Surman Journal for Science and Technology
Vol.02, No.3, Dec 2020, pp. 127 ~ 138
sjst.scst.edu.ly
مجلة صرمان للعلوم والتقنية

Comparison Result of the Case of Compressible Inviscid Flow Problem.

Dr. Hasan Taher M.elkamel [1, a] , Dr. Ir. Bambang Basuno [2, b], Mohammed Ahmed Alazhari [3, c], Elsalhin mispah Mohamed[4,d]

[1, 2,] Faculty of Mechanical and Manufacturing Engineering,Tun Hussein Onn University of Malaysia (UTHM), Parit Raja, Batu Pahat, 86400 Johor, Malaysia

[3] Department of Mechanical Engineering, Higher institution of sciences and technical, Algariat Libya

[4]College of engineering technology – Surman, Libya

[a]hassan.elkamel@gmail.com, [b] Bambangb@uthm.edu.my, [c]alazhariklill@gmail.com, [d] [d]slhancora2@gmail.com

Abstract.

The present work presents a study of Two Dimensional Flow Analysis over NACA0012 and RAE2822 airfoils. here the aerodynamics analyses are carried out by use of Cell Centered Finite Volume scheme. Through this study, a grid generation algorithm has been developed to fulfill the need of the case of two dimensional C-topolgy. There are types of grid generator can be adopted: structured grid or unstructured grid. The first grid approach is normally needs less expenses compared to the second one. Using an ordinary approach in creating the grid of the flow domain by algebraic grid generation method

The validation of the developed computer code carried out by comparing its result with Fluent software and experimental results, it had been found that some discrepancies result in term of Mach number or other flow properties between the developed computer code and fluent software are apparently. By improving grid of the flow domain by use of smoothing technique give the developed computer produce the result in a good agreement with the fluent software and the experimental result as well. Considering the ability of the developed computer code similar to the Fluent software, the present code had been used to evaluate the aerodynamics characteristics for other cases such as fuselages.

Keywords: Airfoil, CFD, Inviscid Flow Analysis

1. Introduction

The present work which deals on the development computer code for solving the Time Averaged Navier Stokes, it used firstly for solving the two dimensional compressible inviscid flow problem governed by Euler equation before goes to the flow problem governed by the Time Averaged Navier Stokes. For the purpose of evaluation on the capability of the developed computer code, the present work focused on solving the flow problem past through airfoil NACA 0012 and airfoil RAE 2822. Various researchers around the world had obtained the experiment result over these two types of airfoils at various flow conditions. Hence, comparison result can be made between the present developed code with the experimental result beside on the comparison result obtained by using Fluent software. The comparison result of the present developed code as inviscid flow solver presented in the following:

2. Governing Equations of Inviscid Two Dimensional Compressible Flow.

The governing equation of fluid flow without viscous effect is known as Euler equation. For the case of flow passes over a body, the Euler equations which can be derived from the conservation of law are written in term of the conservative variables can be given as [3,4]:

$$\partial/_{\partial t} \int_V QdV + \int_S \left\lfloor (E_e)_{nx} + (F_e)_{ny} \right\rfloor ds = 0$$
(1)

with Q written to a Cartesian system, V is the cell volume, *nx* and *ny* are the components of the normal unity vector to the flux face, S is the flux area and E_e and F_e are the components of the convective flux vector. The vectors Q, E_e and F_e are represented by:

$$Q = \begin{Bmatrix} \rho \\ \rho u \\ \rho v \\ e \end{Bmatrix}, \quad E_e = \begin{Bmatrix} \rho u \\ \rho u^2 + p \\ \rho uv \\ (e+p)u \end{Bmatrix} \quad and \quad F_e = \begin{Bmatrix} \rho v \\ \rho uv \\ \rho v^2 + p \\ (e+p)v \end{Bmatrix}$$
(2)

Where ρ the fluid density; u and v the Cartesian components of the velocity vector in the x and y directions, respectively; e the total energy per unit volume of the fluid; and p is the static pressure of the fluid. The method for solving a system equation as defined by Eq. (1) can be done by use of method available in Ref. [1] or [6].

3. Mesh Flow Domain

However, the case of flow past through an airfoil in which the flow domain has relatively a simple flow domain, the unstructured grid can be developed by use of structured grid. This is due to the fact that a structured is more easily programmed than unstructured grid developed by use a Delaunay grid generator. In this respect, the setting up the flow domain past through airfoil can use c-grid topology or O-grid topology. The present work use unstructured grid in C-grid topology. Figure 3.3 shows an airfoil model immersed in the free stream with the incoming velocity U_∞. with make an angle of α with respect to the airfoil chord line. And for more explanations check the reference [8].

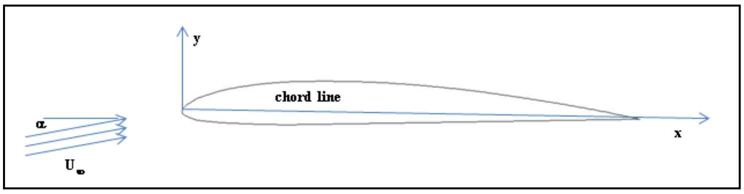


Figure 1.1 Flow past through an airfoil

Figure 1.1 shows the flow domain surrounding the airfoil which modeled to have C-topology as shown in the Figure 1.2 below.

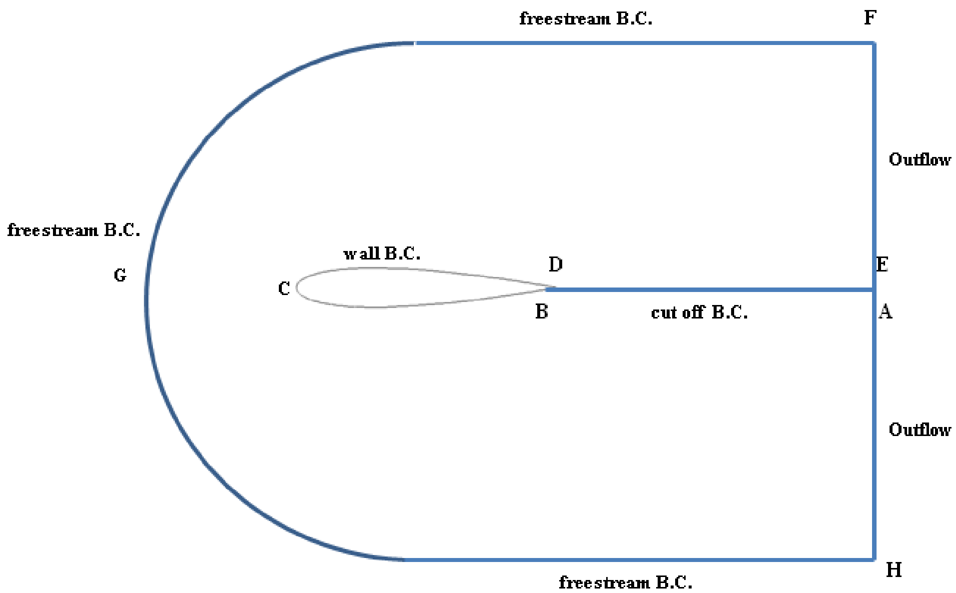
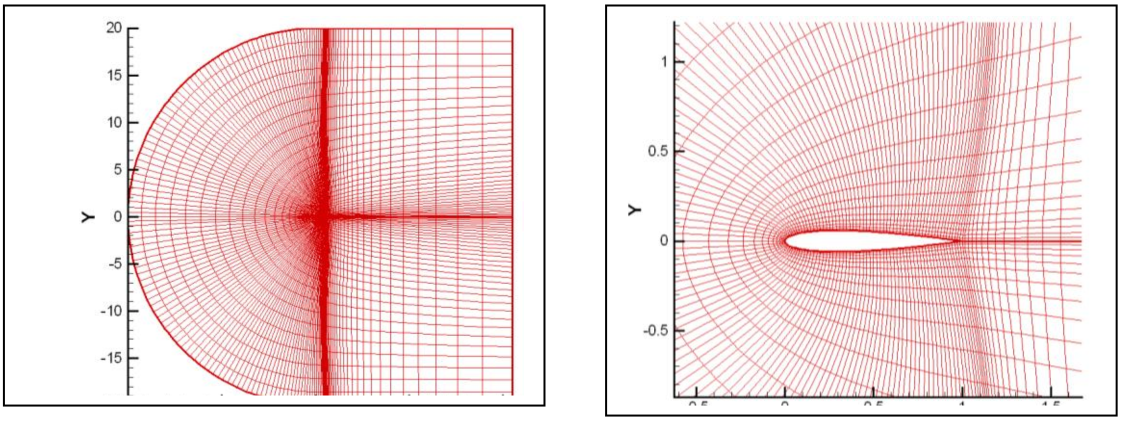


Figure 1.2 The C-topology for the flow past through airfoil and the boundary condition.

Surman Journal for Science and Technology
Vol.02, No.3, Dec 2020, pp. 127 ~ 138

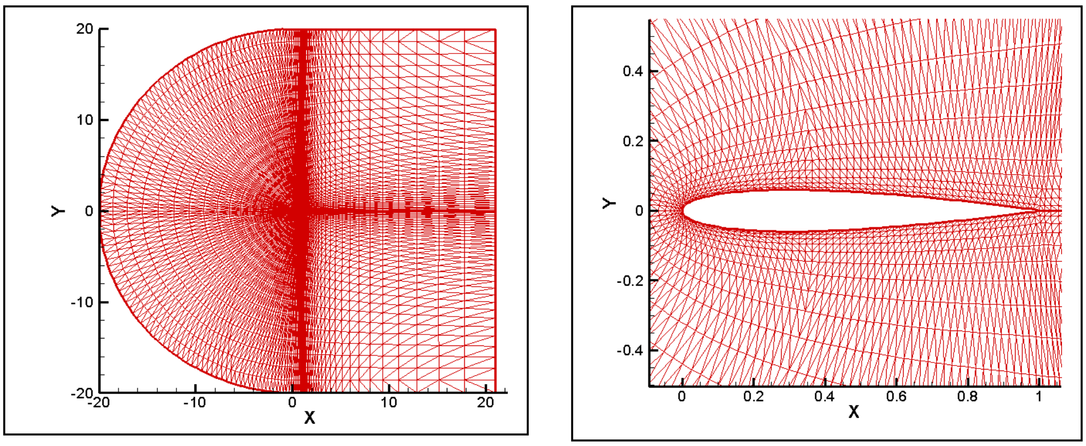
In the C-topology, the line ABCDE represent the inner boundary while the line AHGFE represent the outer boundary. The line AB and DE represent the cut off boundary condition. The outflow boundary condition defined along the line AH and EF, while line HGF represents the free stream boundary condition. To minimize the effect of imposing the outer boundary to the overall solution, the present work uses a distance HF and GE is set 40 times the airfoil chord length. As it had been mention previously the present work use unstructured grid with the Roe's cell center finite volume as the solver for solving the governing equation of fluid motion. The element used in formulating the Finite Volume method is in the triangular form. To obtain a triangular element, the present work start with creating the mesh flow domain in structured form which will generate the mesh of flow domain as shown in the Figure 3.5.



(a) Full view (b) zooming view

Figure 1.3 The mesh flow domain in structured grid representation.

As the structured grid is available, the next step is grid triangulation namely converting from the shape of element from quadrilateral in structured grid become a triangle element. This process will generate the mesh of the flow domain originally as given by Figure 1.3 becomes as shown in the Figure 1.4.



SJST Surman Journal for Science and Technology
Vol.02, No.3, Dec 2020, pp. 127 ~ 138
sjst.scst.edu.ly
مجلة صرمان للعلوم والتقنية

(a) Full view (b) zooming view

Figure 1.4 The mesh flow domain in unstructured grid representation

4. Result and Discussion.

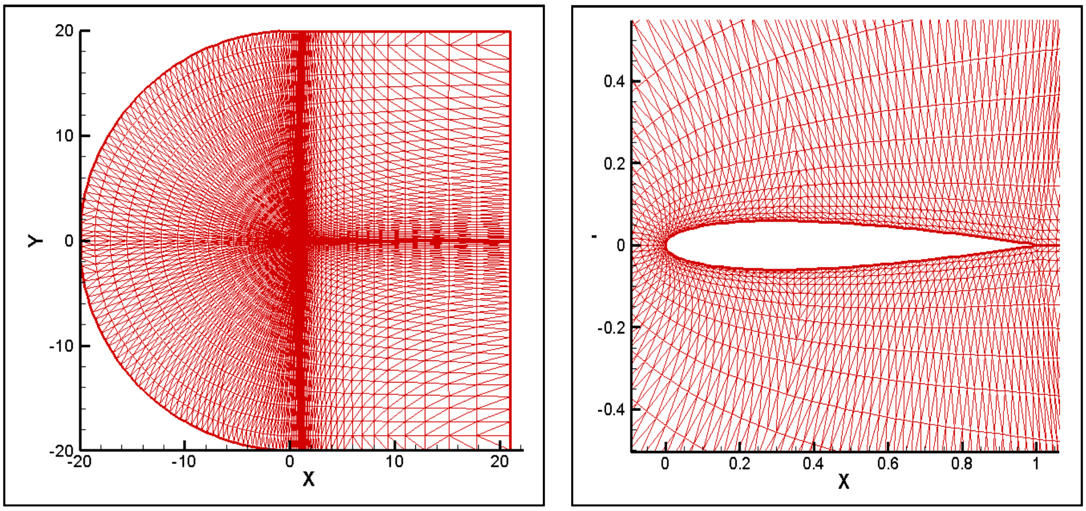
For the purpose of evaluating the present developed computer code in view of the flow problem in hand considered as the flow problem governed by Euler equation are applied to the case of flow problem past through airfoil NACA 0012 and airfoil RAE 2822. The first airfoil represents a symmetrical airfoil. Any solution must provide a symmetrical solution between the solution over the flow domain located at the upper part of airfoil surface with the result provided in the flow domain below the lower surface when the incoming velocity having a zero angle of attack. Table 4.1 show the flow condition had been assigned to these two cases.

Table 4.1 Summary of Inviscid Flow Test Cases

Test Case	Airfoil Profile	M_∞	α	Reference(s)
1	NACA 0012	0.15	0°	Gregory & O'Reilly, NASA R&M 3726, Jan1970
2	NACA 0012	0.15	10°	Gregory & O'Reilly, NASA R&M 3726, Jan1970
3	NACA 0012	0.15	15°	Gregory & O'Reilly, NASA R&M 3726, Jan1970
4	NACA 0012	0.7	1.49°	Christopher L. Rumsey, NASA Langley Research center, May 1988
5	RAE 2822	0.729	2.31°	Cook et al., 1979

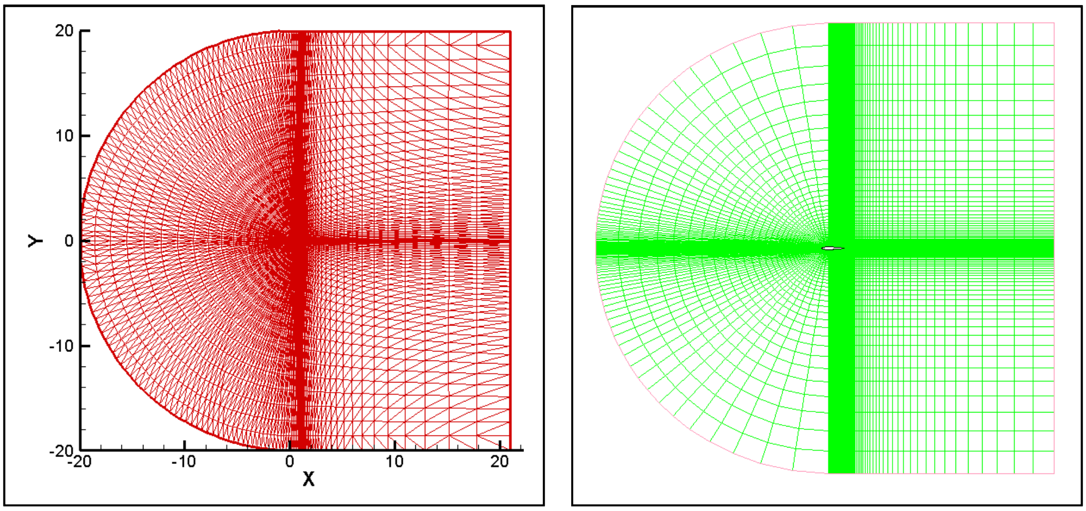
5. Case Study of Flow Past through Airfoil NACA0012 for Inviscid Flow

For the case of flow past through airfoil NACA 0012, two values of Mach number had been selected. The first Mach number value is $M_\infty = 0.15$ and the second one is at the Mach number $M_\infty = 0.7$. The first value Mach number indicates that the flow is incompressible flow, since the physical flow phenomena had been found as far as the Mach number of the free stream below $M = 0.3$, the compressible effect can be ignored. In other word, the flow behaves as incompressible. At this low Mach number the flow analysis are carried out at three different angle of attacks $\alpha = 0^0$, 10^0 and $\alpha = 15^0$. The topology of the flow domain presented in C-topology with unstructured grid as shown in the Figure 2.1. The number of elements are 128 x 64 x 2 elements.



(a) The whole domain Actual (b) Close – up

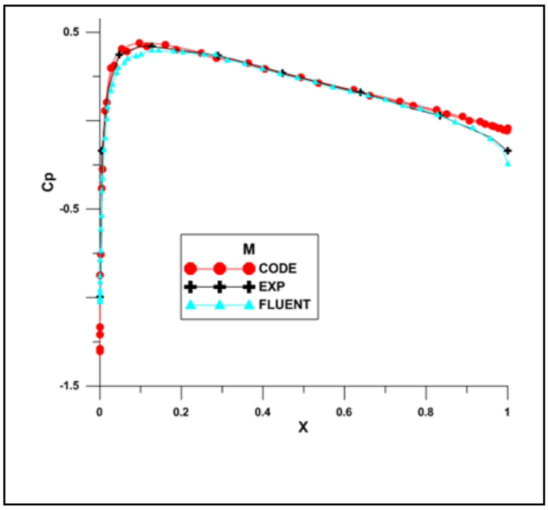
Figure 2.1.a The mesh flow domain over airfoil NACA 0012, mesh size is 128*64*2

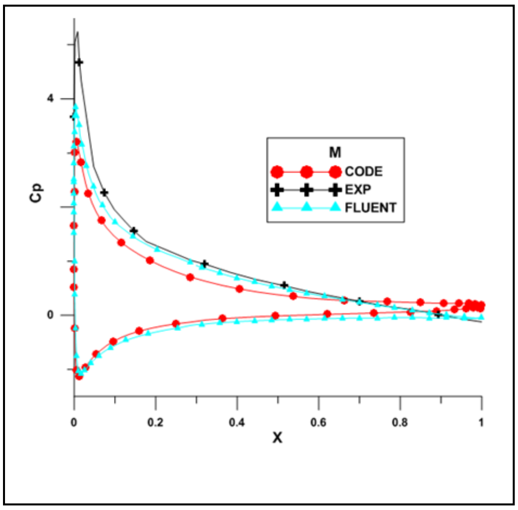


(a) Present grid generation code (b) Grid by Gambit software

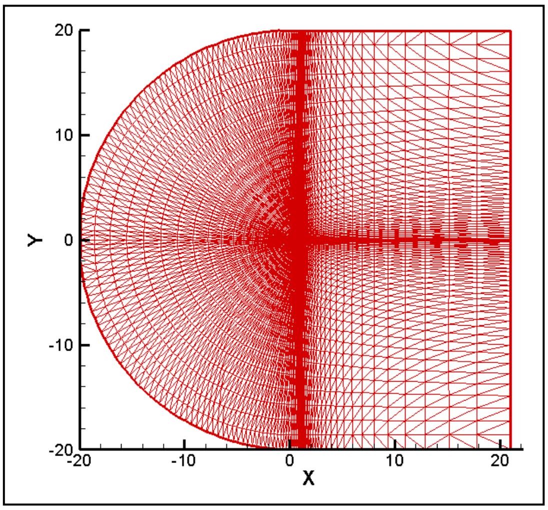
Figure 2.1.b The mesh flow domain over airfoil NACA 0012

Figure 2.2 shows the comparison result in term of pressure coefficient C_p along the airfoil surface between the present code, Fluent software and the experimental result. Here the experimental result used is the experimental provided by (Gregory & O'Reilly, NASA R&M 3726, Jan1970). The flow condition is set at the incoming Mach number velocity $M_\infty = 0.15$ for three different angle of attacks $\alpha = 0^0$, 10.0^0 and $\alpha = 15.0^0$





(a) Angle of Attack $\alpha = 0^0$ (b) Angle of Attack $\alpha = 10^0$

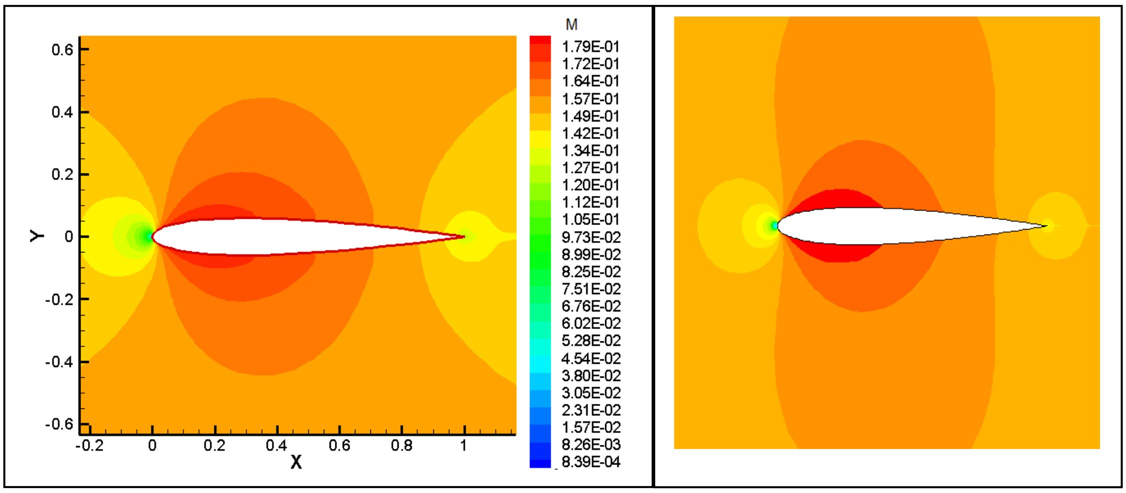


(c) Angle of Attack $\alpha = 0^0$

Figure 2.3 Comparison result of pressure coefficient distribution Cp along the airfoil surfaces for the case Airfoil NACA 0012.

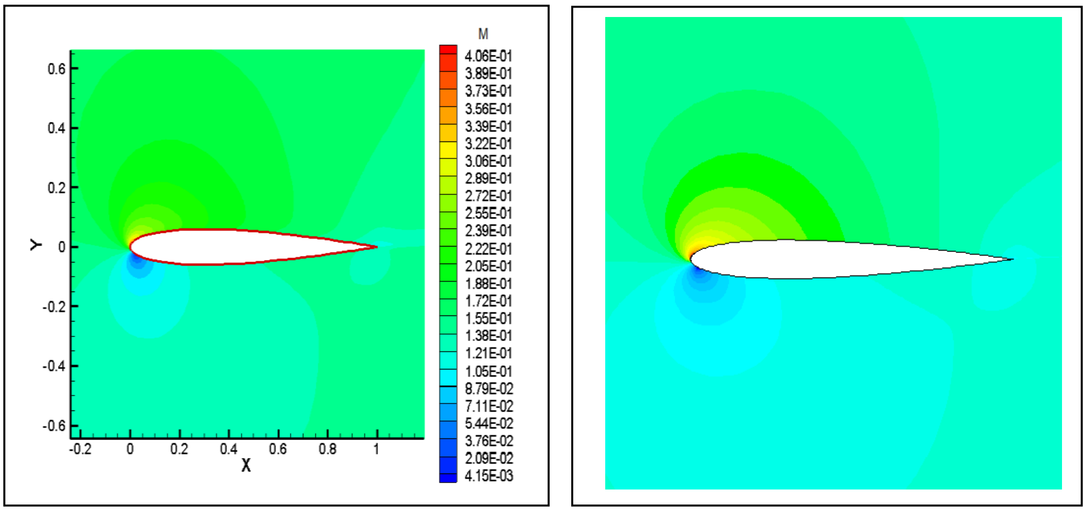
Considering above Figure, one can conclude that the present code is in a good agreement with experimental result as well as with the result obtained by use of Fluent software. The present code is able to produce a symmetrical result between C_p at upper surface and Cp on the lower surface.

In view of the flow pattern over the flow domain presented in term Mach number distribution between the present developed code and Fluent software are depicted in the Figure 2.4 and Figure 2.5 for the case at the angle $\alpha = 0^0,\ 10.0^0$ respectively.



(a) The present code (b) Fluent

Figure 2.3.b: Mach Contour, NACA 0012 airfoil at M_∞=0.15 and α=0° by using the present inviscid code and Fluent software



(a) The present code (b) Fluent

Figure 2.2.b: Mach Contour, NACA 0012 airfoil at M_∞=0.15 and α=10° by using inviscid code and Fluent software

The previous result is for the case of a low Mach number. Physical observation had conducted by various researchers had found that a Mach number M < 0.3, the compressible effects can be ignored and the flow can be treated as incompressible flow. In the case at a high subsonic flow in which the compressible effect has to be taken account, the comparison result between the present inviscid code, Fluent and experimental result in term of pressure coefficient distribution along the airfoil surfaces as shown in the Figure 2.6.

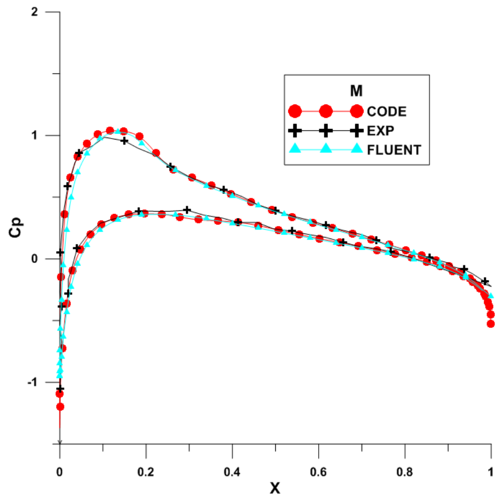
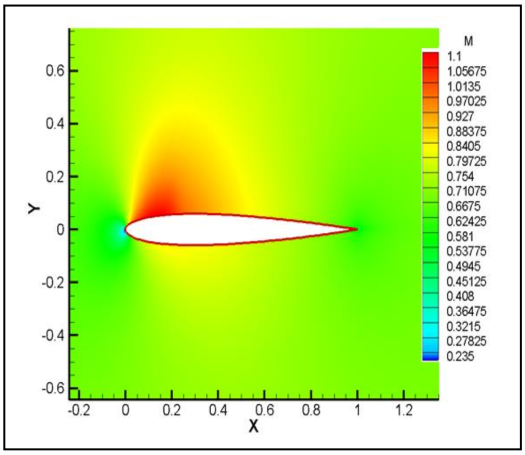
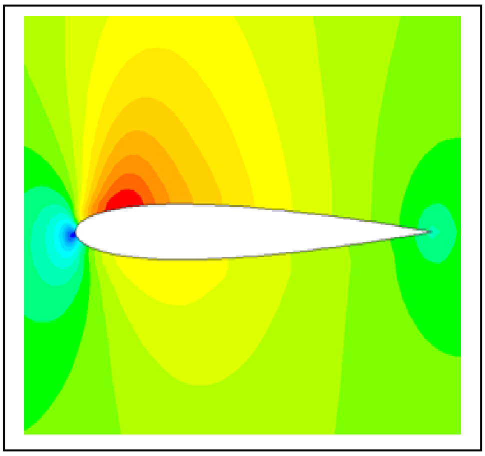


Figure 2.6.: Distribution of Mach Number on NACA0012 at M_∞=0.7 and α = 1.49° by using inviscid code

While Figure 2.7 shows their comparison result in term of the Mach contour over the flow domain between the present code and Fluent software



(a) The present code



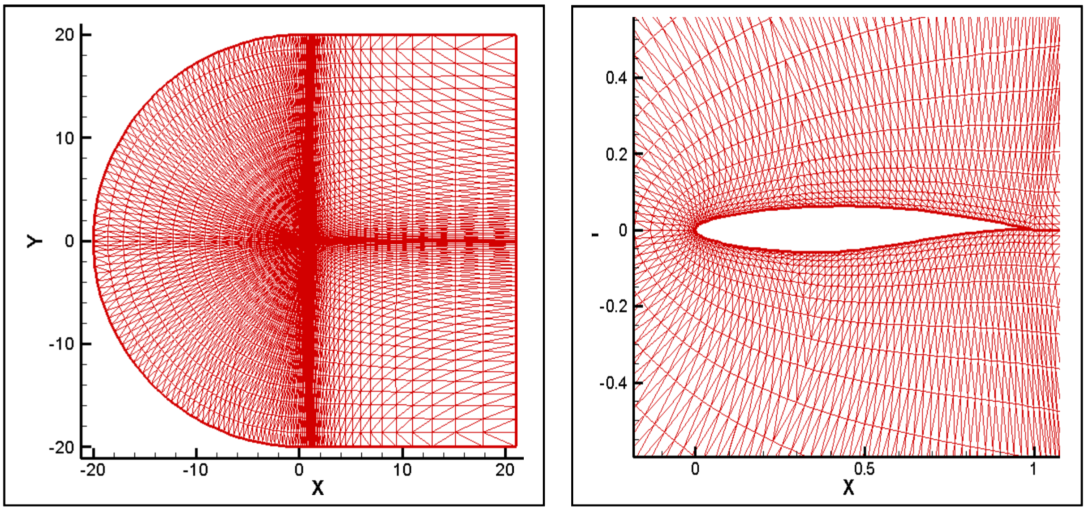
(b) Fluent

Figure 2.7.: Mach Contour, NACA 0012 airfoil at M_∞=0.7 and α=1.49° by using inviscid code and Fluent software

Considering the comparison result as shown in the Figure 2.6 and Figure 2.7 are clearly indicated the present developed code is in a good agreement with the experimental result and also able to produce the result as obtained by Fluent software.

The comparison for other airfoil is applied to the case of flow past through airfoil RAE 2822. The flow condition is set at M=0.729 , α = 2.31° and Re = 6.5 Million. Basically in the inviscid flow analysis, Reynolds number is not needed. However, in the context of the experiment work, the Reynolds number is required to be stated, since the Reynolds number will determine with what happen the flow behavior inside the boundary layer domain. In addition to this, for the case flow past through a streamline body such as the flow past through airfoil, a High Reynolds number will give more guarantee to get the inviscid flow solution close to the experimental results.

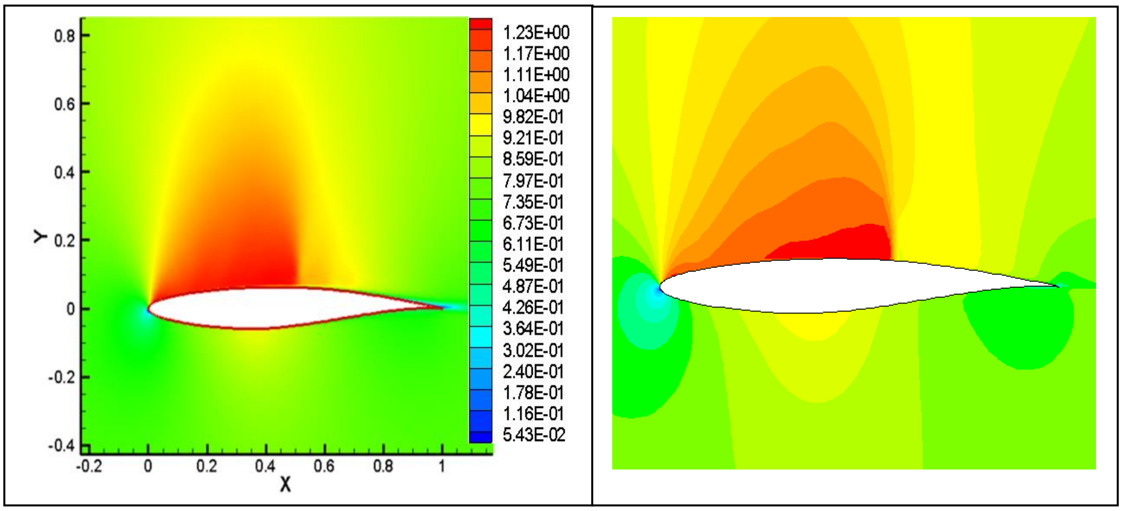
In view of grid topology and mesh flow domain, Figure 2.8 shows the grid topology and the close up of the mesh flow surrounding airfoil had been used in the present work.



(a) The whole domain Actual (b) Close – up

Figure 2.8.a The mesh flow domain over airfoil RAE 2822 with the mesh size is 128*64*2

The comparison of Mach contour between the present work and fluent a s given in the Figure 2.9 below.



(a) The present code (b) Fluent

Figure 2.9.b: Mach Contour, RAE2822 airfoil at M=0.729 and $\alpha = 2.31^{o}$ by using inviscid code and Fluent software

While in the case of pressure coefficient along the airfoil surface their comparison results are given in Figure 2.10.

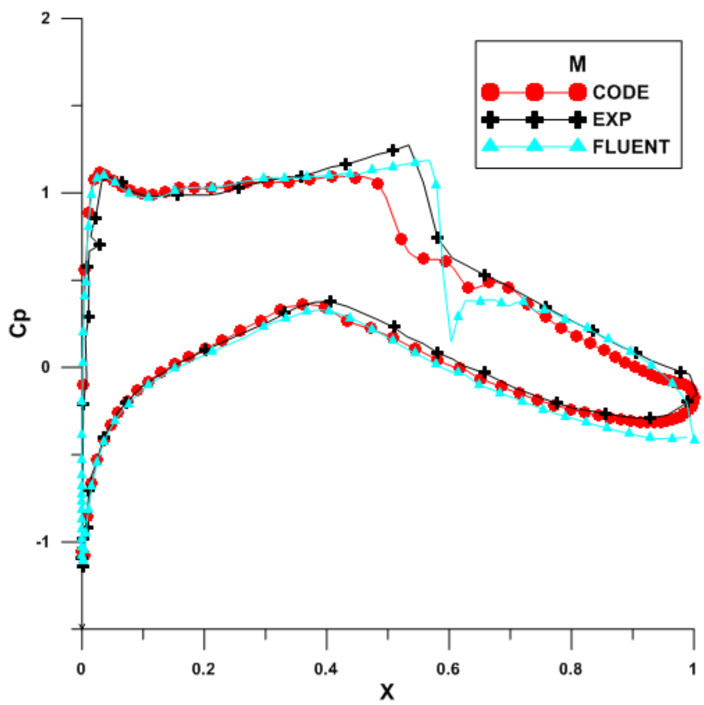


Figure 2.10 Cp Distribution on RAE2822 M=0.729 and α = 2.31° by using inviscid code

6. Conclusion

The validation of the developed computer code is carried out through comparing their result to case of flow past through airfoil where their experimental results are already available in the literature and through rerunning the flow problem by use a Fluent software. Here uses test case of flow past through airfoil NACA 0012 and airfoil RAE 2822 at some various flow conditions. Considering the comparison result shows that the developed computer code as inviscid solver are able to provide the result as obbtained from the experimental and Fluent software.

7. References

[1] LeVeque, Randall J. Finite volume methods for hyperbolic problems. Vol. 31. Cambridge university press, 2002

[2] K. A. Hoffman and S.T .Chiang, Computational-FluidDynamics Volume I, Engineering Education System, USA, 2000.

[3] J. Blazek , Computational Fluid Dynamics: Principles and Applications. Elsevier science. 2008

[4] J.C. Tannehill, D.A. Anderson. D. A, R.H. Pletcher , Computational Fluid Mechanics and Heat Transfer, 2nd Ed., Taylor & Francis, New York, 1997.

[5] K.. A . Hoffman, S.T .Chiang, Computational-Fluid-Dynamics Volume I and II, Engineering Education System, USA, 2000.

[6] J. H. Ferziger, and M. Perić, Computational Methods for Fluid Dynamics, Springer, 1996.

[7] Causon, D. M., C. G. Mingham, and L. Qian. Introductory Finite Volume Methods for PDEs. Bookboon, 2011.

[8] RabahHaoui, Effect of Mesh Size on the Viscous Flow Parameters of an Axisymmetric Nozzle.,"World Academy of Science, Engineering and Technology 59, 2011.

[9] Thompson, Joe F., Bharat K. Soni, and Nigel P. Weatherill, eds. Handbook of grid generation. CRC press, 2010.

[10] T. Cebeci, and A.M.O. Smith, Analysis of Turbulence Boundary Layers, London: Academic Press, 1974.

.