Case Study

Flow over an Airfoil with Ansys Discovery

Developed and curated by the Ansys Academic Development Team

Madhumita Saravana Kumar

education@ansys.com

Summary

Ansys Discovery is a simulation-driven design tool that combines instant physics simulation, high fidelity simulation, and interactive geometry modeling in a single easy-to-use experience. This enables designers and engineers to illustrate a wide variety of concepts in the fields of design, structures, fluids, and heat transfer with the help of simulation.

This case study deals with simulating the flow of air over an airfoil using Ansys Discovery. The aim of it is to understand how lift and drag forces act on the airfoil and to visualize them using the software. The influence of the angle of attack of an airfoil on the lift and drag forces are highlighted.

Table of Contents

1. Introduction

Aerodynamics is a field of engineering that involves the study of external air flow over objects. This is an important field of study as it helps improve the design of different types of objects, ranging from airplanes to bicycle helmets, to increase performance in terms of the fluid forces acting on them. In this introductory case study, we will focus on the influence of shapes on aerodynamics of airfoils used in airplane and race car wings.

2. Forces acting on a plane

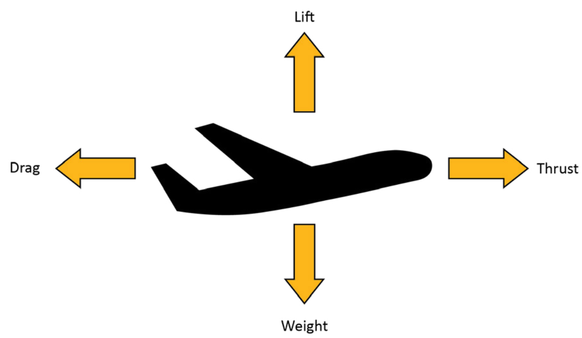

Figure 1. Forces acting on an Airplane

Before we learn about airfoils, it is important to understand what types of forces act on an airplane during flight. When an airplane flies in the air, it is exerted upon by four types of forces as indicated in Figure 1, *viz.*:

- Weight: It is the gravitational force exerted on the plane.
- Lift: The force exerted by the air to lift the plane up in the atmosphere.
- Thrust: The force exerted by the engine by burning aviation fuel to move the plane forward in the intended direction.
- Drag: The overall resistive force exerted by air against the direction of flight.

To maximize the performance of a plane while flying, the airplane, especially the wings, need to have a favorable aerodynamic design, *i.e.*, maximized lift and minimized drag forces.

3. Airfoils

An airfoil is the cross-section of a body that is capable of lift when a fluid flows over it. They are used in airplane wings, racing cars, turbines and many other applications.

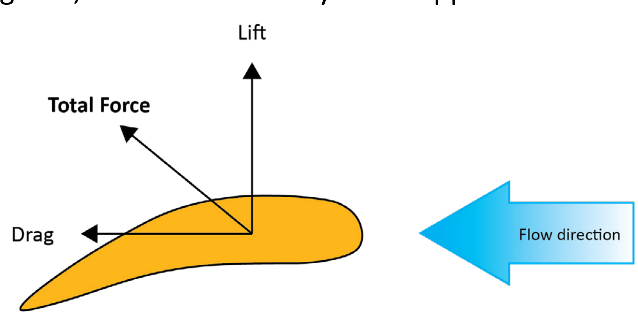

Figure 2. Lift and Drag Forces acting on an airfoil

Ansys / DISCOVERY

Let us consider a simple case of flow over an airfoil as shown in Figure 2, where the fluid exerts a force on the airfoil. The component of the force in the direction of fluid flow is known as drag and the component perpendicular to the direction of flow is known as lift. Airfoils are generally designed in such a way to maximize lift and minimize drag.

3.1 Why does lift and drag occur?

There are several ways of explaining the causes of lift and drag. For the sake of simplicity, this case study will focus on the Bernoulli principle to understand why lift and drag occur in an airfoil. The Bernoulli theorem states that for an incompressible steady state flow with negligible viscous effects, the following sum remains constant [1]:

$$P + \frac{1}{2}\rho v^2 + \rho g h = constant \tag{1}$$

where P is absolute pressure, ρ is density, g is the acceleration due to gravity, v equals velocity, and h is the height above a reference point.

Consider the air flow over an airfoil as shown in Figure 3. The air under the airfoil decelerates due to the resistance in the flow path because of the shape of the airfoil. To compensate for the conservation of mass, the air above the airfoil accelerates[1]. Using Bernoulli's principle, we know that velocity and pressure are related to each other. When the velocity of air increases, the pressure decreases and vice versa [2].

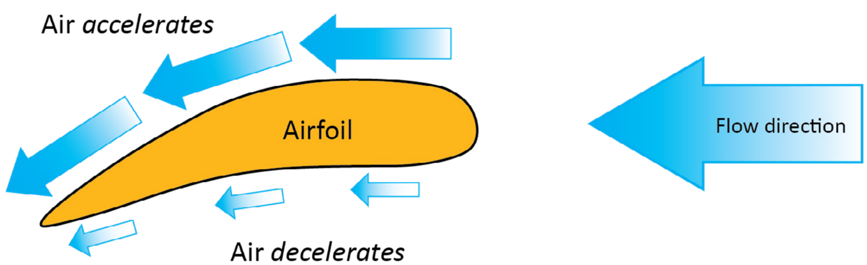

Figure 3. Air flow over the Airfoil

Therefore, we can conclude that the pressure of the air at the bottom of the airfoil is higher than that of the top as show in Figure 4. This difference in pressure exerts a force on the airfoil in the upward direction also known as lift. Drag is caused because of resistance against the flow created by the airfoil. The difference of pressure around the airfoil also contributes to the resistive drag force.

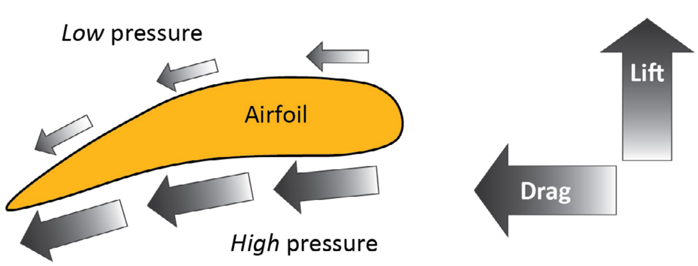

Figure 4. Causes of Lift and Drag

1 Another explanation for the change in velocity is based on the Coanda effect which is the tendency of a fluid to curve towards a surface. This causes flow turning and a difference in the velocity of the fluid on either side of the airfoil.

3.2 Important parameters of an airfoil

Some important parameters to remember in an airfoil can be found in Figure 5

- **Leading edge** is the forward most edge of an airfoil.
- **Trailing edge** is the outward edge at the back of an airfoil.
- **Chord line** is the perpendicular line connecting the leading edge and trailing edge.
- **Camber line** is the line midway between the upper and lower surfaces.
- **Angle of attack** is the angle between the chord line and the direction of flow.

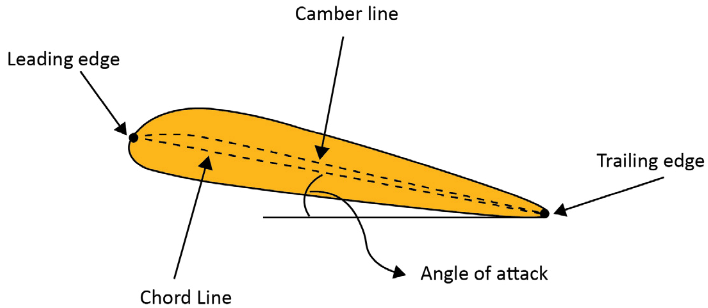

Figure 5. Parameters of an Airfoil

The camber line and angle of attack play a big role in influencing the lift and drag forces acting on an airfoil.

3.3 Shapes of Airfoils

Airfoils come in a variety of shapes and sizes based on its application, as shown in Figures 6 (a)-(c). For example, the shape of a wind turbine blade wing is different to that of wings in a commercial airplane which is different compared to a race car wing. Race cars wings use an inverted airfoil design when compared to airplane wings in order to increase downward forces i.e., traction to prevent toppling while making sharp turns [3].

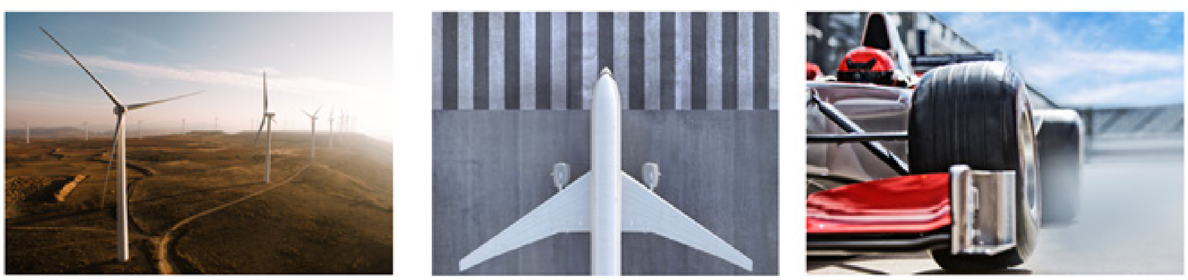

Figure 6. (a) Wind Turbine Blade, (b) Airplane Wing, (c) Race Car Wing

3.4 Influence of Angle of Attack

Angle of attack plays a role to a certain extent on the amount of lift and drag that an airfoil undergoes. The influence of angle of attack is dependent on the shape of the airfoil (asymmetry, camber line, etc.). A generic representation of the influence of angle of attack on lift and drag can be seen in the Figures 7 (a)-(c). Drag increases with angle of attack as the resistance to flow increases whereas lift increases up to a certain extent after which flow separation takes place causing stalling (Figs 8 and 9). Figure 9 indicates the flow separation that occurs when the angle of attack is too high causing stalling.

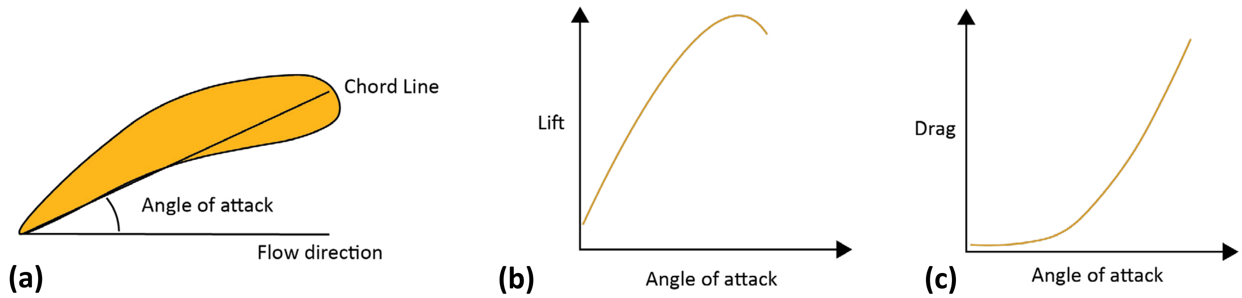

Figure 7. (a) Angle of Attack (b) Influence of Angle of Attack on Lift (c) Influence of Angle of Attack on Drag

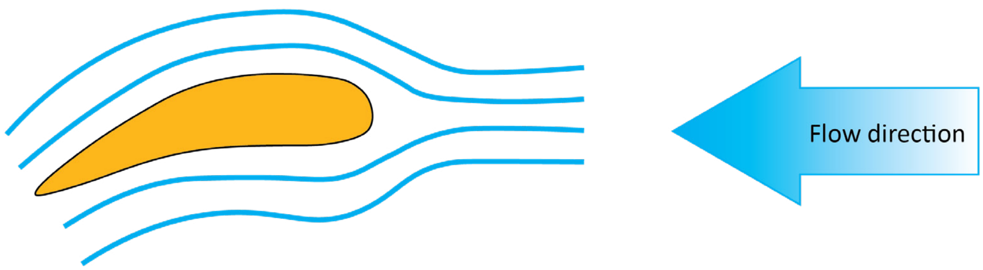

Figure 8. Streamline flow around an airfoil at an acceptable Angle of Attack

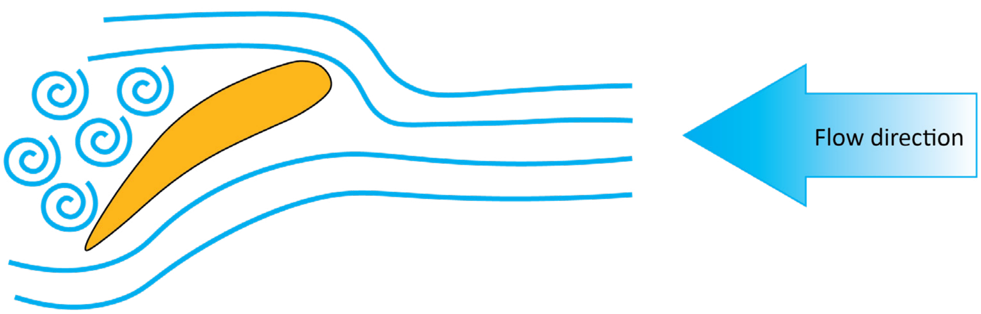

Figure 9. Flow separation at a high Angle of Attack

4. Determining Lift and Drag forces using Ansys Discovery

Ansys Discovery is a simulation-driven design tool that combines geometry modeling, instant physics simulation, and high-fidelity simulation in a single easy-to-use experience. You can find more information on how to access Ansys Discovery through the Ansys Discovery Student Version and Ansys Discovery 3D Design webpage. In this case study, this software has been used to simulate airflow over an airfoil. This fluid flow problem is solved using the following steps highlighted in Figure 10[4]. For more information about setting up the fluid flow problems refer to this lecture presentation available at Ansys Education Resources.

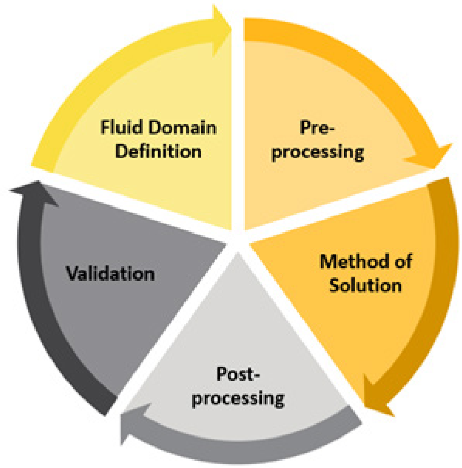

Figure 10. Setting up a fluid flow problem

4.1 Fluid Domain Definition

Consider the airwing model, shown in Figure 11(a). The simulation file (Plane Wing Model.dsco) is provided along with this case study [2]. The fluid enclosure is setup as shown in the Figure 11(b). The material of the fluid is defined as air.

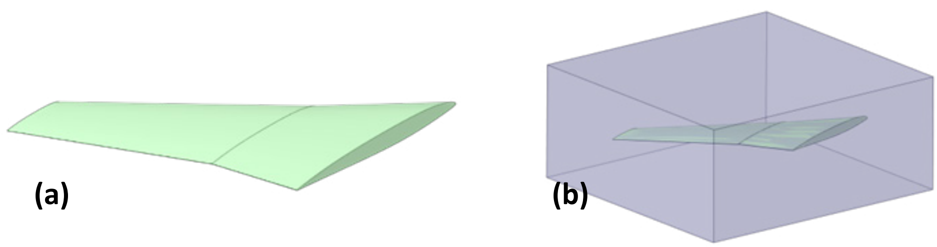

Figure 11. (a) Plane Wing (b) Plane wing with fluid enclosure

4.2 Preprocessing

Consider a low velocity air flow inlet velocity of 40 m/s and outlet relative pressure of 0 Pa. The boundary conditions are then set up as shown in the Figures 12 (a) and (b).

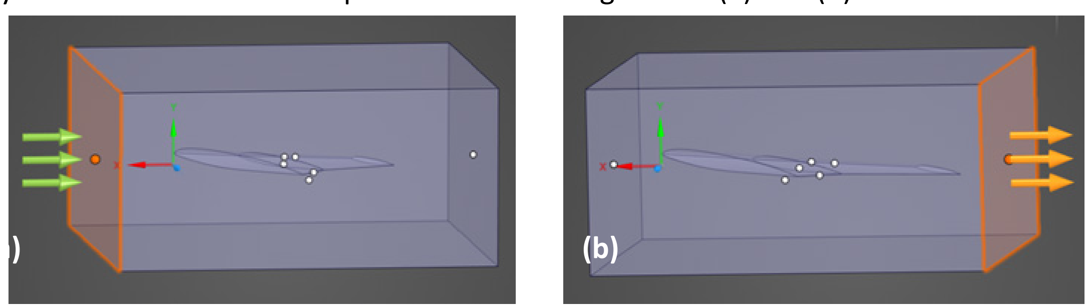

Figure 12. (a) Inlet Boundary Conditions (b) Outlet Boundary Conditions

Force monitors are defined on the surface of the airwing to determine the lift and drag forces acting on it. In this case, the force monitor defined in the X-direction would provide us with drag force and the force monitor defined in the Y-direction would provide us with lift forces. This can be observed in Figure 13. To learn more about how to setup the simulation, you can visit the tutorials on Ansys Learning Forum.

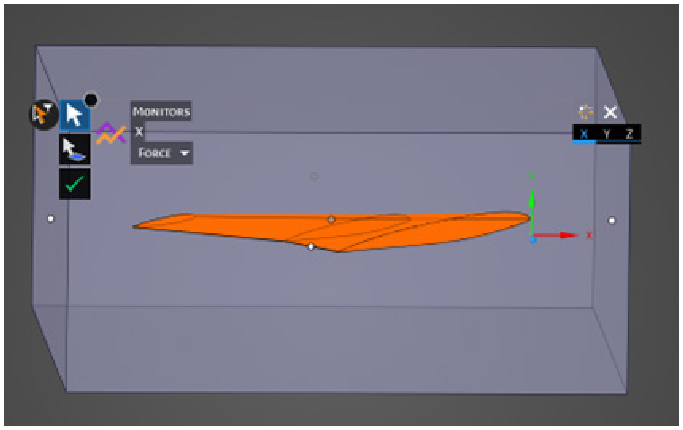

Figure 13. Defining force monitors

4.3 Solution, Post processing and Validation

This model is then solved using the Explore mode (in case your device does not meet the GPU requirements, the refine mode can be used as well, however the simulation will take a few minutes longer to run). The fidelity of the model used in this case study is 26mm. A low fidelity has been chosen in order to reduce computational time. The results of this airflow can be visualized using the results view arc shown in Figure 14(a). It is important to note that the results show below are meant to qualitatively teach the concepts of lift and drag in an airfoil and are not indicative of an accurate engineering solution.

Let us start with velocity using the directional plot option. This type of plot can be moved along the cross-section and rotated as required. In Figure 14(b), we can clearly see that the velocity at the top of the airfoil is higher than that of the bottom. By moving the directional plot along the length of the airwing we notice that this holds true for the span of the wing.

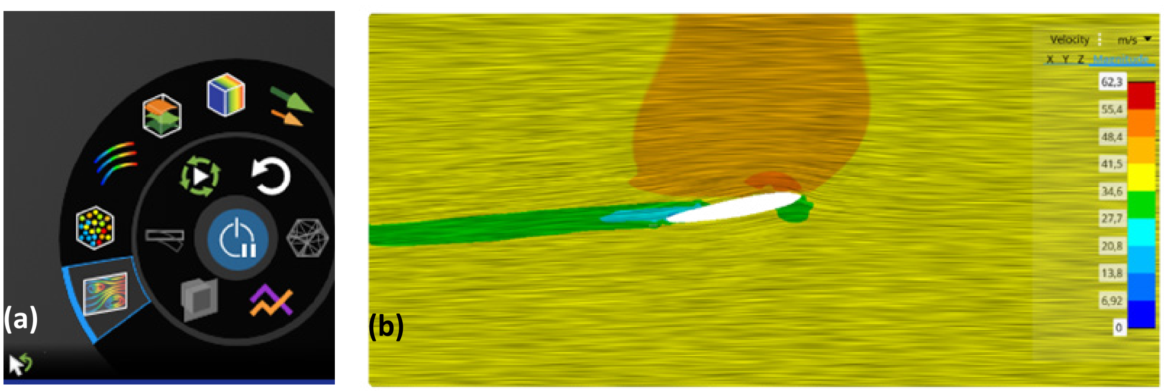

Figure 14. (a) Results View Arc -Directional Plot (b) Directional Plot - Velocity Plot

Using Bernoulli's theorem, we can predict that this would cause a low-pressure region at the top compared to that of the bottom. This can be confirmed by plotting the pressure over the span of the wing. Figure 15(a) indicates how to plot the pressure contour on the faces of the wing. In Figures 15(b) and 15(c), we can clearly observe that the pressure at the top of the airfoil is lower than that of the bottom of the airfoil. The pressure distribution does not appear smooth because of the coarse mesh, to a smoother pressure distribution a finer mesh should be used.

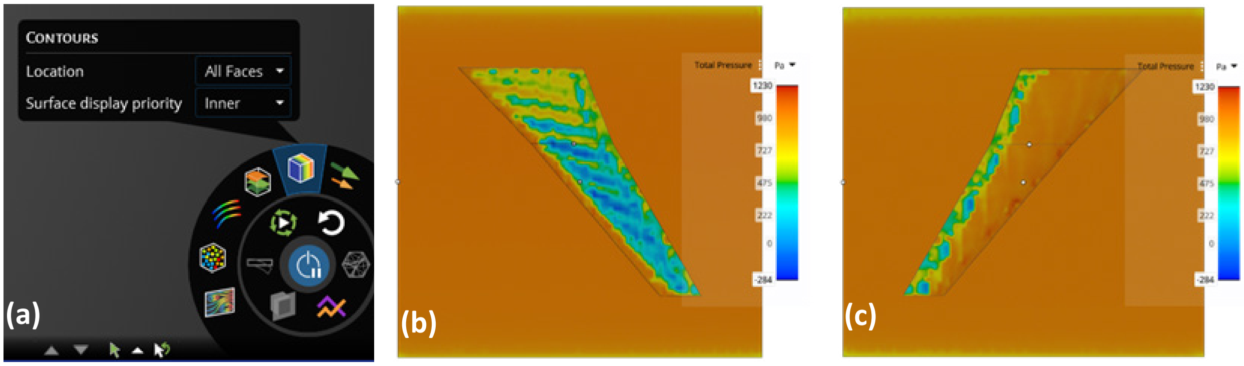

Figure 15. (a) Results View Arc – Contour Plot (b) Pressure Contour – Top View (c) Pressure Contour – Bottom View

This difference in pressure and velocity result in lift and drag forces being exerted on the airfoil. This can be measured using the force monitors we defined earlier. We obtain the value of lift and drag forces as 386 N and 71 N respectively.

4.4 Influence of Angle of Attack

Let us now see how air flow over an airfoil is influenced by the angle of attack. Let us rotate the wing by 10˚ about the Z axis. This results in an increased lift and drag forces of 580 N and 220 N respectively. This shows us that increased lift because of higher angle of attack comes with the compromise of increased drag. Let us now increase the angle of attack to 60˚. This results in flow separation as shown in Figure 16(b). This causes lift force to decrease to 377N but drag force to increase to 1110N. This explains why airplanes do not take off at a very high angle of attack as this would cause the plane to stall leading to an accident.

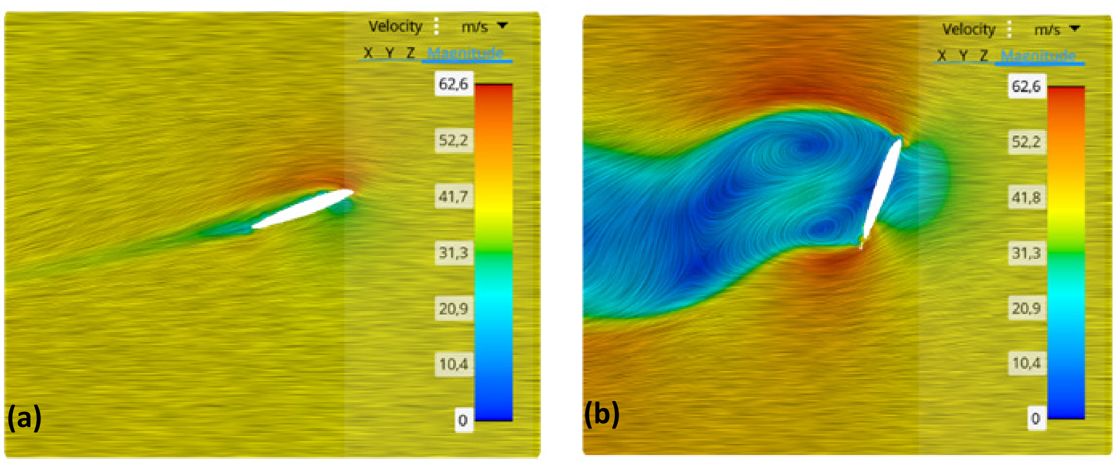

Figure 16. (a) angle of attack 10˚ (b) angle of attack 60˚

5. From the Ideal to the Real world

In reality, the lift and drag forces required by an aircraft is different based on the phase of the flight. That is why aircraft wings are designed with flaps and slats to form a mechanism as opposed to a single component. This allows the pilot to adjust the geometry of the airfoil to the phase of the flying. For example, the aircraft require high lift forces during takeoff whereas during cruising the aircraft does not require high lift forces but require low drag forces to increase speed and decrease consumption of the fuel at the same time.

Figure 17. Airplane during takeoff and cruising

6. Further Steps

This case study was an example of a simple aerodynamic analysis of an airplane wing. A more complex analysis would include using a finer mesh and studying the exact influence of the angle of attack on lift by parameterizing the angle of attack in Discovery and finding out at which value does stalling takes place. Additionally, the influence of the camber line and angle of attack can be studied further by modeling the wing with slats and flaps. The Refine mode of Discovery or our flagship product Fluent can be used to get more accurate engineering solutions that can be used to improve the design of the airfoil based on the type of usage.

7. References

1. F. White, H. Xue, 'Fluid Mechanics', 9th edition, McGraw Hill, Pg 404-410
2. 'Understanding Airplane Lift and Physics', Exploring the Physics of Lift | Ansys Innovation Courses
3. 'Exploring Drag and Physics', Exploring the Physics of Drag | Ansys Innovation Courses
4. M. Saravana Kumar, L. Mohee, Ansys Education Resource – 'Lecture Unit: Introduction to Fluid Mechanics with Ansys Discovery', Lecture Unit: Introduction to Fluid Mechanics with Ansys Discovery | Ansys

Use and Reproduction

Document Information

This case study is part of a set of teaching resources to help introduce students to structures, fluids, or heat transfer (physics areas supported by Ansys Discovery).

Ansys Education Resources

To access more undergraduate education resources, including lecture presentations with notes, exercises with worked solutions, microprojects, real life examples and more, visit www.ansys.com/education-resources.

Feedback

If you notice any errors in this resource or need to get in contact with the authors, please email us at education@ansys.com.

Acknowledgments

Susannah Cooke, Luca Masi, Kaitlin Tyler

ANSYS, Inc.
Southepointe
2600 Ansys Drive
Canonsburg, PA 15317
U.S.A.
724.746.3304
ansysinfo@ansys.com