CFD analysis of the aerodynamic characteristics of biconvex airfoil at compressible and high Mach numbers flow

Ebrahim Hosseini¹

© Springer Nature Switzerland AG 2019

Abstract

In the present study, numerical investigation of the turbulent flow over a biconvex airfoil at compressible and high Mach numbers flow is done using computational fluid dynamics (CFD). The flow is considered as turbulent, two-dimensional, steady and compressible. For this purpose, three Reynolds number of 2.4×10^7 , 2.9×10^7 and 3.3×10^7 are considered. The simulations are implemented using the commercial software Ansys Fluent 16. The results are obtained with Reynolds-averaged Navier–Stokes (RANS), and for simulating the flow turbulence, SST $k-\omega$ turbulence model is carried out. The results show that the lift coefficient (C_L) and drag coefficient (C_D) increase by the increment of the angle of attack (α). The lift-to-drag ratio (C_L/C_D) is improved by increasing the Mach number (Ma) and cause to delay the boundary layer separation. Increasing the Mach number affects the stall angle which causes to increase it from $\alpha = 22^\circ$ to $\alpha = 30^\circ$ from Ma = 1 to Ma = 1.4.

Keywords Biconvex airfoil · Computational fluid dynamics (CFD) · Mach number · Lift coefficient · Drag coefficient

List of symbols

c	Airfoil chord length
α	Angle of attack
ρ	Density
C_D	Drag coefficient
U_∞	Free stream velocity
C_L	Lift coefficient
Y^+	Normal distance in wall coordinates
P	Pressure
C_p	Pressure coefficient
Re	Reynolds number
τ	Shear stress
ε	Turbulent dissipation
k	Turbulent kinetic energy
μ	Viscosity

1 Introduction

Supersonic flow over aeronautical configurations has a wide scope in the aerospace applications [1–3]. Biconvex and double-wedge airfoils are extensively employed in aerospace engineering and many works have been done by researches [4, 5]. The Supersonic aircraft uses biconvex and double wedge airfoil but any analysis data for these two airfoils is not available easily [6]. It is important to have the knowledge of aerodynamic parameters in order to design wings. To analyze and obtain the results of supersonic flight, CFD method can be helpful to understand wing analysis at supersonic flow. Askari et al. [4] numerically studied compressible flow around the double wedge and biconvex airfoils by CFD. They concluded that the aerodynamic coefficients gained from both analytical and numerical methods were in good agreement. Olejniczak et al. [7] investigated numerically and experimentally a double-wedge airfoil and measured the surface pressure and heat transfer coefficient. They indicated that

✉ Ebrahim Hosseini, ebrahim.2019.hosseini@gmail.com | ¹Department of Mechanical Engineering, Dezful Branch, Islamic Azad University, Dezful, Iran.

separation zones in numerical analyses are smaller than in experiment results. Raghunathan et al. [8] numerically considered oscillations of the shock wave over biconvex airfoil. They found that the shock-induced separation has a great impact on the origin of shock oscillations. Al-Garni et al. [9] experimentally and numerically investigated the aerodynamic coefficients of the double-delta wing. Their results of the surface pressure coefficient distribution and vortex breakdown location were in excellent agreement with experimental data. Hamid et al. [10] numerically investigated the compressible flow over a biconvex circular arc airfoil. They found that the unsteady shock movement generates the transient shock-boundary layer interaction and leads to create the separation. Rahman et al. [11] numerically analyzed the self-excited shock oscillation over a biconvex circular arc airfoil with and without cavity. Their results demonstrated that the airfoil with cavity dramatically decreased the flow field unsteadiness. Also, Rahman et al. [12] modified the geometry of the baseline airfoil to consider the effects of cavity size on the control of transonic internal flow. They concluded that the average RMS of pressure oscillation around the airfoil with an open cavity has decreased dramatically.

Although many numerical investigations have been done to investigate the aerodynamic performance and flow characteristics of biconvex airfoils but rare studies have been focused on the simulation of turbulent flow around the thin symmetric airfoils especially the biconvex airfoil at compressible and high Mach numbers flow using CFD technique. For this purpose, three different $Ma = 1$, $Ma = 1.2$ and $Ma = 1.4$ are selected to consider the flow characteristics of the biconvex airfoil and simulate the flow separation at high Mach numbers. Moreover, all steps of the investigation including aerodynamic performance, simulation, calculation and comparison of various Mach numbers and angels of attacks are discussed in detail.

2 Grid generation and boundary conditions

A C-type grid is generated using the mesh generation module of the Ansys software. The C-type grid is applied to form a regular mesh. The grid extends from 12 chords upstream to 20 chords downstream and the upper and lower boundary extends 12 chords from the profile which is shown in Fig. 1. The upstream should be selected at a distance where the flow regains its normal state and the presence of the object in the flow causes no changes in that location.

Different computational domains are selected to consider domain extent independence test as shown in Fig. 2. For this purpose, the pressure coefficient (C_p) is calculated at Mach number of 1 and angle of attack of 20° . It is found that the domain with 12 chords upstream to 20 chords downstream of the airfoil is sufficient for the present simulation. Inlet, upper and lower boundaries are considered

Fig. 2 Domain extent independence study

Fig. 1 Boundary conditions and a view of the entire grid generation

as inlet velocity and outlet are considered as pressure outlet. Furthermore, the no-slip boundary condition is used for the airfoil wall. A closer view of the grid generation is illustrated in Fig. 3.

3 Governing equations and numerical method

The compressible Navier–Stokes equations are selected as governing equations of the flow field. These governing equations can be expressed as follows for steady and two-dimensional flow [13]:

$$\frac{\partial(\rho u_x)}{\partial x} + \frac{\partial(\rho u_y)}{\partial y} = 0 \quad (1)$$

$$\partial_x(\rho u_x u_x) + \partial_y(\rho u_x u_y) = -\frac{\partial P}{\partial x} + \partial_x \tau_{xx} + \partial_y \tau_{xy} \quad (2)$$

$$\partial_x(\rho u_x u_y) + \partial_y(\rho u_y u_y) = -\frac{\partial P}{\partial y} + \partial_x \tau_{xy} + \partial_y \tau_{yy} \quad (3)$$

$$\begin{aligned} \tau_{xx} &= -\frac{2}{3}\mu\left(\frac{\partial u_x}{\partial x} + \frac{\partial u_y}{\partial y}\right) + 2\mu\left(\frac{\partial u_x}{\partial x}\right) \\ \tau_{xy} &= -\frac{2}{3}\mu\left(\frac{\partial u_x}{\partial x} + \frac{\partial u_y}{\partial y}\right) + \mu\left(\frac{\partial u_x}{\partial x} + \frac{\partial u_y}{\partial x}\right) \\ \tau_{yy} &= -\frac{2}{3}\mu\left(\frac{\partial u_x}{\partial x} + \frac{\partial u_y}{\partial y}\right) + 2\mu\left(\frac{\partial u_y}{\partial y}\right) \end{aligned} \quad (4)$$

where u is velocity, μ is the viscosity, ρ is density, P is pressure and τ is shear stress.

In this study, RANS equations are solved with the SST k – ω turbulence model for turbulence simulation. SST k – ω turbulence model provides good predictive capability for flows with separation [14–17].

The transport equations for the SST k – ω model are given by [14]:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left(\Gamma_k \frac{\partial k}{\partial x_j}\right) + G_k - Y_k \quad (5)$$

$$\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_i}(\rho \omega u_i) = \frac{\partial}{\partial x_j}\left(\Gamma_\omega \frac{\partial \omega}{\partial x_j}\right) + G_\omega - Y_\omega + D_\omega \quad (6)$$

where Γ_k and Γ_ω define as the effective diffusivity of k and ω , respectively. G_k and G_ω indicate the generation of k and ω due to mean velocity gradients, respectively. Y_k and Y_ω are the dissipation of k and ω , respectively. D_ω denotes the cross-diffusion term.

All present simulations are carried out with the density-based finite volume solver of the commercial software ANSYS Fluent 16 due to the fluid compressibility assumption. In addition, SIMPLE coupled algorithm [13, 18, 19] is adopted for pressure–velocity coupling and upwind second order method is used for discretizing the governing equations.

4 Grid independence study

Grid independence study is conducted for different cells by considering C_L and C_D as parameters. Four different grids with cell numbers of 15,144, 29,571, 52,170 and 77,321 are generated to evaluate grid independency which all the results are compared to each other. In order to calculate lift and drag coefficients, $Ma = 1$ and $\alpha = 20^\circ$ are used which is indicated in Table 1. There is a negligible difference between the results of the smallest grid and the grid with 52,170 cells. Thus, in order to save computation time and gain better accuracy, the grid with 52,170 cells are adopted to compute the results. Figures 4a, b and 5a, b show the lift and drag coefficients for $\alpha = 16^\circ$ and 20° , respectively. Airfoil is considered as biconvex with a thickness of 0.07 which its maximum thickness is 0.4. The distance of the nearest node from the

Fig. 3 A closer view of the grid generation

Table 1 Grid independence study at $\alpha=20^\circ$ and $Ma=1$

Grid	Number of cells	Growth factor	Height of the first cell	Y^+	C_L	C_D
#1	15,144	1.1	5×10^{-4}	7.52	1.06	0.37
#2	29,571	1.1	1×10^{-4}	4.38	1.22	0.49
#3	52,170	1.1	1×10^{-5}	0.86	1.32	0.61
#4	77,321	1.1	2×10^{-6}	0.51	1.324	0.614

Fig. 4 Grid independence study at $\alpha = 16^\circ$ and $Ma = 1$ for **a** lift coefficient and **b** drag coefficient **Fig. 5** Grid independence study at $\alpha = 20^\circ$ and $Ma = 1$ for **a** lift coefficient and **b** drag coefficient

Fig. 6 Comparison between the pressure coefficient (C_p) with the results of Tulita et al. [20] at $\alpha=0^\circ$

airfoil surface is 1×10^{-5} which certifies that the near wall Y^+ is kept less than 1 as shown in Table 1.

5 Validation

The changes of pressure coefficient (C_p) at $\alpha=0^\circ$ in terms of the airfoil chord length are compared with the results of Tulita et al. [20], which numerically studied the flow control techniques on a biconvex airfoil for verifying the accuracy of the computational results (Fig. 6). The results are in good agreement with the results of Tulita et al. [20], suggesting the accuracy of the computational results.

6 Results and discussion

6.1 Changes of lift and drag coefficients

Figure 7a depicts the changes of lift coefficient (C_L) based on the angle of attack (α). In this figure, three different $Ma = 1$, $Ma = 1.2$ and $Ma = 1.4$ are compared with each other. The lift coefficient enhances with the increment of the angle of attack so that the increase of lift coefficient for the Mach number of 1.4 is greater than the Mach number of 1 and 1.2, which this increment is more significant at $\alpha=22^\circ$. The value of the lift coefficient increases by increasing the Mach number. At angles of attack less than 4° , the values of lift coefficient are almost the same for all three Mach numbers so that the difference of these values is about 5%. As it is obvious, the stall occurs at $\alpha=22^\circ$ at a $Ma=1$ and also stall occurs at $Ma=1.2$ and $Ma=1.4$ at $\alpha=28^\circ$ and $\alpha=30^\circ$, respectively. It is important to note that the stall occurs at higher angles of attack by increasing the Mach number.

Also, Fig. 7b demonstrates the changes of drag coefficient (C_D) based on the angle of attack for three $Ma = 1$, $Ma = 1.2$ and $Ma = 1.4$. Similar to the lift coefficient, by increasing the angle of attack, the drag coefficient has also increased. The value of the drag coefficient decreases by increasing the Mach number. The lowest amount of drag coefficient has obtained at $Ma=1.4$. At $\alpha=32^\circ$ which the highest amount of drag coefficient is obtained, the drag coefficient at $Ma=1$ is about 8% and 30% higher than $Ma=1.2$ and $Ma=1.4$, respectively.

Fig. 7 Changes of **a** lift coefficient and **b** drag coefficient for three different Mach numbers

Fig. 8 Changes of lift-to-drag ratio based on the angle of attack for three different Mach numbers

6.2 Changes of the lift-to-drag ratio

Figure 8 indicates the changes of lift-to-drag ratio (C_L/C_D) based on the angle of attack (α). In this figure, three different $Ma = 1$, $Ma = 1.2$ and $Ma = 1.4$ are compared with each other. As it is obvious, the lift-to-drag ratio is greater at lower angles of attack and with the increase of angle of attack, this ratio decreases with a large slope. The value of C_L/C_D at $Ma = 1$ is less than two other Mach numbers. Moreover, the lift-to-drag ratio decreases with the same slope in all three Mach numbers at angles of attack from 14° to 26° .

6.3 Changes of pressure coefficient (C_p) around a biconvex airfoil

The pressure coefficient (C_p) for the three Mach numbers of 1, 1.2 and 1.4 at $\alpha = 12^\circ$ and 20° is compared with each other (Fig. 9). By considering these cases, it can be seen that the pressure coefficient increases on the upper and lower surfaces of the biconvex airfoil by increasing Mach number, so that these changes are less with the increment of Mach number. The pressure coefficient can be written as [21]:

$$C_p = \frac{p - p_\infty}{\frac{1}{2} \rho_\infty u_\infty^2} \quad (7)$$

where P_∞ is the free flow pressure, ρ_∞ is the fluid density of free flow, u_∞ is the free flow velocity and P is the point

Fig. 10 Velocity contours at $Ma = 1$

Fig. 9 Changes of pressure coefficient at **a** $\alpha = 12^\circ$ and **b** $\alpha = 20^\circ$ for three different Mach numbers

Fig. 11 Velocity contours at Ma = 1.2

Fig. 14 Pressure contours at Ma = 1.2

Fig. 12 Velocity contours at Ma = 1.4

Fig. 15 Pressure contours at Ma = 1.4

Fig. 13 Pressure contours at Ma = 1

pressure where the pressure coefficient is taken into account.

6.4 Velocity and pressure contours around a biconvex airfoil

Figures 10, 11 and 12 shows the velocity contours around a biconvex airfoil at $\alpha = 26^\circ$ for three Mach numbers of 1, 1.2 and 1.4. As it can be seen, a deep stall occurs at this angle when the Mach number is equal to 1. Increasing of Mach number causes to delay the flow separation. In Fig. 10, flow separation occurs near the trailing edge of the airfoil at 0.84 of the airfoil chord length at a Mach number of 1.2. Finally, flow separation occurs near the leading edge of the airfoil at 0.96 of the airfoil chord length at a Mach number of 1.4.

Moreover, Figs. 13, 14 and 15 demonstrate the pressure contours around a biconvex airfoil at $\alpha = 26^\circ$ for three Mach numbers of 1, 1.2 and 1.4. As it can be seen, the pressure on the upper surface of the airfoil has its lowest value and it has a maximum value on the lower surface of the airfoil and the maximum pressure observes near the leading edge at the lower surface.

7 Conclusions

This study mainly focused on the numerical investigation of the turbulent flow over a biconvex airfoil at compressible and high Mach numbers flow using CFD technique. The flow is considered as turbulent, two-dimensional, steady and compressible. In addition, three Reynolds number of 2.4×10^7 , 2.9×10^7 and 3.3×10^7 are considered. The steady RANS equations are solved with the SST $k-\omega$ turbulence model for simulating the turbulence. The results indicated that the lift coefficient enhanced with the increment of angle of attack so that the increase of lift coefficient for the $Ma = 1.4$ is greater than the $Ma = 1$ and $Ma = 1.2$. The drag coefficient has also increased with the increment of the angle of attack, and as the Mach number increased at the same angles of attack, the value of the drag coefficient decreased. By considering the value of the lift-to-drag ratio, it was concluded that the value of lift-to-drag ratio decreased with the same slope in all three Mach numbers. The stall occurred at $\alpha = 22^\circ$, 28° and 30° for $Ma = 1$, 1.2 and 1.4 , respectively. It is clear that stall occurred at higher angles of attack by increasing Mach number.

Compliance with ethical standards

Conflict of interest No conflict of interest was declared by the author.

References

- Hodson JD, Christopherson AP, Deaton JD, Pankonien AM, Reich GW, Beran PS (2019) Aeroelastic topology optimization of a morphing airfoil in supersonic flow using evolutionary design. AIAA Scitech 2019 Forum
- Hari N, Schetz JA, Kapania RK (2019) Numerical prediction of interference drag of a strut-surface intersection in supersonic flow. AIAA Scitech 2019 Forum
- Manshadi MD, Aghajanian S (2018) Computational aerodynamic optimization of wing-design concept at supersonic conditions by means of the response surface method. *J Braz Soc Mech Sci Eng* 40(5):254–265
- Askari S, Shojaeefard MH, Goudarzi K (2011) Numerical and analytical solution of compressible flow over double wedge and biconvex airfoils. *Eng Comput* 28(4):441–471
- Keith TG, Cioc S, Jiang H (2019) Spreadsheet computations of a symmetric double wedge airfoil in supersonic flow. *Int J Mech Eng Educ* 0(0):1–16
- Tharkude TS, Li ZH (2018) Simulation study of supersonic natural laminar flow on wing with biconvex airfoil. *IJERT* 7(3):1–8
- Ovsyannikov MP (1976) Allowance for gas blowing in supersonic flow over a wedge. *J Eng Phys Thermophys* 30(6):663–668
- Raghunathan S, Gillan MA, Cooper RK, Mitchell RD, Cole JS (1999) Shock oscillations on biconvex aerofoils. *Aerosp Sci Technol* 3(1):1–9
- Al-Garni AZ, Saeed F, Al-Garni AM (2008) Experimental and numerical investigation of 65-degree Delta and 65/40-degree double-delta wings. *J Aircr* 45(1):71–76
- Hamid MA, Hasan AT, Alimuzzaman SM, Matsuo S, Setoguchi T (2014) Compressible flow characteristics around a biconvex arc airfoil in a channel. *Propul Power Res* 3(1):29–40
- Rahman MM, Hasan AT, Islam AS, Matsuo S, Setoguchi T (2015) Computation of transonic internal flow around a biconvex airfoil with cavity. *J Mech Sci Technol* 29(6):2415–2421
- Rahman MM, Hasan T, Rabbi MS (2017) Effects of cavity size on the control of transonic internal flow around a biconvex circular arc airfoil. AIP conference proceedings, 1851(1). AIP Publishing
- Fatahian E, Nichkoohi AL, Fatahian H (2019) Numerical study of the effect of suction at a compressible and high Reynolds number flow to control the flow separation over Naca 2415 airfoil. *Prog Comput Fluid Dyn* 19(3):170–179
- Menter FR (1994) Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA J* 32(8):1598–1605
- Catalano P, Amato M (2003) An evaluation of RANS turbulence modeling for aerodynamic applications. *Aerosp Sci Technol* 7(7):493–509
- Menter FR, Kuntz M, Langtry R (2003) Ten years of industrial experience with the SST turbulence model. *Turbul Heat Mass Transf* 4(1):625–632
- Fatahian H, Salariyan H, Eshagh Nimvari M, Fatahian E (2018) Numerical study of suction and blowing approaches to control flow over a compressor cascade in turbulent flow regime. *Int J Autom Mech Eng* 15(2):5326–5346
- Genç MS, Kaynak U, Yapici H (2011) Performance of transition model for predicting low Re aerofoil flows without/with single and simultaneous blowing and suction. *Eur J Mech B Fluids* 30(2):218–235
- Monir HE, Tadjfar M, Bakhtian A (2014) Tangential synthetic jets for separation control. *J Fluids Struct* 45(1):50–65
- Tulita C, Raghunathan S, Benard E (2004) Drag reduction and buffeting alleviation in transonic periodic flow over biconvex aerofoils. In: 24th international congress of the aeronautical sciences, pp 1–13
- Vuddagiri A, Halder P, Samad A, Chaudhuri A (2016) Flow analysis of airfoil having different cavities on its suction surface. *Prog Comput Fluid Dyn Int J* 16(2):67–77

Publisher's Note Springer Nature remains neutral with regard to jurisdictional claims in published maps and institutional affiliations.