



Case Study

Transonic aerodynamics of NACA 0012 airfoil using Ansys Fluent

Developed and curated by the Ansys Academic Development Team

Ravindra Arjun Shirsath
Ansys Academic Development Team

education@ansys.com

Summary

Flying faster has always been the dream of human beings ever since we learned to fly. The invention of jet engines allowed aircraft to fly at much higher speeds. However, investigating an external flow past an airfoil at relatively higher speeds has always been fascinating as well as challenging. The strong need for computational solutions arises as conducting experiments to capture underlying flow physics, particularly at high speeds is not only time consuming but also costly. Further, the data acquisition and flow visualization techniques are complex as well as limited. Thus, Ansys Fluent, which is a popular commercial CFD software with vast capabilities including geometry preparation, meshing, solutions, and post processing makes it easy to predict accurate flow physics under various circumstances. Particularly, the accuracy and ability to predict formation of shocks, pressure and velocity variations and flow visualization at such high speeds are outstanding.

In the present case study, steady state simulations are performed over a NACA 0012 airfoil over a wide range of Mach numbers and are compared with the existing literature to show the accuracy of the simulations. The investigations are further extended to understand the concepts of critical Mach number and drag divergence Mach number. The results include the variation of lift and drag coefficients with Mach number, pressure and Mach number distribution and flow visualization. The present study will serve as a starting point for mechanical, aeronautical, and aerospace engineering students to better understand the fundamental concepts associated with external aerodynamics along with the visualization of phenomenon of shock formation and drag divergence.

Table of Contents

1. Introduction.....	3
2. Problem Statement	3
3. Geometry and Mesh.....	4
4. Solution Methodology.....	4
5. Results and Discussion	4
5.1 Purely subsonic flow past an airfoil	5
5.2 Mixed flow past an airfoil	6
6. Further Steps.....	11
7. References	11

1. Introduction

The aerodynamic characteristics of an airfoil greatly differ depending on the operating conditions. The lift and drag coefficients along with the pressure and velocity distribution are distinctly different in transonic flow as compared to low-speed subsonic flow. Transonic flow occurs when there is mixed sub- and supersonic local flow in the same flow field. Even if the incoming free stream Mach number is subsonic, due to the acceleration of flow particularly around an airfoil mostly on the upper surface the flow may become supersonic. Usually, the supersonic region of the flow is terminated by a shock wave, allowing the flow to slow down to subsonic speeds. Shock waves are basically very thin regions across which the flow properties such as temperature, pressure, velocity, density *etc.* vary drastically. Depending on an angle that the shock wave makes with the incoming velocity vector, they can be classified as normal shocks if the angle is **90°** and an oblique shock otherwise.

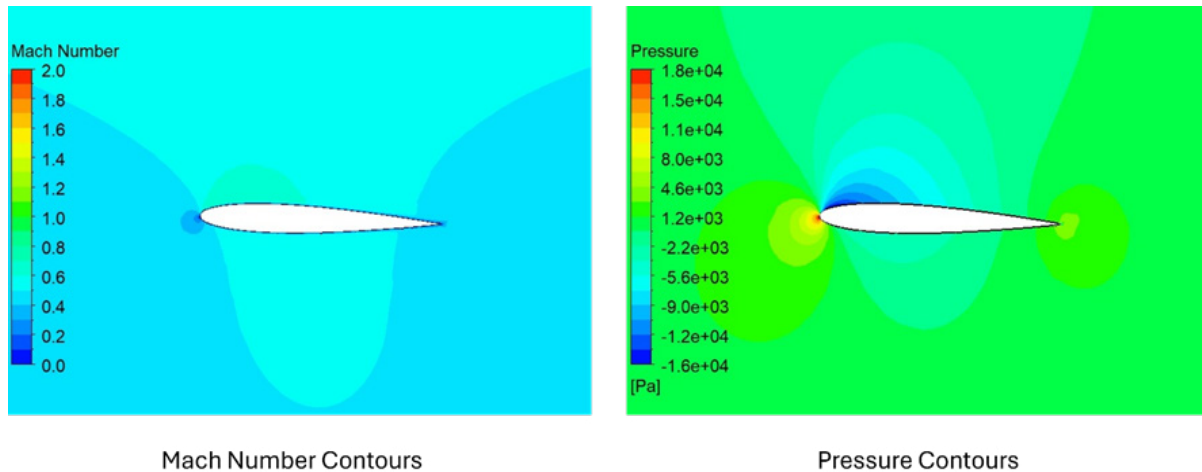


Figure 1 : Mach number and pressure variation over NACA 0012 airfoil at $\alpha=2^\circ$ and $M_\infty=0.5$.

The variation of Mach number around NACA 0012 airfoil at $\alpha=2^\circ$ and free stream Mach number $M_\infty=0.5$ is shown in Figure 1(a). It is seen that as the flow passes over an airfoil, the flow is slowed down on the lower surface and accelerated on the upper surface but not exceeding $M=1$ anywhere in the domain. It is this acceleration of the flow that causes low pressure region on the upper surface of an airfoil as seen from Figure 1(b), thereby resulting in net pressure difference between the upper and lower surfaces and hence causes positive lift to be produced. Since the incoming free stream Mach number is relatively low, no region of sonic or supersonic flow is observed anywhere around the airfoil and thus this condition is called 'fully subsonic flow'. With increasing free stream Mach number, these things are bound to change and hence the effect of increasing Mach number is investigated in this case study.

2. Problem Statement

In the present work, the aerodynamic characteristics of a NACA 0012 airfoil at $\alpha=2^\circ$ are investigated over a range of free stream Mach number, $0.5 < M_\infty < 1.5$. Increasing Mach number can cause shock stall and drag divergence, which can be understood by visualizing the pressure field, and investigating changes in the lift and drag coefficients.

3. Geometry and Mesh

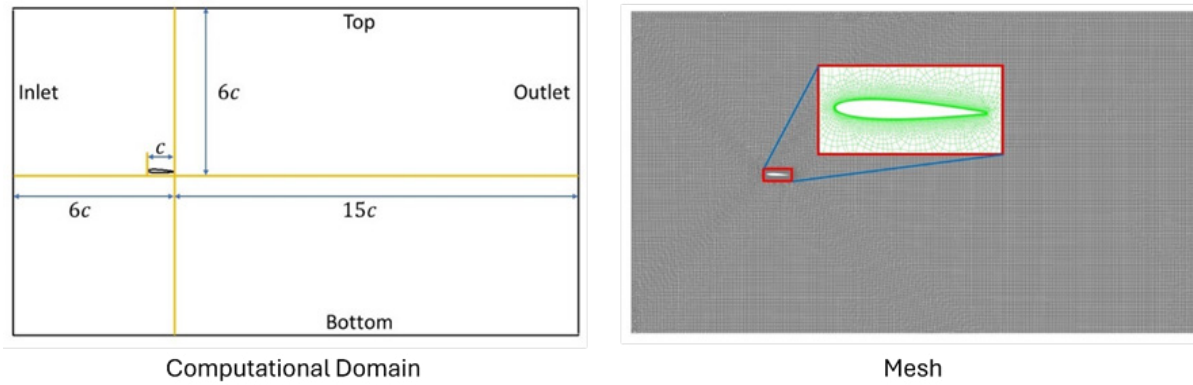


Figure 2 : Details of the geometry, mesh and computational domain.

The airfoil considered for present study has NACA 0012 (symmetric) profile. The computational domain used is shown in Figure 2(a) that extends $6c$ upstream as well as on top and bottom while $15c$ downstream as it is mostly the downstream region which is highly influenced by the presence of the airfoil. The trailing edge of airfoil coincides with the origin while. The mesh used for airfoil has $\approx 114,000$ nodes resulting in $\approx 113,000$ elements. Highly refined mesh is employed in the vicinity of the airfoil with 50 layers of inflation and specified first layer height of 1.3×10^{-6} to ensure that near wall phenomenon is captured correctly and the specific requirements of the selected turbulence model in terms of y^+ are met. Though the mesh contains both quadrilateral and triangular elements, most of the domain predominantly consists of quadrilateral elements as seen from Figure 2(b).

4. Solution Methodology

Using Ansys Fluent, the following methodology is used. Numerical solver is set up with pressure-based type, absolute velocity formulation and pressure-velocity coupling is dealt with SIMPLE algorithm. The working fluid is chosen to be an air as an ideal gas and dynamic viscosity modeled using Sutherland law. The least square cell-based method is employed for estimating the gradients while second order upwind scheme is used for discretizing the equations. The inlet is specified to be pressure far-field with the respective free stream Mach number, M_∞ with the turbulence properties specified through turbulent intensity of 1% and turbulent viscosity ratio of 10. Top and bottom boundaries are also imposed with the same condition while the outlet is set to be pressure outlet. Zero-gauge pressure is initialized throughout. The boundary at the airfoils is modeled to be solid wall. The selection of turbulence model is crucial, especially for the flows at higher Mach number. When using $k-\omega$ (SST) model, our computational results are seen to agree well with the experimental results, hence, B (SST) is used throughout the analysis. The criterion for convergence is set to be 10^{-5} and additionally the force coefficients on the airfoil are monitored individually ensuring that simulations are run for enough iterations.

5. Results and Discussion

This section explains the key results obtained by increasing the free stream Mach number while keeping the angle of attack fixed at $\alpha = 2^\circ$. The formation of shock wave and corresponding pressure distribution is shown with the help of $C_p - x/c$ plot. The movement of shock with subsequent increase in Mach number is further shown through Mach number and pressure contours. The flow separation caused by shock wave resulting in rising drag and falling lift coefficients is illustrated further.

5.1 Purely subsonic flow past an airfoil

The effect increasing Mach number from $M_\infty=0.5$ to $M_\infty=0.6$ on the Mach and pressure contours as seen from Figure 3 clearly indicates that though the flow is accelerated mostly on the upper surface, it never reaches sonic condition anywhere. Noticeably a small region of highly subsonic flow is also seen at the leading edge along with a very low-pressure region on the top surface close to the leading edge.

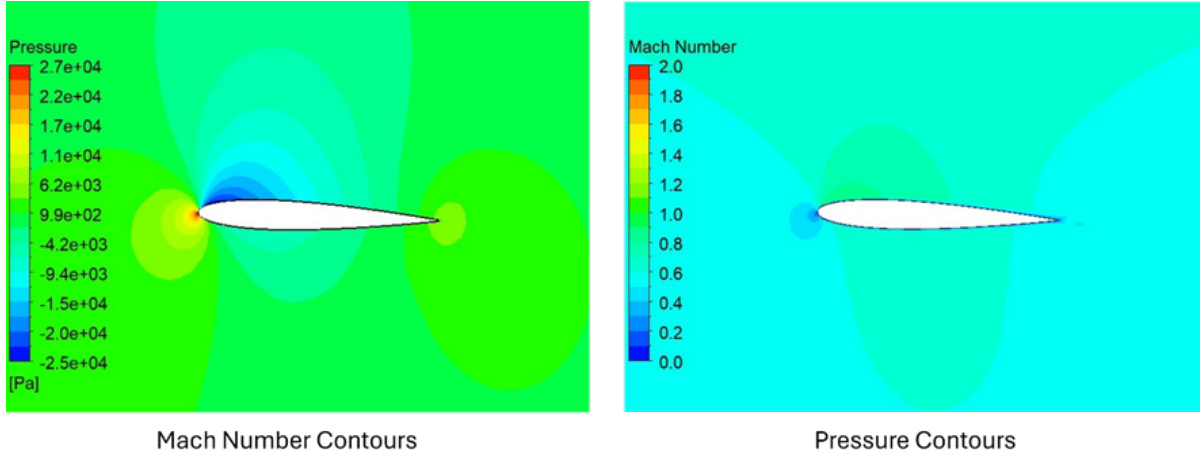


Figure 3 : Mach number and pressure variation over NACA 0012 airfoil at $\alpha=2^\circ$ and $M_\infty=0.6$.

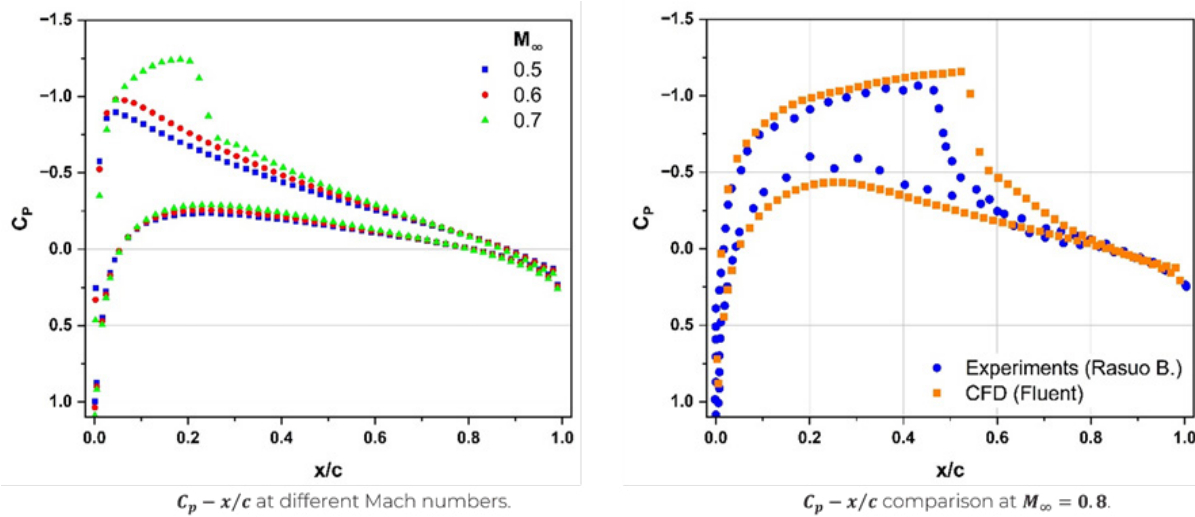


Figure 4 : Variation of pressure coefficient at $\alpha=2^\circ$ and $M_\infty=0.5, 0.6$, and 0.7

As can be seen more precisely from $C_p - x/c$ graphs that for $M_\infty < 0.7$ the flow is purely subsonic and a typical C_p distribution is observed from the left graph in Figure 4. However, somewhere in between, for $0.6 < M_\infty < 0.7$ the flow is accelerated to an extent that it becomes supersonic in some region, this marks the beginning of mixed flow pattern in which the flow is partly subsonic and partly supersonic. The region of supersonic flow is terminated by a shock and thus there occurs a sudden jump in the flow properties as is evident from $C_p - x/c$ graphs that for $M_\infty=0.7$ in Figure 4(left) and is compared with the experimental findings from Rasuo B.[1] in right graph in Figure 4. The mixed flow pattern has many complications and thus needs careful understanding.

5.2 Mixed flow past an airfoil

It is now known that due to acceleration of the flow, the Mach number at some point on the surface of an airfoil is higher than the free stream Mach number. Thus, with increasing the free stream Mach number, there occurs a condition where the flow at least at a point reaches the sonic condition. This phenomenon is of particular interest.

Critical Mach Number (M_{cr}) – It is the free stream Mach number at which a sonic flow is observed for the first time at least at a point on the surface of an airfoil and is called the lower limit of transonic flow region or mixed flow pattern.

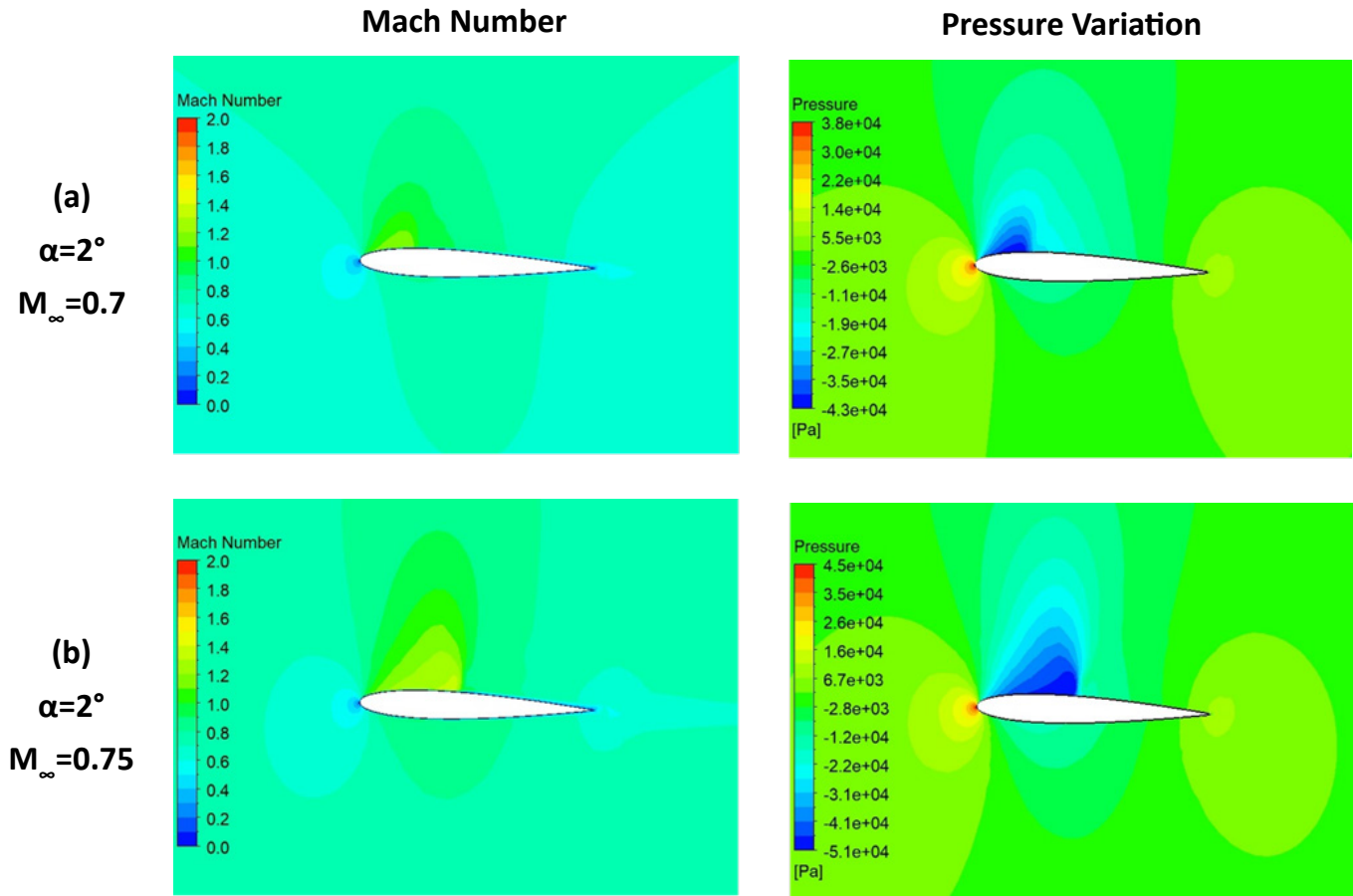


Figure 5 : Mach number and pressure variation over NACA 0012 airfoil at $\alpha = 2^\circ$ and (a) $M_\infty = 0.7$ and (b) $M_\infty = 0.75$.

Once the free stream Mach number exceeds the critical Mach number, a small region of supersonic flow, terminated by the normal shock is observed on the upper surface of an airfoil as shown in Figure 5(a). Further increase in free stream Mach number increases the size of this supersonic region thereby moving the normal shock towards the trailing edge as shown in Figure 5(b).

With further increase in free stream Mach number a similar supersonic region terminated by normal shock is formed on the lower surface as well, whose size goes on increasing with increasing free stream Mach number. The Mach and pressure contours with increasing free stream Mach numbers are shown in Figure 6a-d.

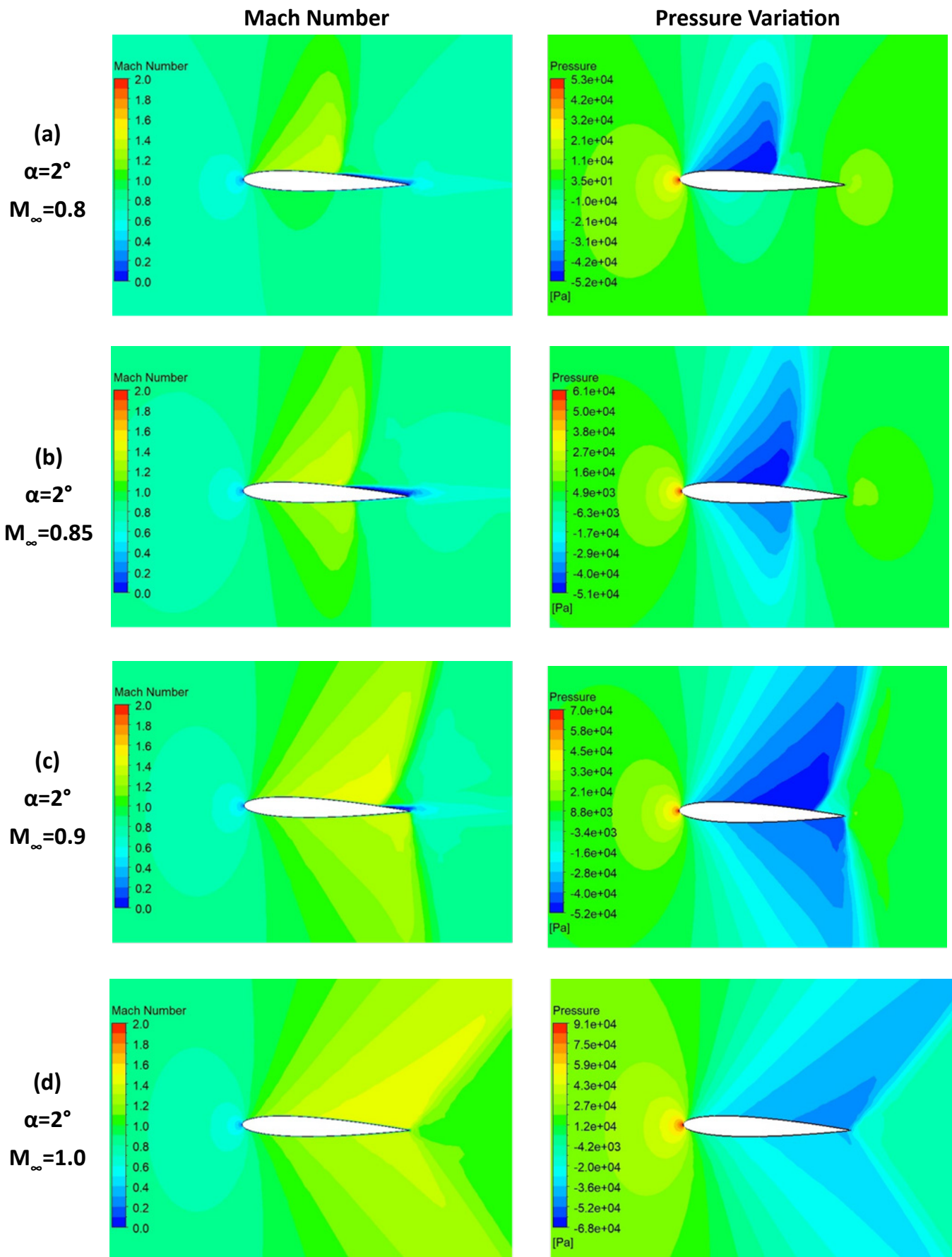


Figure 6 : Mach number and pressure variation over NACA 0012 airfoil at $\alpha = 2^\circ$ and
(a) $M_\infty = 0.8$, (b) $M_\infty = 0.85$, (c) $M_\infty = 0.9$, and (d) $M_\infty = 1.0$.

While both the shocks gradually move backwards, it is important to note that the lower surface shock moves towards the trailing edge faster than the upper surface shock due to the relatively high-pressure region on the lower side of an airfoil. When both the shocks have reached the trailing edge, further increase in free stream Mach number will only change their angle, resulting in “Bifurcated trailing edge shock pattern”. It appears that now the flow has entirely become supersonic but remember that the free stream Mach number has just reached unity.

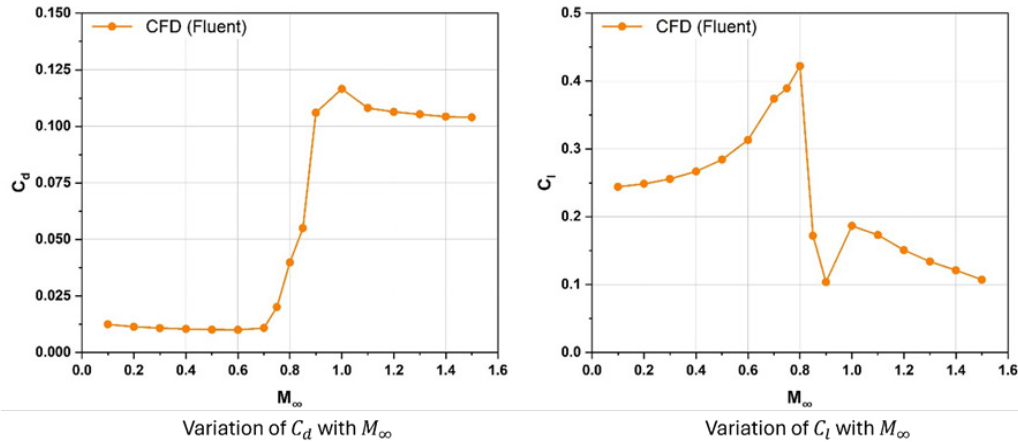


Figure 7 : Variation of drag and lift coefficient with M_∞

It is important to estimate the drag and lift coefficients to understand the complications of this type of mixed flow pattern. As seen from Figure 7(left) the C_d marginally decreases with increasing Mach number up to $M_\infty = 0.6$ and corresponding C_l increases considerably. As observed earlier, the sonic region appears at $M_\infty = 0.7$ indicating that the critical Mach number is somewhere in between $0.6 < M_{cr} < 0.7$. However, it appears that reaching critical Mach number is not a problem as though the C_d increases slightly, there is a considerable increase in C_l . However around $M_\infty \approx 0.8$, the C_d starts increasing rapidly and the C_l decreases drastically. It is this Mach number that is more important and called as drag divergence Mach number.

Drag divergence Mach number (M_{dd}) – It is the free stream Mach number at which the shock terminating the region of supersonic flow on the upper surface moves backwards and becomes strong enough to cause the flow separation. The phenomenon of separation due to shock is called shock-stall and associated drag rise is called as transonic drag rise.

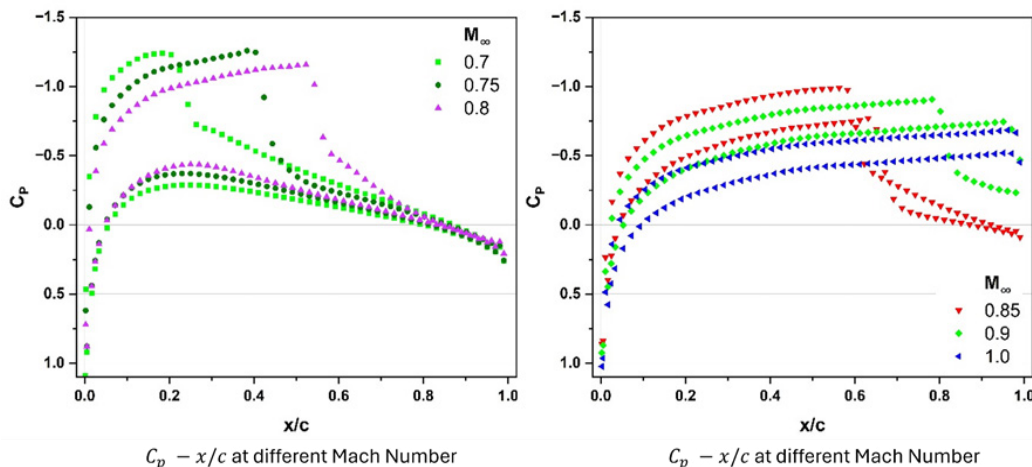


Figure 8: . C_p - x/c at different Mach numbers.

The C_p - x/c plots from Figure 8 shows clearly the formation and movement of the terminating shock at different intermediate free stream Mach numbers. Being the region of low pressure, the upper surface is seen to have larger changes in C_p before and after the shock.

The trend of mixed flow pattern does not end once the free stream Mach number exceeds unity. The moment $M_\infty > 1$ a detached bow shock is formed at a distance ahead of the leading edge resulting in a region of subsonic flow between the shock and the leading edge as illustrated in the following Mach and pressure contours.

However with further increase in the free stream Mach number, this detached bow shock tends to move closer to the leading edge thereby reducing the size of the subsonic region as observed from Figure 7 a-e with corresponding pressure contours alongside. Until this point, there still happens to be a mixed flow pattern. Further increase in free stream Mach number brings a stage when the detached bow shock moves close and gets attached to the leading edge, resulting in a fully developed supersonic flow all around the airfoil and this marks the end of the region of mixed flow pattern, corresponding free stream Mach number is called as shock wave attachment Mach number.

Shock wave attachment Mach number (M_{SA}) – It is defined as the free stream Mach number at which the detached bow shock moves close and gets attached to the leading edge of an airfoil marking an end to the mixed flow pattern and resulting in fully supersonic flow there after.

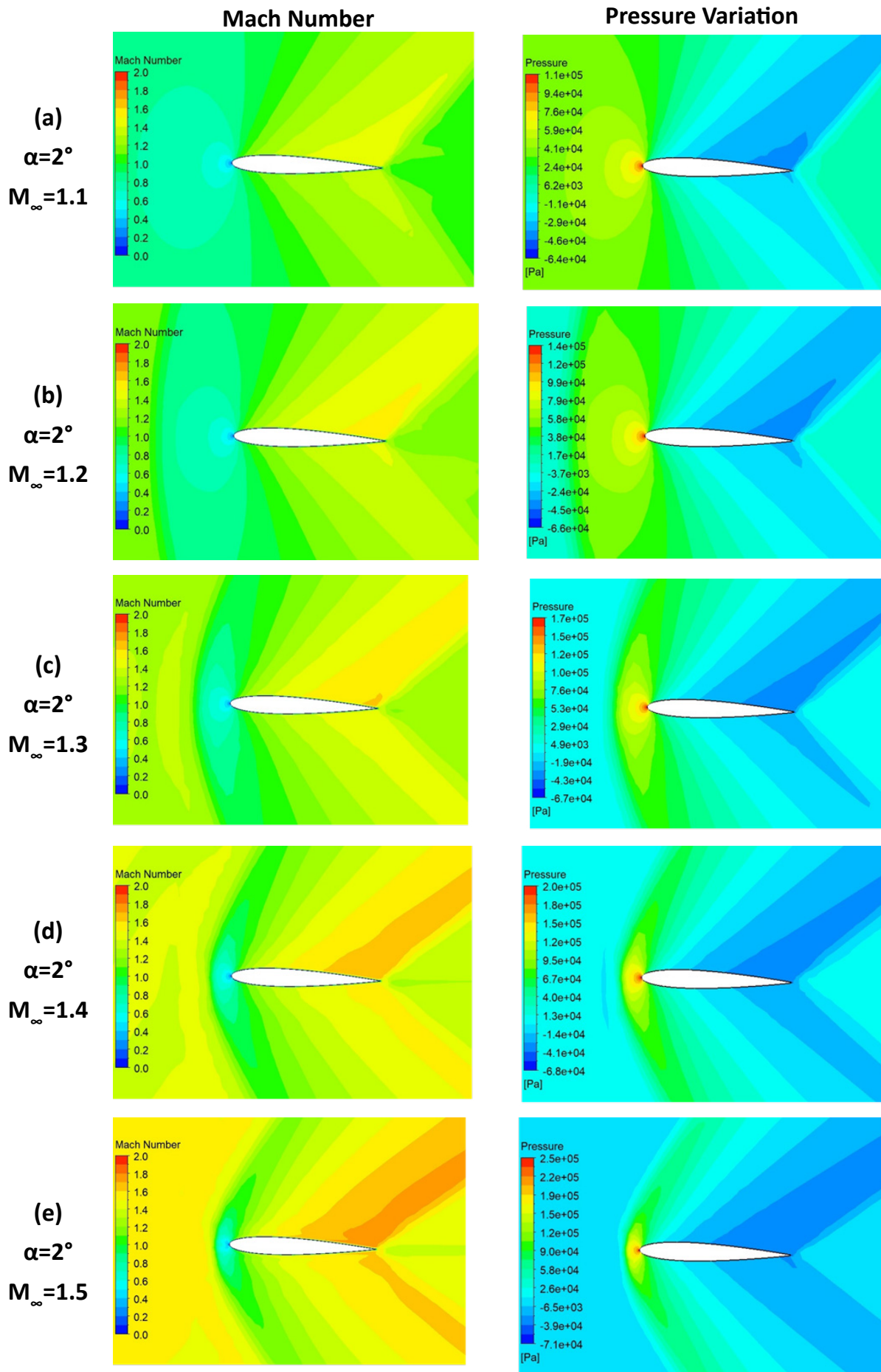


Figure 9: Mach number and pressure variation over NACA 0012 airfoil at $\alpha=2^\circ$ and (a) $M_\infty=1.1$, (b) $M_\infty=1.2$, (c) $M_\infty=1.3$, (d) $M_\infty=1.4$, and (e) $M_\infty=1.5$

6. Further Steps

In the present case study, the high speed aerodynamic characteristics of a single NACA 0012 airfoil are investigated using Ansys Fluent. The results obtained for the pressure distribution over an airfoil are seen to fairly match with the experimental data available in the literature. The investigation clearly shows that the highly complicated phenomenon of formation of shocks that results in mixed flow pattern can be investigated using advanced CFD software such as Ansys Fluent to predict the aerodynamic characteristics accurately. With the help of flow visualization through the contours of Mach number and pressure it is easy to demonstrate and understand the formation and movement of shock with varying Mach number. The concepts of critical, drag divergence and shock wave attachment Mach numbers can be shown more precisely.

These steady state investigations show preliminary analysis and the same can be extended to further investigate the performance of different airfoils, effect of thickness, camber and leading-edge nose radius of the airfoils. The effect of an angle of attack for a fixed Mach number can also be investigated. Additionally, transient simulations are expected to allow more investigation of how the shock wave and flow separation develop.

7. References

- [1] Rasuo B., An Experimental and Theoretical Study of Transonic Flow About NACA 0012 Airfoil, 24th Applied Aerodynamics Conference, 5 – 8 June, 2006, San Fransisco, California. AIAA 2006 - 3877.
Charles L. Landson, Effects of Independent Variation of Mach and Reynolds Numbers on the Low-Speed Aerodynamic Characteristics of the NACA 0012 Airfoil Section, NASA Technical Memorandum 4074, 1988.
- [2] Gregory N. and O'Reilly C. L., Low-Speed Aerodynamic Characteristics of NACA 0012 Airfoil Section, including the Effect of Upper Surface Roughness Simulating Hoar Frost, Ministry of Defence, Aeronautical Research Council, Reports and Memoranda No. 3726, 1970.

© 2024 ANSYS, Inc. All rights reserved.

Use and Reproduction

The content used in this resource may only be used or reproduced for teaching purposes; and any commercial use is strictly prohibited.

Document Information

This case study is part of a set of teaching resources to help introduce students to topics related to fluids.

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

Here at Ansys, we rely on your feedback to ensure the educational content we create is up-to-date and fits your teaching needs.

Please click the link here out a short survey (~7 minutes) to help us continue to support academics around the world utilizing Ansys tools in the classroom.

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

If you've ever seen a rocket launch, flown on an airplane, driven a car, used a computer, touched a mobile device, crossed a bridge or put on wearable technology, chances are you've used a product where Ansys software played a critical role in its creation. Ansys is the global leader in engineering simulation. We help the world's most innovative companies deliver radically better products to their customers. By offering the best and broadest portfolio of engineering simulation software, we help them solve the most complex design challenges and engineer products limited only by imagination.

visit www.ansys.com for more information

Any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. All other brand, product, service and feature names or trademarks are the property of their respective owners.

© 2024 ANSYS, Inc. All Rights Reserved.