Applied and Computational Mathematics
2015; 4(1-1): 12-17
Published online February 9, 2015 (http://www.sciencepublishinggroup.com/j/acm)
doi: 10.11648/j.acm.s.2015040101.13
ISSN: 2328-5605 (Print); ISSN: 2328-5613 (Online)

Comparison between Finite Volume Method (FVM) Based on Inviscid and Viscous Flow with Experimental and Fluent Results

Abobaker Mohammed Alakashi*, **Bambang Basuno, Hasan Taher. M. Elkamel**

Department of Aeronautic, University Tun Hussein Onn Malaysia, Johor, Malaysia

Email address:
abobaker.alakashi@gmail.com (A. M. Alakashi), bambangb@uthm.edu.my (B. Basuno), hassan.elkamel@gmail.com (H. T. M. elkamel)

To cite this article:
Abobaker Mohammed Alakashi, Bambang Basuno, Hasan Taher. M. Elkamel. Comparison between Finite Volume Method (FVM) Based on Inviscid and Viscous Flow with Experimental and Fluent Results. *Applied and Computational Mathematics*. Special Issue: New orientations in Applied and Computational Mathematics. Vol. 4, No. 1-1, 2015, pp. 12-17. doi: 10.11648/j.acm.s.2015040101.13

Abstract: The Finite Volume Method (FVM) is currently the most popular method in CFD. The main reason is that it can resolve some of the difficulties that the other methods have. Finite volume methods are a class of discretization schemes that have proven highly successful in approximating the solution of a wide variety of conservation law systems [1]. Finite volume method can be classified into three groups: (1) Cell-centered scheme, (2) Cell-vertex scheme with overlapping control volumes and (3), Cell-vertex scheme with dual control volumes [2]. The present work used Finite volume based Cell Cell-centered. This approach used the grid cell identical to its control volume. While in view of a manner the grid cells in this work can be defined numerically, it can follow as a structured grid based on Elliptic grid generation PDEs [3]. Computer code had been developed by using a cell centered Finite volume scheme combined with structured grid approach. The computer codes applied for the case of compressible flow past through an airfoil NACA 0012, in which the flow problem can be treated as purely inviscid flow or as the flow with viscous effect but considered to be as a laminar flow. The comparison result presented in term of pressure coefficient Cp for different angle of attack using available experimental result and the result provided by Fluent software. In term for the case of flow problem treated as an inviscid flow, both the developed computer code and Fluent software produce the result closed to the experimental result. However if the developed computer code as well as fluent software treated the flow problem to include the viscous effect by considering them as a laminar flow both are slightly deviate with the experimental results. Strictly speaking the present developed computer code give a similar result as the experimental result, which both showing that this type of airfoil having a sensitive effect to the angle of attack. A small change of angle of attack will produce a significant change to the location of shock will occurred.

Keywords: Finite Volume Method, Cell Cell-Centered scheme, Navier-Stokes, Euler Equation

1. Introduction

All CFD codes contain three main elements: (1) A pre-processor, which is used to input the problem geometry, generate the grid, and define the flow parameter and the boundary conditions to the code. (2) A flow solver, which is used to solve the governing equations of the flow subject to the conditions provided. (3) A post-processor, which is used to massage the data and show the results in graphical and easy to read format [4]. In the last decades, finite volume methods have been greatly successful in solving engineering models of flows in porous media on complex geometrical domain, because the finite volume formulation works on general polygonal and polyhedral meshes [5]. Several discretization methods have been implemented to stimulate inviscid incompressible fluid and viscous incompressible fluid. Among them, the Finite Volume Method (FVM) Cell-centered scheme has proved to be simple yet very efficient in computing such flows [6]. However, to employ the above method has to make a mesh flow domain appropriately. This can be done by firstly choosing the grid topology. When providing a great grid mesh with flexibility as was applied in this is study Elliptic grid generation PDEs on simple geometry in the case of external an airfoil NACA 0012 model, will provide a clear and accurate results [7,8]. In other hand, in fluid mechanics, the Euler equations and Navier-Stokes equations are applied to depict motion state of inviscid incompressible fluid and viscous incompressible fluid, respectively. In case of the inviscid flow (i.e. when viscosity = 0 and there is no heat conduction), then the Navier-Stokes equation reduces to the Euler equation, so the latter is a special instance of the former [5]. The analogy here is similar to solving for the acceleration of a box sliding down on an incline, when there is and where there is not kinetic friction; of course the idealized scenario with no friction present is easier to solve. But, that viscosity can be roughly thought of as liquid friction or thickness, and should not be ignored in describing motion of the boundary layer or when turbulence is present [9.10]. The Euler equation is essentially Newton's second law applied on a flowing infinitesimal volume element and it addresses conservation of mass, momentum, and energy absent the effect of viscosity. Therefore it is an easier idealized scenario which was solved a long time ago via a technique known as potential theory. Euler equation will in fact yield the famous Bernoulli equation when there is a further assumption of incompressibility (= constant density) [3,11]. It should also be mentioned that proving the general existence of solutions for the Navier-Stokes equations equation is still one of the major unsolved problems in mathematics [12, 13]. The present work focused on the development of computer code based on Cell Centered scheme combined with structured grid. The governing equation used here is a two dimensional compressible laminar flow, in which the solution of inviscid flow can be obtained through switching the viscosity equal to zero. This computer code applied for the case of flow past through Naca 0012 airfoil at a fixed Mach number for four different angle of attacks in which the experimental results are available. The comparison result in term of pressure coefficient distribution between the present code, Fluent software and experimental results indicate that the present code able to produce as the experimental result as well as the Fluent software. The sensivity in term of the shock wave location due to angle of attack owned by this type of airfoil also be able captured by the present developed computer code.

2. Governing Equations

2.1. Governing Equations Of Viscous Two Dimensional

The dimensionalized governing equations of the fluid flow are given respectively by the continuity equation

$$\frac{\partial u^*}{\partial x^*}+\frac{\partial v^*}{\partial y^*}=0 \qquad (1)$$

x −momentum equation

$$u^*\frac{\partial u^*}{\partial x^*}+v^*\frac{\partial v^*}{\partial y^*}=-\frac{1}{\rho}\frac{\partial p^*}{\partial x^*}+\upsilon\left(\frac{\partial^2 u^*}{\partial x^{*2}}+\frac{\partial^2 u^*}{\partial y^{*2}}\right) \qquad (2)$$

y −momentum equation

$$u^*\frac{\partial v^*}{\partial x^*}+v^*\frac{\partial v^*}{\partial y^*}=-\frac{1}{\rho}\frac{\partial p^*}{\partial y^*}+\upsilon\left(\frac{\partial^2 v^*}{\partial x^{*2}}+\frac{\partial^2 v^*}{\partial y^{*2}}\right) \qquad (3)$$

Where u and v are the velocity components in the x and$y$ directions respectively, p is the pressure, ρ is the constant density, and $\upsilon$is the viscosity.

Using the dimensionless definitions, [12,15],

$$t=\frac{t^*U}{h},\ x=\frac{x^*}{h},\ y=\frac{y^*}{h},\ u=\frac{u^*}{U},\ v=\frac{v^*}{U},\ p=\frac{p^*}{\rho U^2}$$

The governing equations (1) to (3) become

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \qquad (4)$$

$$u\frac{\partial u}{\partial x}+v\frac{\partial v}{\partial x}=\frac{\partial p}{\partial x}+\frac{1}{R_e}\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right) \qquad (5)$$

$$u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}=\frac{\partial p}{\partial y}+\frac{1}{R_e}\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right) \qquad (6)$$

Where $R_e=Uh/v$ is the Reynolds number

2.2. Governing Equations of Inviscid Two Dimensional

For the simplicity in the explanation of the Cell Centered Finite volume, the governing equation will be used here is the governing equation for purely inviscid flow. As in usually adopted in inviscid flow analysis, let the flow variables are denoted by p, ρ, u, v, E and H which they are representing the pressure, density, Cartesian velocity components, total energy and total enthalpy respectively. For a perfect gas one has.

$$\mathrm{E}=\frac{\mathrm{p}}{(\gamma-1)\rho}+\frac{1}{2}(\mathrm{u}^2+\mathrm{v}^2) \quad \text{and} \quad \mathrm{H}=\mathrm{E}+\frac{\mathrm{p}}{\rho} \qquad (7)$$

Where γ is the ratio of specific heat, 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 written in integral form for a region Ω with boundary $d\Omega$ as:

$$\frac{\partial}{\partial t}\iint_{\Omega}\vec{\mathrm{U}}\mathrm{dxdy}+\oint_{\mathrm{d}\Omega}\left(\vec{\mathrm{F}}\mathrm{dy}-\vec{\mathrm{G}}\mathrm{dx}\right)= \qquad (8)$$

Where x and y are Cartesian coordinates and

$$\vec{\mathrm{U}}=\begin{pmatrix}\rho\\ \rho u\\ \rho v\\ \rho E\end{pmatrix},\quad \vec{\mathrm{F}}=\begin{pmatrix}\rho\\ \rho u^2\\ \rho uv\\ \rho uH\end{pmatrix},\quad \vec{\mathrm{G}}\begin{pmatrix}\rho\\ \rho uv\\ \rho v^2+p\\ \rho vH\end{pmatrix}$$

The discretization procedure follows the method of lines in decoupling the approximation of the spatial and temporal terms. The computational domain is divided into quadrilateral cells as in the sketch figure (1), and a system of ordinary differential equations is obtained by applying equation (1) to each cell separately. The resulting equations can be solved by several alternative time stepping schemes.

In general, the conservation laws is applied to the cell ABCD as shown in figure (1)

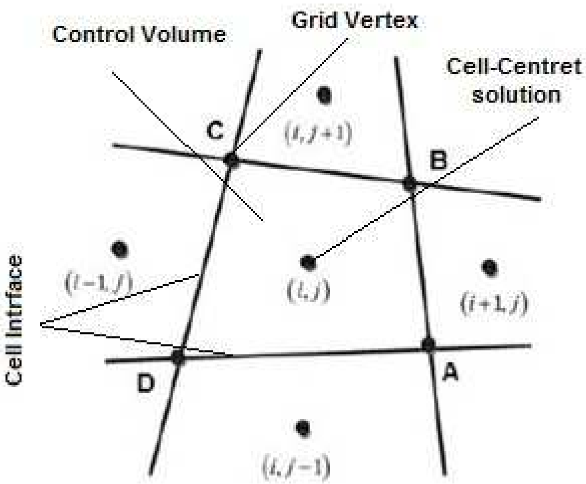

Figure 1. *Cells in the finite-volume approach.*

The grid cell ABCD as shown in the figure above has a side AB surface vector $\vec{S}_{AB}$ and its normal direction $\hat{j}_{AB}$ is defined as:

$$\vec{S}_{AB} = \Delta y_{AB}\hat{i} - \Delta x_{AB}\hat{j}$$

$$\hat{j}_{AB} = (y_B - y_A)_i - (X_B - X_A)_j$$

While the area of grid cell ABCD [1, 2] can be obtained from:

$$\Omega_{ABCD} = \frac{1}{2}\left|\vec{X}_{AC} - \vec{X}_{BD}\right|$$

Where

$$\vec{X}_{AB} = \vec{X}_B - \vec{X}_A$$

Or in term of coordinate point (X, Y) as:

$$\Omega_{ABCD} = \frac{1}{2}[(X_C - X_A)(y_D - y_B) - (X_D - X_B)(y_C - y_A)]$$

Determination of the flux vector can be done in various approaches. The flux E crossing the side surface AB denoted as $\vec{E}_{AB}$ can be yielded using one of the following approaches.

a. Average of fluxes

$$E_{AB} = E\left(\frac{Q_{ij} - Q_{i+1,j}}{2}\right)$$

Where:

$$E_{ij} = E(Q_{ij})$$

a. Flux of the average flow variable:

$$E_{AB} = E(\frac{Q_{ij} + Q_{i+1,j}}{2})$$

b. Average of fluxes in A&B

$$E_{AB} = \frac{1}{2}(E_A + E_B)$$

In the last approach, approach c, the flux E_A can be determined by firstly defining the flow variable Q at A as given below:

$$Q_A = \frac{1}{4}\left(Q_{ij} + Q_{i+1,j} + Q_{i+1,j-1} + Q_{i,j+1}\right) \quad (9)$$

Then the average of the fluxes E_A becomes:

$$E_A = \frac{1}{4}\left(E_{ij} + E_{i+1,j} + E_{i+1,j-1} + E_{i,j+1}\right) \quad (10)$$

In defining the flux vectors E and H as explained above, the implementation of the finite volume makes the Eq. (1) can be transformed into an ordinary differential equation with respect to time as given below[12,13].

$$\frac{\partial}{\partial t}Q_{ij}\Delta X\Delta y + \left(E_{i+\frac{1}{2},j} - E_{i-\frac{1}{2}}\right)\Delta y + \left(Q_{i,j+\frac{1}{2}} - Q_{i,j\frac{1}{2}}\right)\Delta X = H_{ij}\Delta X\Delta y \quad (11)$$

Or

$$\frac{\partial Q_{ij}}{\partial t} + \frac{(E_{i+\frac{1}{2},j} - E_{i-\frac{1}{2},j})}{2} + \frac{(Q_{i,j+\frac{1}{2}} - Q_{i,j-\frac{1}{2}})}{\Delta y} = H_{ij} \quad (12)$$

Through above equation, the flow problems are solved. However to solve above equation one has to make a mesh flow domain appropriately. This can be done by firstly choosing the grid topology [2, 16, 6].

3. Grid Generation

Grid (mesh) generation is, of course, only a means to an end: a necessary tool in the computational simulation of physical field phenomena and processes. Grid (mesh) generation is truly a worldwide active research area of computational science

3.1. Elliptic Grid Generation

This method proved in computational space producing a smooth grid in the entire domain. Thus, high quality, boundary orthogonal grids can be generated also allows the user to prescribe the angle between a grid line and boundary and to control the grid spacing and the expansion ratio near surfaces, which made it the most popular method among all of the methods

This type of grid generation is motivated by the maximum principle for elliptic partial differential equations. Where the inverse grid transformation, $\xi(x, y)$, $\eta(x, y)$ as the solution of

$$\xi_{xx} + \xi_{yy} = 0 \quad (13)$$

$$\eta_{xx} + \eta_{yy} = 0 \quad (14)$$

When $0 \le \xi \le 1$ and $0 \le \xi \le 1$ are monotone on the boundaries. It then follows from the maximum principle that ξ and η will stay between these values. Furthermore, there will be no local extreme in the interior, and thus grid lines cannot fold. The equations (14) are formulated in the $x - y$ domain, and have to be transformed to the unit square, so can solve them there [16]. We use the unknown transformation itself to transform the equations (14). The transformed system then becomes

$$(x_\eta^2 + y_\eta^2)x_{\xi\xi} - 2(x_\xi x_\eta + y_\xi y_\eta)x_{\xi\eta} + (x_\xi^2 + y_\xi^2)x_{\eta\eta} = 0$$

$$(x_\eta^2 + y_\eta^2)y_{\xi\xi} - 2(x_\xi x_\eta + y_\xi y_\eta)y_{\xi\eta} + (x_\xi^2 + y_\xi^2)y_{\eta\eta} = 0$$

Specifying normal derivatives is here equivalent to specifying the distance between the first and second grid lines. These equations are then approximated by, e.g.

$$x_\xi = (x_{i+1,j} - x_{i-1,j})/2$$

$$x_\eta = (x_{i,j+1} - x_{i,j-1})/2$$

$$x_{\xi\xi} = x_{i+1,j} - 2x_{i,j} + x_{i-1,j}\text{etc}$$

Where now the index space $1 \le i \le n_i$ and $1 \le j \le n_j$ is a uniform subdivision of the (ξ,η) coordinates. $\xi = (i-1)/(ni-1)$, $\eta = (j-1)/(n_j-1)$. The number of grid points is specified as $n_i \times n_j$.

To introduce more control over the grid, so called control functions are introduced into (6)[14, 17]. The system then becomes

$$\xi_{xx} + \xi_{yy} = P(\xi,\eta) \quad (15)$$

$$\eta_{xx} + \eta_{yy} = Q(\xi,\eta) \quad (16)$$

Where, P, Q the known functions which using for the control of concentration of inner grid points. By interchanging the independent and the dependent variables, we obtain the transformed system in the transformed computational domain [8].

$$\alpha x_{\xi\xi} - 2\beta x_{\xi\eta} + \gamma x_{\eta\eta} + J^2(Px_\xi + Qx_\eta) = 0,$$

$$\alpha y_{\xi\xi} - 2\beta y_{\xi\eta} + \gamma y_{\eta\eta} + J^2(Py_\xi + Qy_\eta) = 0,$$

$$\alpha = x_\eta^2 + y_\eta^2,$$

$$\beta = x_\xi x_\eta + y_\xi y_\eta$$

$$\gamma = x_\xi^2 + y_\xi^2$$

$$J = x_\xi y_\eta + x_\eta y_\xi$$

4. Results and Dissection

4.1. The Compressible Flow Past Through Naca 0012 Airfoil

The airfoil geometry of NACA0012 as shown in figure2.a with zooming their mesh flow domain developed by used of Elliptic grid generator as depicted in the Figure 2.b. [16].

As an external flow problems, here the boundary condition had been applied with the incoming Mach-number M = 0.7550, static pressure [Pa] = 150.E+3 and static temperature [K] = 350.0. The analysis carried out for four different angle of attacks, namely at Angle of attack [deg]= 1.25^0, $2.^0$, 2.34^0 and 2.410 respectively, These four setting angle of attack produce their pressure coefficient distribution and make a comparison between the present code, fluent and experimental result as shown in the Figure 3 to Figure 6. These four angles of attack can be considered as a small angle of attack. The flow past through airfoil at small angle of attacks are normally behave as inviscid flow. So considering these four figures, they are clearly indicating that the inviscid present code provide the result which very close to the experimental result as well as to the inviscid Fluent software. However if the flow problem treated as viscous flow but considered as the laminar flow both the viscous present code as well as the fluent both produce a similar result but they are way from the experimental result.

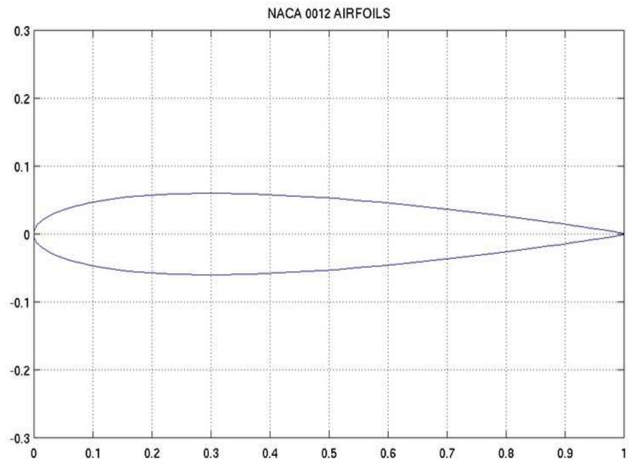

Figure 2.a. *Cross section of NACA 0012 AIRFOILS.*

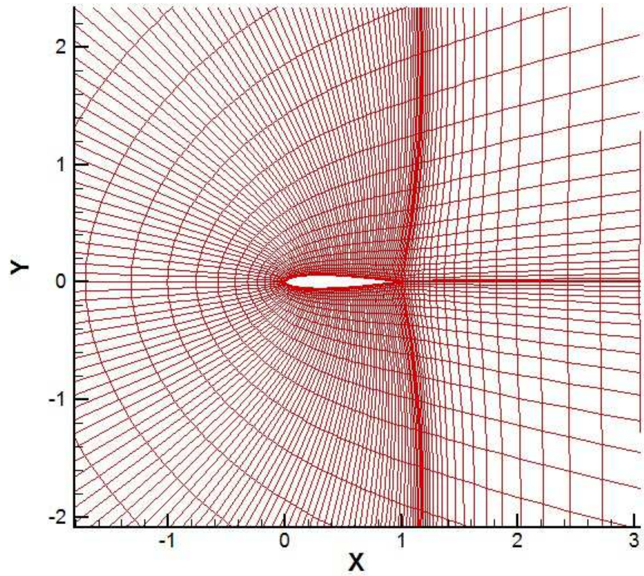

Figure 2.b. *Grid generation Cross section of NACA 0012 AIRFOILS.*

5. Conclusions

The present developed computer code based on cell-centered finite volume discretization technique had been successfully developed. The present code is able to produce the result as provided by Fluent software as well as by the experimental result. The effect of angle of attack to the flow behavior can also be captured, as it happen experimentally, by the present developed code for the case of flow past through airfoil NACA 0012.

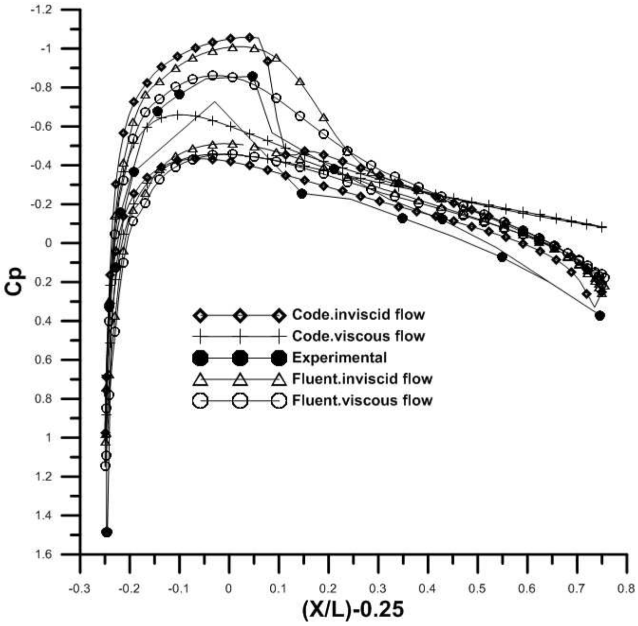

Figure 3. *Cpdistribution on NACA 0012 AIRFOILS at angle of attack 1.250^0.*

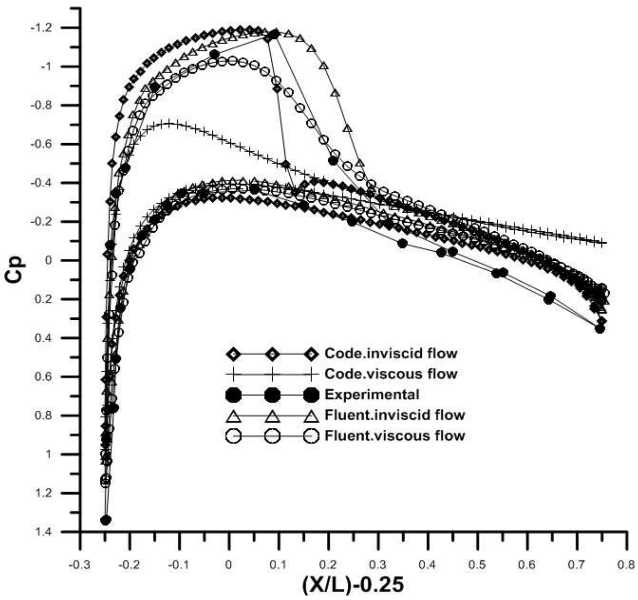

Figure 4. *Cpdistribution on NACA 0012 AIRFOILS at angle of attack 2^0.*

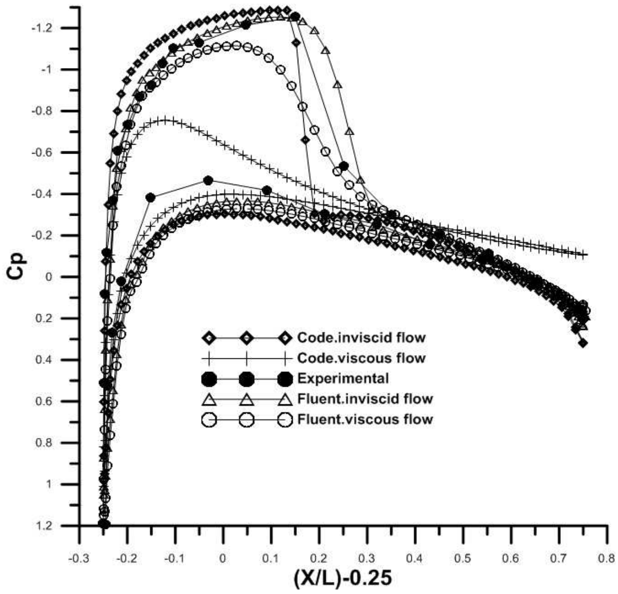

Figure 5. *Cpdistribution on NACA 0012 AIRFOILS at angle of attack 2.340^0.*

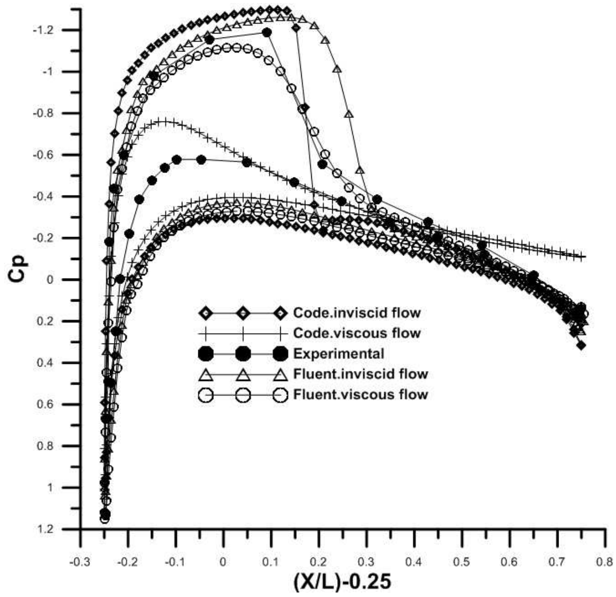

Figure 6. *Cpdistribution on NACA 0012 AIRFOILS at angle of attack 2.410^0.*

References

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

[2] J. Blazek, 'Computational Fluid Dynamics Principles and Applications' 2nd Ed.Switzerland, 2001

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

[4] J. D. Anderson, Jr, Computational Fluid Dynamics, the basics with applications, McGraw-Hill, 1995.

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

[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.

[11] H. Sclichting, and K, Gertsen, Boundary layer theory. 8th Revised and Enlarged Edition, Germany: Springer, pp.392-400 2000.

[12] Sorenson, Reese L. "A computer program to generate two-dimensional grids about airfoils and other shapes by the use of Poisson's equation." NASA Technical Memorandum 81198 (1980): 1-58.

[13] Morton, Keith W., and David Francis Mayers. Numerical solution of partial differential equations: an introduction. Cambridge university press, 2005.

[14] T.D. Taylor, "Numerical Method for Predicting Subsonic, Transonic, and Supersonic Flow", France, Tech. Rep. AGARD-AG-187, 1974.

[15] D.G. Korn, "Computation of Shock Free Transonic Flow for Airfoil Design", AEC Research and Development Report, New York, Courant Inst. of Mathematics and Science, NYO-1480-125, 1969

[16] C. Kroll, M. Aftosmis, and D. Gaitonde, "An Examination of Several High Resolution Schemes Applied to Complex Problems in High Speed Flows", Final Report Wright Laboratory, Ohio, AD-A250 814, 1992.

[17] A. Arias, O. Falcinelli, N.J. Fico, and S. Elaskar, "Finite Volume Simulation of a Flow Over a Naca 0012 Using Jameson, Maccormack, Shu and Tvd Esquemes", Mechanical Computational, vol. 16, Córdoba, pp. 3097-3116, 2007.