SEMARAK ILMU PUBLISHING

CFD Letters

Journal homepage:
https://semarakilmu.com.my/journals/index.php/CFD_Letters/index
ISSN: 2180-1363

Comparison of Pressure Distribution of Naca 0012 Between CFD Code and Experimental

Fatimah Yusop[1,*], Syafiq Syazwan Mustafa[1], Mariah Awang[1], Nik Normunira Mat Hassan[1], Hafsa Mohd Noor[2]

[1] Faculty of Engineering Technology, Universiti Tun Hussein Onn Malaysia, Johor, Malaysia
[2] Centre of Diploma Studies, Universiti Tun Hussein Onn Malaysia, Johor, Malaysia

ARTICLE INFO

Article history:
Received 10 December 2021
Received in revised form 15 January 2022
Accepted 16 January 2022
Available online 24 February 2022

Keywords:
CFD; Euler equation

ABSTRACT

Computational Fluid Dynamics (CFD) is a tool to solve engineering problem. Commercial CFD code are being used most engineering problem but rarely for external aerodynamic problem due some limitation of features. Compare to in house CFD code that able to written in certain area of interest and also implement specific discretization scheme to increase accuracy. In this work, the CFD code are developed by using high resolution scheme (ROE scheme and TVD scheme) for flow past through airfoil NACA 0012. Experimental result of wind tunnel is obtained from literature which are from Gregory and O'Reilly at Mach number M = 0.13 for different values of angle of attack at α= 0° and α = 10°. Another wind tunnel result from Haris at Mach number = 0.8 for different angles of attack α= 0° and α= 3.86°. Comparison results from the both codes indicate that developed CFD code by using TVD scheme able to give the closest result for both experimental.

1. Introduction

In CFD, mathematical modelling is used to represent the flow problem. Navier-Stokes equation is the highest hierarchy of governing equation that able to represent all flow phenomena. Unfortunately, to solve directly the equation is very difficult. Therefore, simplification of equation is doing according the flow condition [1].

For the inviscid flow governed by Euler equations. This equation is simply obtained from the Navier-Stokes equation through eliminating the viscous term. The Euler equation is representing as non-linear partial differential equation (PDE) [2]. For the case of supersonic speed, Euler equation can be a classified as hyperbolic PDE. Meanwhile, for the case of transonic flow, it can be classified as an elliptic-hyperbolic.

For many practical aerodynamic applications, Euler equation is relatively accurate for representing the flow field which includes both rotational and discontinuous (shock) phenomena in the flow and providing an excellent approximation for lift induced drag and wave drag. Furthermore,

* *Corresponding author.*
E-mail address: fatimahy@uthm.edu.my (Fatimah Yusop)

https://doi.org/10.37934/cfdl.14.2.3541

a robust Euler solver is an essential part of any Navier-Stokes solver. In addition to this, Euler equations promised to provide more accurate solutions of transonic flows [3].

Numerical scheme is one of the factors that contribute the accuracy of the CFD code. Higher resolution scheme such TVD scheme that designed to produce monotonic condition to prevent oscillatory behavior able to give high accuracy of result [4, 5]. The CFD code which developed based on TVD criteria had been widely used as CFD code for aircraft aerodynamics analysis such ENSOLV, EURANUS, ZEN, EDGE, and others [3]. Others application in fluid dynamics also used TVD such for 1D nozzle problem [6, 7], 2D symmetrical model [8] and many others.

Therefore, the papers present the comparison result of CFD code with wind tunnel result from literature. The CFD codes was developed to solve the compressible Euler equation for the flow over around NACA 0012. The first computer code is the code developed based on FDM according to Davis-Yee TVD scheme, while the second computer code was developed by the use of FVM according to Roe scheme.

2. Methodology

In CFD there are involves three processes namely pre-processing, solver and post processing.

2.1 Pre-Processing

Pre-processing involves the process meshing the flow domain. This paper implementation of C-topology in defining the flow domain as shown Figure 1.

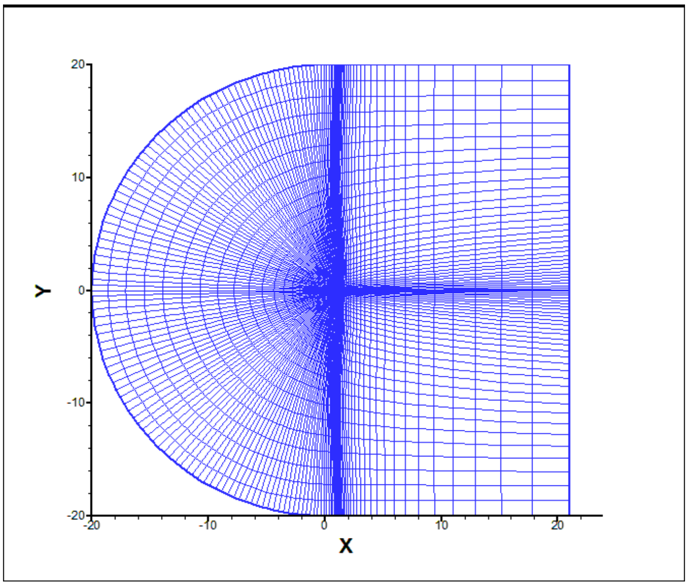

Fig. 1. Flow domain

2.2 Solver

Solver is the part where the CFD code was developed. The code was developed to solve the flow problem governed by compressible Euler equation. Therefore, unsteady, 2D compressible inviscid flows make the Euler equations in differential and in conservative form and vector notation can be written as [9, 10]

$$\frac{\partial Q}{\partial t}+\frac{\partial E}{\partial x}+\frac{\partial F}{\partial y}=0$$

In above equation Q is a state vector of dependent variables while E and F are the flux vector in x and y direction respectively. These three vectors are defined as

$$Q=\begin{bmatrix}\rho\\ \rho u\\ \rho v\\ \rho e_t\end{bmatrix},\quad E=\begin{bmatrix}\rho u\\ \rho u^2+p\\ \rho uv\\ \rho u\left(e_t+\frac{p}{\rho}\right)\end{bmatrix}\text{ and }\quad F=\begin{bmatrix}\rho v\\ \rho vu\\ \rho v^2+p\\ \rho v\left(e_t+\frac{p}{\rho}\right)\end{bmatrix}$$

The primitive variables are the density ρ, the velocity components u and v, and the pressure p. The total energy per unit of mass e_t as below

$$e_t = e+\frac{u^2+v^2}{2}$$

Numerical scheme that used are TVD scheme and Roe scheme. TVD scheme is not as specific as a scheme but more as properties. Therefore, TVD scheme contains a variety of scheme. That means, TVD scheme can be summarized to have the following physical characteristics [9, 11]

i. Entropy condition – A decrease of entropy associated with expansion shocks must not be admitted.
ii. Monotonic condition – This condition must be enforced to prevent oscillatory behavior in the numerical scheme.
iii. Total Variation Diminishing (TVD) – The total variation of any physically admissible solution must not be allowed to increase time. Basically, can be defined as follow

$$TV=\int\left|\frac{\partial u}{\partial x}\right|dx$$

A numerical scheme is said to be the total variation diminishing (TVD) if,

$$TV\left(u^{n+1}\right)\leq TV\left(u^{n}\right)$$

Meanwhile, for Roe scheme is based on a characteristic decomposition of the flux differences. The Roe scheme used flux formula at the interface of a control volume that is equal to the average fluxes of left and right states minus a differencing term which splits the difference of the fluxes on both sides of the control volume. The basic equation is expressed as below [12, 13].

$$\left(\vec{F}_c\right)_R-\left(\vec{F}_c\right)_L=(\bar{A}_{ROE})_{I+1/2}\left(\vec{Q}_R-\vec{Q}_L\right)$$

where A_{Roe} is Roe matrix and L and R the left and right state respectively.

2.3 Post-Processing

In the purpose of presenting the result which describes the flow past through object internally or as the case of external flow problems use a TECPLOT software. Hence the output of flow variables has to be arranged in such a way readable by TECPLOT software.

3. Results

The experimental result will be from the wind tunnel over NACA 0012 in terms of pressure coefficient distribution provided by Gregory and O'Reilly [14]. The wind tunnel test is carried out at the flow Mach number of 0.13 for different values of angle of attack, at α = 0° and α = 10°. Figure 2 shows the comparison result between two developed computer codes (TVD FDM and Roe FVM) at α= 0° which give a very good agreement to each other.

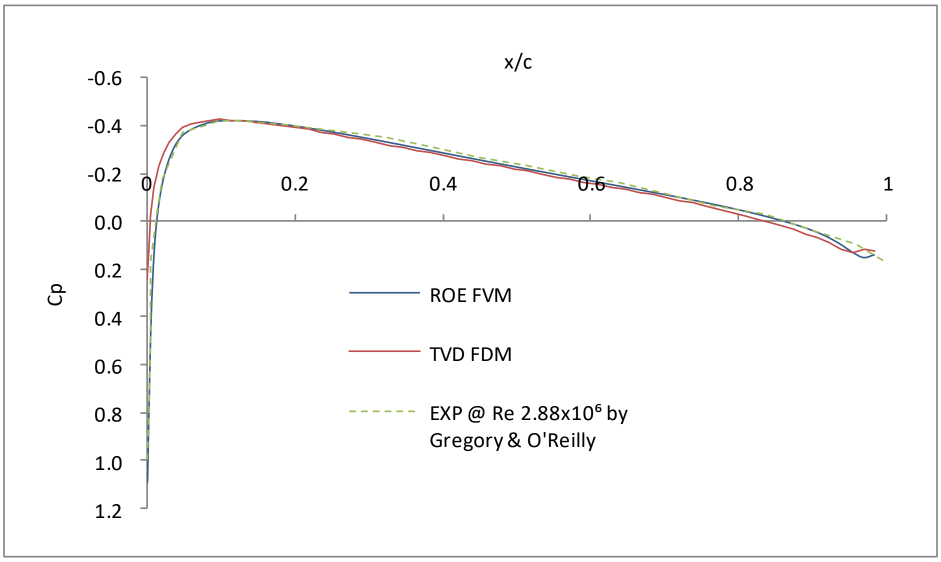

Fig. 2. Comparison of pressure coefficient distribution at α= 0° and M = 0.13

Figure 3 below shows the comparison result between TVD FDM, Roe FVM and the wind tunnel experimental result at α = 10°. The developed computer codes give their result close to each other, where both codes produced pressure coefficient curve with the same trend as provided by the experiment but a little bit shifted up.

Another experiment that related to airfoil has been carried out by Harris [15]. If the experimental work by Gregory and O'Reilly are carried out at a low Mach number. Haris did the experimental at high subsonic Mach number at M = 0.8. At this speed, the shock wave may exist in the flow field. Harris set the experiment at the angles of attack α= 0° and α= 3.86°. Based on these two angles of attack, the comparison results between the two developed computer codes and Harris, in terms of the pressure coefficient distribution is shown in Figure 4 and Figure 5 respectively. At α= 0°, The Roe FVM code provides result that shows the shock position is close to the experimental result. When TVD FDM is slightly placed the shock wave will move further downstream. When the angle of attack increases to α= 3.86°, the experiment found that the shock located will move more upstream compared to the location of shock at zero angle of attack. It is necessary to be noted that, both computer codes are able to produce the pressure coefficient in a good agreement with experiment in the region relatively away from the shock point.

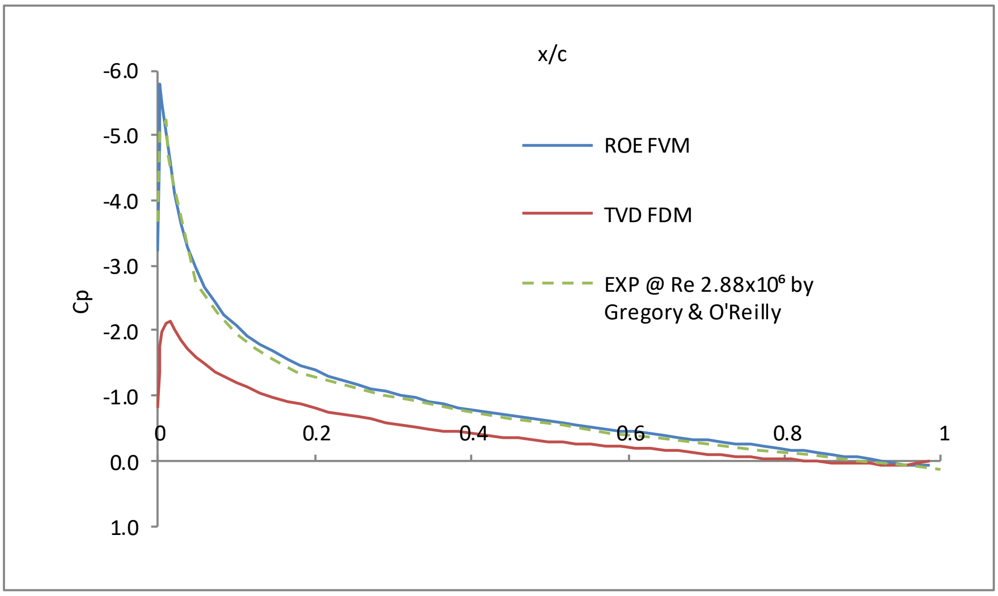

Fig. 3. Comparison of pressure coefficient distribution at α= 10° and M = 0.13

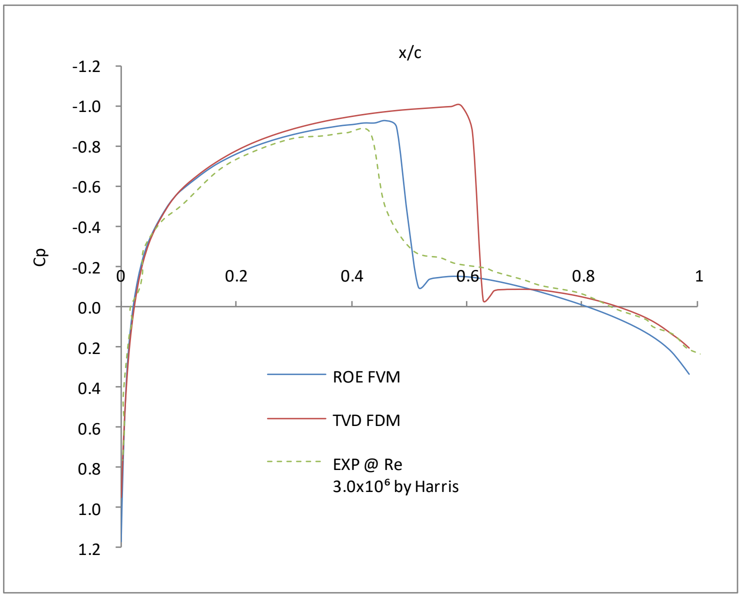

Fig. 4. Comparison of pressure coefficient distribution at α= 0° and M = 0.8

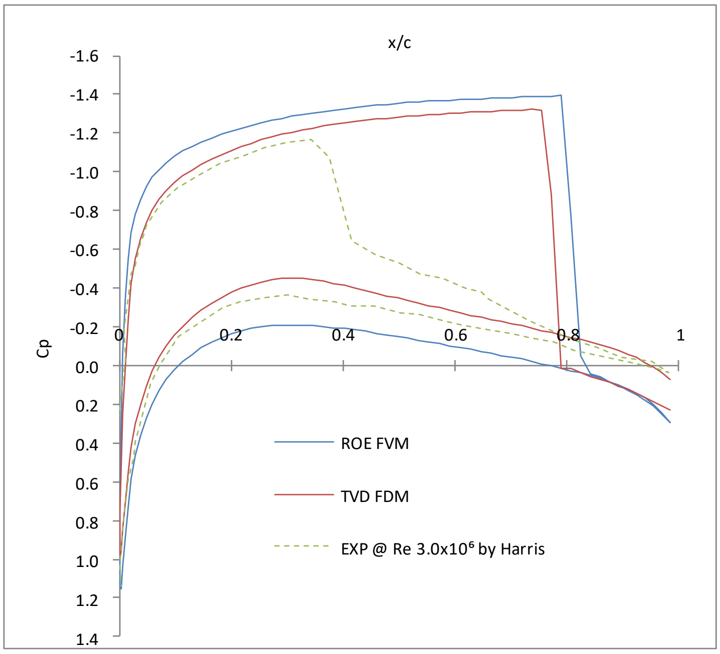

Fig. 5. Comparison of pressure coefficient distribution for at α= 3.86° and M = 0.8

4. Conclusions

The compared result for the case of flow past through a symmetrical airfoil NACA 0012 indicates that the developed computer code based on the Modified Fourth Order Runge-Kutta with Davis-Yee TVD scheme represents the computer code which are able to produce the result close to the experimental result compared to the result provided by ANSYS-FLUENT software or the CFD code based Roe finite volume method.

Acknowledgement
This research was supported by Universiti Tun Hussein Onn Malaysia (UTHM) funds.

References
[1] Hu, Zhe, Wenyong Tang, Hongxiang Xue, Xiaoying Zhang, and Jinting Guo. "Numerical simulations using conserved wave absorption applied to Navier–Stokes equation model." *Coastal engineering* 99 (2015): 15-25. https://doi.org/10.1016/j.coastaleng.2015.02.007

[2] Chen, Gui-Qiang G., Jun Chen, and Mikhail Feldman. "Stability and asymptotic behavior of transonic flows past wedges for the full Euler equations." *Interfaces and Free Boundaries* 19, no. 4 (2018): 591-626. https://doi.org/10.4171/IFB/394

[3] Vos, J. B., Arthur Rizzi, D. Darracq, and E. H. Hirschel. "Navier–Stokes solvers in European aircraft design." *Progress in Aerospace Sciences* 38, no. 8 (2002): 601-697. https://doi.org/10.1016/S0376-0421(02)00050-7

[4] Kroll, Norbert, Datta Gaitonde, and Michael Aftosmis. "A systematic comparative study of several high resolution schemes for complex problems in high speed flows." In *29th Aerospace Sciences Meeting*, p. 636. 1991. https://doi.org/10.2514/6.1991-636

[5] Chen, Ming-Hsiung, Chen-Chi Hsu, and Wei Shyy. "Assessment of TVD schemes for inviscid and turbulent flow computation." *International journal for numerical methods in fluids* 12, no. 2 (1991): 161-177. https://doi.org/10.1002/fld.1650120205

[6] Rosly, Nurhayati, Qamarul Ezani Kamarudin, and Bambang Basuno. "Comparative Study on Various Numerical Schemes for Solving Nozzle Flow Problems." *CFD Letters* 12, no. 6 (2020): 93-106. https://doi.org/10.37934/cfdl.12.6.93106

[7] Gu, Kai Xuan, Zhi Qiang Li, Jun Jie Wang, Yuan Zhou, Hong Zhang, Bing Zhao, and Wei Ji. "The effect of cryogenic treatment on the microstructure and properties of Ti-6Al-4V titanium alloy." In *Materials science forum*, vol. 747, pp. 899-903. Trans Tech Publications Ltd, 2013. https://doi.org/10.4028/www.scientific.net/MSF.747-748.899

[8] Yusop, Fatimah, Bambang Basuno, and Zamri Omar. "Application of Modified 4th Order Runge Kutta-TVD Scheme for the Flow Past through Symmetrical Model." In *Applied Mechanics and Materials*, vol. 315, pp. 181-185. Trans Tech Publications Ltd, 2013. https://doi.org/10.4028/www.scientific.net/AMM.315.181

[9] Chung, T. J. *Computational fluid dynamics*. Cambridge university press, 2002. https://doi.org/10.1017/CBO9780511606205

[10] Hoffmann, Klaus A., and Steve T. Chiang. "Computational fluid dynamics volume I." *Engineering Education System* (2000).

[11] Chang, Yu-Li. *Development of a CFD code using TVD schemes and advanced turbulence models for incompressible flow simulations*. Michigan Technological University, 1997.

[12] Blazek, Jiri. *Computational fluid dynamics: principles and applications*. Butterworth-Heinemann, 2015. https://doi.org/10.1016/B978-0-08-099995-1.00012-9

[13] van Leer, Bram. "An introduction to the article "Reminiscences about difference schemes" by SK Godunov." *Journal of Computational Physics* 153, no. 1 (1999): 1-5. https://doi.org/10.1006/jcph.1999.6270

[14] Gregory, Nigel, and C. L. O'reilly. "Low-speed aerodynamic characteristics of NACA 0012 aerofoil section, including the effects of upper-surface roughness simulating hoar frost." (1970).

[15] Harris, Charles D. *Two-dimensional aerodynamic characteristics of the NACA 0012 airfoil in the Langley 8 foot transonic pressure tunnel*. No. NASA-TM-81927. 1981.