Available online at www.elixirpublishers.com (Elixir International Journal)

Mechanical Engineering

Elixir Mech. Engg. 53 (2012) 11695-11698

ISSN: 2229-712X

Analysis of Bi-Convex Aerofoil using CFD Software at Supersonic and Hypersonic Speed

S. S. Benadict Bensiger[1] and N. Prasanth[2]

[1]Department of Aeronautical Engineering, Lord Jegannath College of Engineering & Technology, PSN Nagar, Ramanathichanputhur, Kumarapuramthoppur, Kanyakumari District-629402.

[2]Anna University Chennai, India.

ARTICLE INFO

Article history:
Received: 27 March 2012;
Received in revised form:
19 November 2012;
Accepted: 28 November 2012;

Keywords
Bi-convex aerofoil,
Symmetric,
Pressure coefficient,
Lift coefficient,
Drag coefficient.

ABSTRACT

The aim of this paper is to design and analyse a biconvex aerofoil with slightly curved leading edge and estimating the coefficient of pressure (C_P), coefficient of lift (C_L) and coefficient of drag (C_D) at supersonic and hypersonic speeds. The aerofoil is symmetric and has a thickness of 6%. Gambit and Ansys Fluent are two best CFD software used for the analysis. The aerofoil was designed and meshed using Gambit software. Good quality boundary layer mesh can be easily generated using Gambit. The mesh generated was analyzed using Ansys Fluent software. About 1000 iterations were used for the simulation purpose. The pressure coefficient, lift coefficient and drag coefficient for the designed aerofoil were obtained. It was proven that the biconvex aerofoil can also be used for hypersonic speeds.

Introduction

Computational Fluid Dynamics (CFD) is the science of predicting fluid flow, heat transfer, mass transfer, chemical reactions, and related phenomena by solving the mathematical equations which govern these processes using a numerical process. The result of CFD analyses is relevant engineering data used in conceptual studies of new designs, detailed product development, troubleshooting, redesign etc. CFD analysis complements testing and experimentation reduces the total effort required in the laboratory. It provides a qualitative (and sometimes even quantitative) prediction of fluid flows by means of mathematical modeling (partial differential equations), numerical methods (discretization and solution techniques), and software tools (solvers, pre- and postprocessing utilities). CFD enables scientists and engineers to perform 'numerical experiments' (i.e. computer simulations) in a 'virtual flow laboratory'. The broad area of CFD leads to many different specialized technology areas. These include grid generation, flowfield discretization algorithms, efficient solution of large systems of equations, massive data storage and transmission technology methods and computational flow visualization.

Tulia et al [1], using CFD analyses, the effects of flow control techniques, as the contour bump and the surface cooling concepts, are separately investigated in transonic periodic flow over 14% and 18% biconvex aerofoils. D. H. Williams et al [2] tested 5% thick bi-convex aerofoil using Compressed Air Tunnel and estimated the values of C_L and C_D. W. P. Jones and Sylvia W. Skan [3] developed a method for the calculation of the aerodynamic forces on an oscillating aerofoil. Aerodynamic lift and pitching moment derivatives for a 5 per cent thick, symmetrical, circular-arc aerofoil at Mach numbers M = 1.4, 1.5 and 2.0 are given for a range of frequencies and compared with values obtained on the basis of the flat plate theory. The effect of thickness appears to be important at the lower values of M, and the results indicate that the flat plate theory is not sufficiently accurate. N. Gregory et al [4] estimated the aerodynamic characteristics of NACA 0012. Robert J. McGhee et al [5] conducted an investigation in the Langley low-turbulence pressure tunnel to determine the low-speed two-dimensional aerodynamic characteristics of a 17% thick airfoil designed for general aviation applications. Here we designed a biconvex aerofoil having 6% thickness. The profile was designed and imported into the Gambit software meshed and the generated mesh was imported into the Ansys Fluent software. The pressure coefficient, lift coefficient and drag coefficient for the designed aerofoil were obtained using the simulation process.

Design and Modeling

An aerofoil (or airfoil) is a two-dimensional object, the shape of the cross section of the wing at right angles to the wing span, with the function of producing a controllable aerodynamic force by its motion through the atmosphere. The bi-convex aerofoil is mostly symmetric and used in supersonic speeds. The aerofoil has a thickness of 6%. For unit chord the profile was designed. The profile was tabulated in table. 1.

Meshing

The meshing of the aerofoil was done using gambit software. Boundary meshing should be used for the flow analysis. The mesh was generated for the aerofoil with unit chord. In an external flow such as that over an airfoil, we defined a farfield boundary and mesh the region between the airfoil geometry and the farfield boundary. It is a good idea to place the farfield boundary well away from the airfoil since we used the ambient conditions to define the boundary conditions at the farfield. The maximum aspect ratio in this mesh is quite high. This is acceptable because these cells are close to the airfoil wall surfaces. This is needed for the turbulence model

Tele:
E-mail addresses: sanbenadict@gmail.com

being used, since it ensures the first grid point is in the viscous sublayer. The less effect it has on the flow and so more accurate is the farfield boundary condition. The generated mesh was show in fig. 1.

Table 1: Aerofoil profile

X	Y	Z
1	0	0
0.95	0.00707	0
0.9	0.01259	0
0.8	0.02099	0
0.7	0.02635	0
0.6	0.02917	0
0.5	0.03	0
0.4	0.02927	0
0.3	0.02709	0
0.2	0.02332	0
0.15	0.02067	0
0.1	0.01729	0
0.075	0.01516	0
0.05	0.01255	0
0.025	0.00903	0
0.0125	0.00646	0
0	0	0
0.0125	-0.00646	0
0.025	-0.00903	0
0.05	-0.01255	0
0.075	-0.01516	0
0.1	-0.01729	0
0.15	-0.02067	0
0.2	-0.02332	0
0.3	-0.02709	0
0.4	-0.02927	0
0.5	-0.03	0
0.6	-0.02917	0
0.7	-0.02635	0
0.8	-0.02099	0
0.9	-0.01259	0
0.95	-0.00707	0

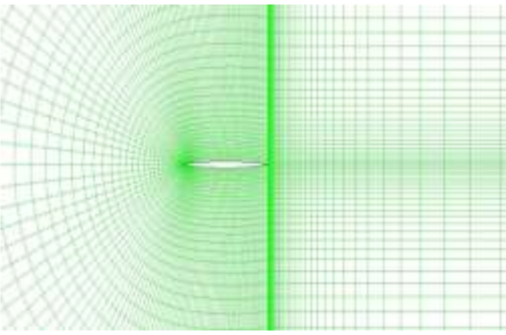

Fig. 1. Aerofoil Mesh

Simulation

The meshed aerofoil was imported into Ansys Fluent and simulated. The mesh scale should be in metre. The mesh was checked for errors. The solver used is steady state density based solver. Energy equation was used for the simulation. This is needed because the flow is compressible and we used ideal gas equation. The one-equation Spalart-Allmaras model was chosen. This is a relatively simple turbulence model that has been shown to give good results for boundary layers subjected to adverse pressure gradients, particularly where there is no or only mild separation. Setting zero operating pressure which means that all pressures set in FLUENT will be absolute. The pressure-far-field boundary type is applicable only when the density is calculated using the ideal-gas law. It is important to place the far-field boundary far enough from the object of interest. The pressure, angle of attack and the temperature were given as input corresponding to the Mach number and altitude.

The solution method used is implicit formulation and Roe-FDS flux type. The implicit formulation is more stable and can be driven much harder to reach a converged solution in less time. The Green-Gauss Node based gradient method is used. This is slightly more computationally expensive than the other methods but is more accurate. Second Order Upwind for flow and turbulence discretization was selected for the simulation. The Second Order Upwind schemes were used to accurately predict drag. The Courant number (CFL) determines the internal time step and affects the solution speed and stability. The CFL for the density-based implicit formulation is 5.0. It is often possible to increase the CFL to 10, 20, 100, or even higher, depending on the complexity of problem. A lower CFL is required during startup (when changes in the solution are highly nonlinear), but it can be increased as the solution progresses. Then the inputs were initialized and the simulation was done. The static pressure contours for Mach 2, 3, 4, 5, 6 & 7 are shown in fig 2, 3, 4, 5, 6 & 7 respectively. Also the pressure coefficient distribution for Mach 2, 3, 4, 5, 6 & 7 are shown in fig 8, 9, 10, 11, 12 & 13 respectively.

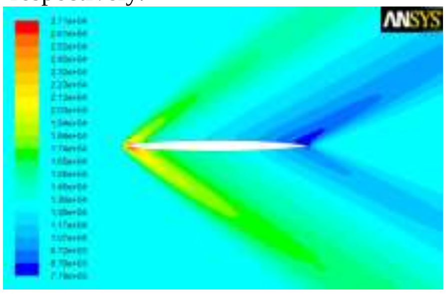

Fig. 2: Static pressure at Mach 2

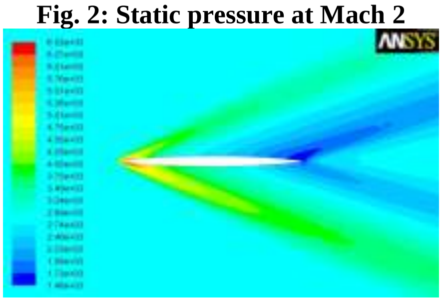

Fig. 3: Static pressure at Mach 3

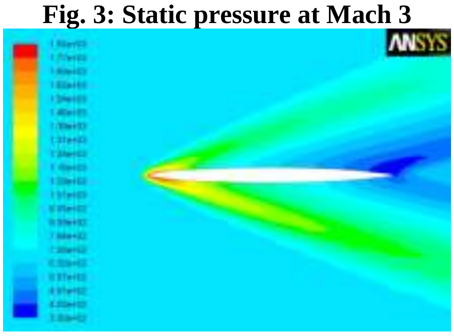

Fig. 4: Static pressure at Mach 4

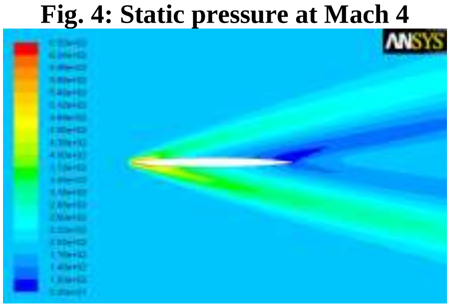

Fig. 5: Static pressure at Mach 5

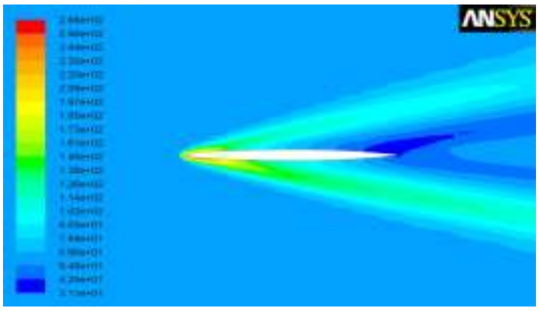

Fig. 6: Static pressure at Mach 6

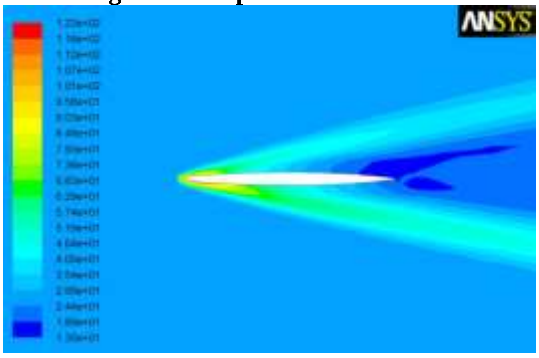

Fig. 7: Static pressure at Mach 7

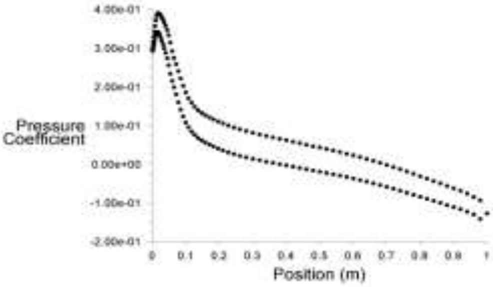

Fig. 8: Pressure coefficient distribution at Mach 2

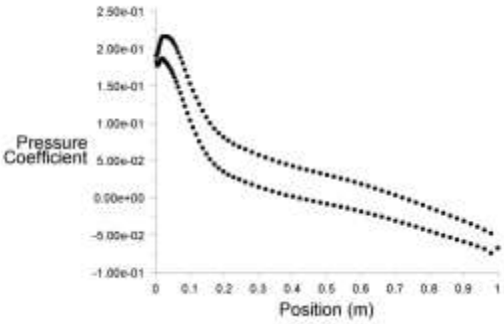

Fig. 9: Pressure coefficient distribution at Mach 3

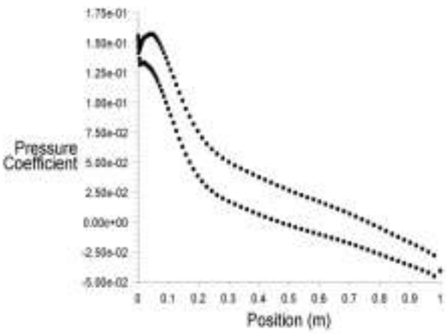

Fig. 10: Pressure coefficient distribution at Mach 4

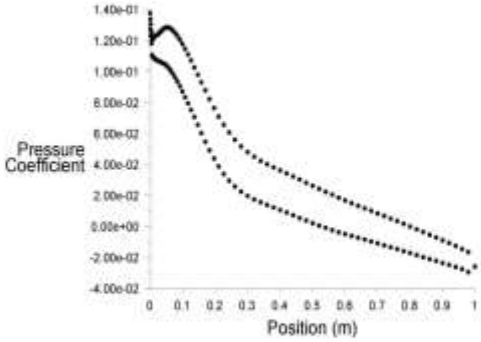

Fig. 11: Pressure coefficient distribution at Mach 5

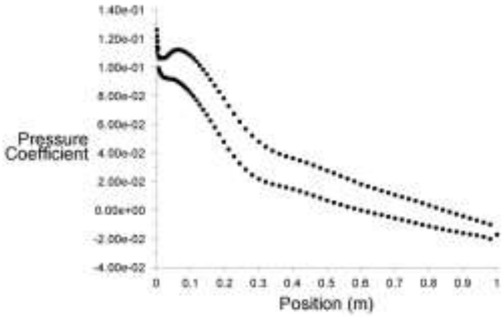

Fig. 12: Pressure coefficient distribution at Mach 6

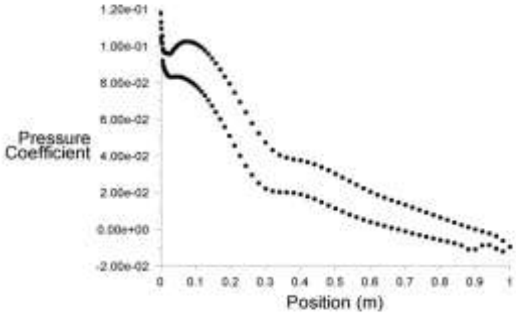

Fig. 13: Pressure coefficient distribution at Mach 7

Conclusion

The simulation process was done using Ansys Fluent and the coefficient of lift and coefficient of drag were found out. About 1000 iterations were used to calculate the lift and drag coefficients. These values were found separately for different Mach numbers. Coefficient of lift and the coefficient of drag for corresponding Mach number were tabulated in table 2. The maximum and minimum values for the pressure coefficients were also estimated and the values were tabulated in table 3. From the result obtained it shows that the lift coefficient decreases as the Mach number increases and the coefficient of drag also decreases. This phenomenon shows that the increasing the velocity (or over speeding) may bring the shock waves less effect. Thus the wave drag is reduced. From this it shows that the bi-convex aerofoil can be used in supersonic speeds as well as hypersonic speeds.

Table 2: Coefficient of lift and coefficient of drag for supersonic and hypersonic speeds

Mach Number	Coefficient of lift, C_L	Coefficient of drag, C_D
2	0.061031	0.025150
3	0.038115	0.018415
4	0.028740	0.015385
5	0.023669	0.013868
6	0.020610	0.013106
7	0.018449	0.012748

Table 3: Maximum and minimum Coefficient of pressure for supersonic and hypersonic speeds

Mach number	Minimum coefficient of pressure, C_P	Maximum coefficient of pressure, C_P
2	-0.14229	0.390354
3	-0.07367	0.21653
4	-0.0452	0.15743
5	0.029721	0.137601
6	-0.02047	0.126022
7	-0.01314	0.11773

Reference

[1]. C. Tulita, S. Raghunathan, E. Benard, Drag Reduction and Buffeting Alleviation in Transonic Periodic Flow over Biconvex Aerofoils, 24th International Congress of the Aeronautical Sciences, pp. 1-13, 2004.

[2]. D. H. Williams, B.Sc. and A. H. Bell, Tests on a 5-percent Biconvex Aerofoil in the Compressed Air Tunnel, Aeronautical Research Council Reports and Memoranda, 1950.

[3]. W. P. Jones, M. A. and Sylvia W. Skan, Aerodynamic Forces on Biconvex Aerofoils Oscillating in a Supersonic Airstream, Aeronautical Research Council Reports and Memoranda, 1953.

[4]. N. Gregory and C. L. O'Reilly, Low-Speed Aerodynamic Characteristics of Naca 0012 Aerofoil Section, including the Effects of Upper-Surface Roughness Simulating Hoar Frost, Aeronautical Research Council Reports and Memoranda, 1973.

[5]. Robert J. Mcghee and William D. Beasley, Low-Speed Aerodynamic Characteristics of a 17 -Percent Thick Airfoil Section Designed for General Aviation Applications, National Technical Information Service, 1973.