3D H-grid of reduced scale as shown in Figures 4 and 5. After modeling mesh is developed using suitable calculation such as number of nodes around wing as 200 nodes, along upstream-40 nodes, downstream-70 nodes and upside and downside as 30 nodes and also maintaining the spacing between one node to another as 0.0001 m and aspect ratio of cell is 1.005 m.

Case 1: Wind tunnel simulation model with blockage ratio of 16.67%.

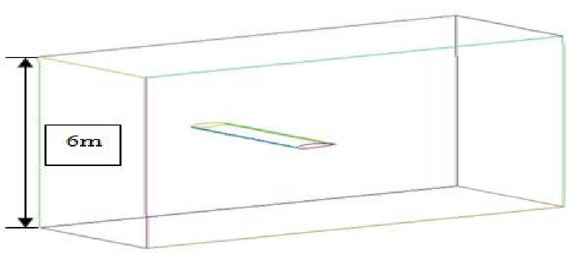

Figure 1. Case-1 3D model.

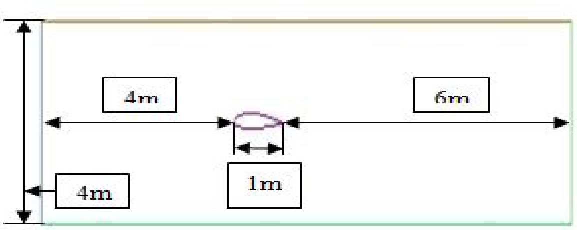

Figure 2. Top view of Case-1 3D model.

In Case 1, the model is developed with the required dimensions as shown in Figures 1 and 2. In this model the total number of cells maintained is 720000 and aspect ratio of cell is 1.005 near to wing and away from the wing, the aspect ratio of cell is 1.2. There are different types of grids such as O, H and C types of grids. In this work the selected grid is structured H-grid mesh. For this model the analysis is carried out by treating the model as wind tunnel simulation model.

$$Wind\ tunnel\ blockage\ ratio = \frac{Projected\ area\ of\ test\ body}{Cross\ sectional\ area\ of\ wind\ tunnel}$$

$$Wind\ tunnel\ blockage\ ratio = \frac{4*1}{4*6}*100$$

$$Wind\ tunnel\ blockage\ ratio = 16.67\%$$

Case 2: Wind tunnel simulation model with blockage ratio of 50%.

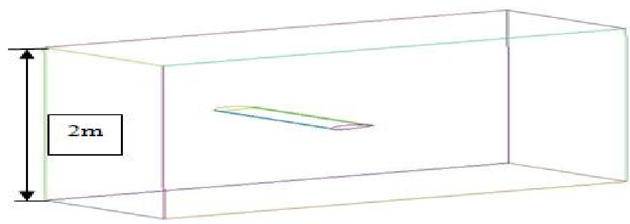

Figure 3. Case-2 3D model.

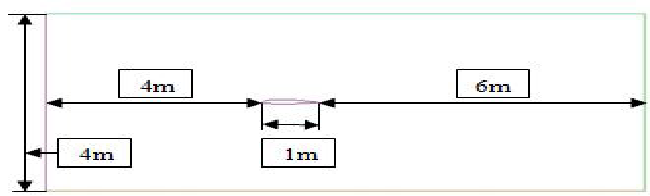

Figure 4. Top view of Case 2 3D model.

In Case 2, the model is developed with the required dimensions as shown in Figures 3 and 4. In this model the total number of cells maintained is 640000 and aspect ratio of cell is 1.005 near to wing and away from the wing, the aspect ratio of cell is 1.2. In this work the selected grid is also structured H-grid mesh for Case 2 model. For this model the analysis is carried out by treating this model also as wind tunnel simulation model.

$$Wind\ tunnel\ blockage\ ratio = \frac{Projected\ area\ of\ test\ body}{Cross\ sectional\ area\ of\ wind\ tunnel}$$

$$Wind\ tunnel\ blockage\ ratio = \frac{4*1}{4*2}*100$$

$$Wind\ tunnel\ blockage\ ratio = 50\%$$

Case 3: Free stream simulation model.

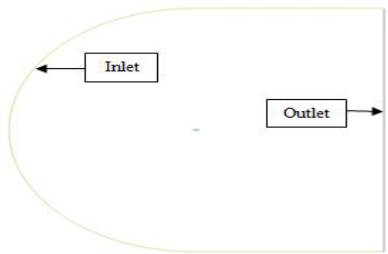

Figure 5. 2D model.

In Case 3, 2D model is developed as per dimensions as shown in Figure 5. This model is treated as free stream model by giving boundary conditions inlet and outlet as pressure far field. Here a C-H type of structured grid is selected.

Table 1. Geometry and boundary layer details for Case 1 and Case 2

Reference Area	4 m^2
Wing Span	4 m
Chord Length	1 m
Moment reference point (X,Y,Z)	(0, 0, 0)
Y+	1
Y	0.00000519 m

For Wind Tunnel Simulation Models:
At The Wall (Top, Bottom and Sides):
Atmospheric Pressure = 101325 pa
Temperature = 300 k
At the Velocity Inlet:
Velocity = 69.437m/s or Mach number = 0.2
Gauge pressure = 0pa
Atmospheric Pressure = 10135pa
Temperature = 300 k
At the Pressure Outlet:
Pressure = 101325 pa
Temperature = 300 k
For Free Stream Model:
At Pressure Far Field:
Mach No = 0.2 or Velocity = 69.437 m/s
Temperature = 300 k
Atmospheric pressure = 101325 pa

Results and Discussion on Aerodynamic Characteristics

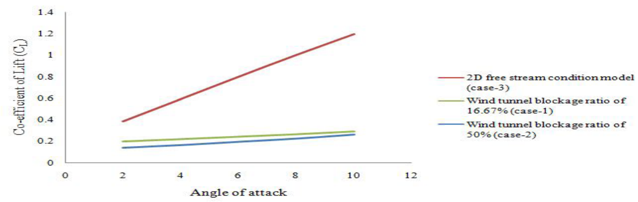

Figure 6. Coefficient of lift vs. angle of attack.

The above Figure 6 shows coefficient of lift (C_L) vs. angle of attack. As the angle of attack increases coefficient of lift that is lift force also increases since lift force is directly proportional to lift coefficient. This is happening for all three cases because by virtue of its shape alone, an airfoil will generate lift as air flows over it. However, even more lift can be produced by the airfoil if it is tilted with respect to the airflow. This tilt is called an airfoil's angle of attack. When the wing is tilted, the air is flowing over the top surface of wing flows even faster than the air flowing underneath. As the difference in the speed of the two airflows increases, the difference in pressure also increases. So, as its angle of attack increases, the wing generates more lift. Here important observation is that for free stream simulation of Case 3, the lift co-efficient is more ultimately lift force is more since lift is directly proportional to lift coefficient. While in wind tunnel blockage ratio of 16.67% of Case 1 model, the lift force is more compared to wind tunnel blockage ratio of 50% of Case 2 model due to blockage effects and also due to compressibility of flow field that is while the fluid flowing over the wing and hitting the top bottom wall of tunnel, the flow reflected over to the wing. That tends to decrease the lift.

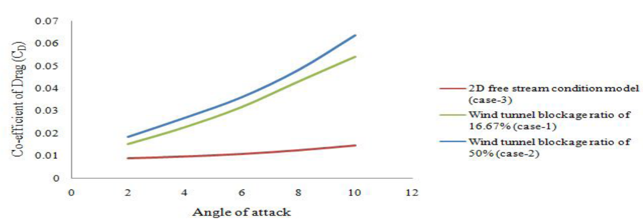

Figure 7. Coefficient of drag vs. angle of attack.

In the above Figure 7, coefficient of drag (C_D) vs. angle of attack is drawn. Even as the angles of attack increase, the drag coefficient also increases because if the angle of attack increases, the velocity of fluid will increase due to shape of airfoil and density of oncoming fluid also more which causes the increment of drag force with increase in angle of attack. But the drag is more in the wind tunnel blockage ratio 50% of Case 2 model compared to the free stream simulation model of Case 3 and wind tunnel blockage ratio of 16.67% of Case 1 model since while the fluid flowing in the tunnel and hitting the top and bottom wall of tunnel and getting reflected towards the wing tends to decrease the lift and increases the drag which means more compressibility of flow fluid in blockage ratio of 50% of Case 2 model due to its less width to height ratio in comparison of Case 1 model width to height ratio.

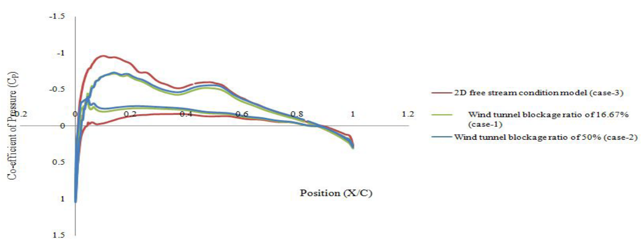

Figure 8. Coefficient of pressure vs. position (X/C) 2 degree angle of attack.

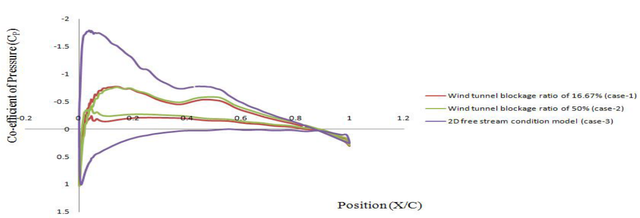

Figure 9. Coefficient of pressure vs. position (X/C) 6 degree angle of attack.

The above Figures 8 and 9 indicate coefficient of pressure (C_p) vs. position (X/C) along the chord length of aerofoil at 2 and 6 degrees angle of attack respectively. Here important observation is that co-efficient of pressure is directly proportional to angles of attack which mean that the distribution of pressure over the aerofoil is more for higher angles of attack due to increase in lift force. In case of wind tunnel simulation since the compressibility of flow field is more in blockage ratio of 50% of Case 2 model since its more wind tunnel blockage ratio which means that there is more velocity and also less lift force compared to wind tunnel blockage ratio 16.67% of Case 1 model. Hence in Case 2 model more the compressibility of flow field since its more wind tunnel blockage ratio and less lift force, the pressure co-efficient is less compared to wind tunnel simulation of Case 1 model. The pressure co-efficient is more in free stream simulation of Case 3 model because there is no such restrictions. Further to confirm this phenomenon, the analysis of all three cases for 2 degree angle of attack shown in Figure 8 and for 6 degree angle of attack shown in Figure 9 was carried out. If the angle of attack is increased, the coefficient of pressure also increases due to increase in lift force as shown in the above Figures.

3. Conclusion

Three cases of models such as two wind tunnel models and one free stream model have been developed in ANSYS ICEM CFD software with suitable dimensions. For same models, the simulation is carried out in ANSYS FLUENT software. The convergence criteria for continuity, x and y velocities and turbulence was set to $1*10^{-5}$. This analysis was carried out on a unit chord airfoil with a Reynolds number of 5 million. The following conclusions can be drawn from the study:

- In case of wind tunnel simulation results the coefficient of lift is less by 15% when compared to 2D free stream simulation results.
- 2. With wind tunnel blockage ratio of 16.67% of Case 1 have highest coefficient of lift compared to Case 2 of wind tunnel blockage ratio of 50%.
- With wind tunnel blockage ratio of 50% of Case 2 have highest coefficient of drag compared to Case 1 of wind tunnel blockage ratio of 16.67%
- Pressure distribution is more in case of 2D free stream model compared to wind tunnel simulation models.
- It is seen that angle of attack is directly proportional to pressure coefficient.

4. Acknowledgment

I would like to express my sincere gratitude to Dr. K. V. Sreenivas Rao for his guidance and assistance in this study work. I also sincerely thank Dr. V. Ramesh for his support and guidance to carry out this study work successfully.

5. References

1. Maskell EC. A theory of the blockage effects on bluff bodies and stalled wings in a closed wind tunnel. Ministry of Aviation, Aeronautical Research Council Reports 3400; 1963.
2. Perzon S. On blockage effects in wind tunnels - A CFD study. SAE Technical Paper 2001-01-0705; 2001. DOI:10.4271/2001-01-0705.
3. Yen JC, Martindale WR, Duell EG, Amette SA. Determining blockage ctions in climatic Wind tunnels using CFD. SAE, 2003-01-0936; 2003. DOI:10.4271/2003-01-0936.
4. Srineevas G, Abhiram P. Flow analysis of wing under critical mach number. International Journal of Current Engineering and Technology. 2014 Feb; S2:432–5.
5. Sahini D. Wind Tunnel Blockage Corrections. A thesis paper. 2004.