Validation Solution of Incompressible Viscous Flow Over Multi-Element Airfoil

Dr. Asya A. Gabbasa[1], Dr. Tarak Assaleh[2], and Siddig Dabbashi[2]
1 Dept. of Mechanical Engineering, Faculty of Engineering, Zawia University
2 Dept. of Mechanical Engineering, Faculty of Engineering, Sabrata University

Abstract:

Computation fluid dynamics (CFD) simulations are very important tools, which use algorithms and numerical methods for analyzing and solving problems in fluid mechanics. Computer programs are adopted in order to simulate the interaction between gases and liquids with boundary conditions. The programs have the ability to undertake speedy and accurate simulations of complex scenarios, which include turbulent and transonic flows.

This paper aims to study an incompressible, viscous, turbulent flow over multi-element airfoils. The governing equations (NS) are solved by using k-epsilon model and SIMPLE algorithm turbulent, steady-state, incompressible flow using the commercial code ANSYS Fluent. Detailed studies are done at different angle of attack flight conditions. By using k-epsilon model and simple algorithm we have obtained the accurate results; that is successful in comparing with wind tunnel results of NACA23012

Keywords: Fluid mechanics, Naiver-Stokes equations, viscous fluid flow

Nomenclature

$\rho\overline{u'^2}$, $\rho\overline{v'^2}$, $\rho\overline{w'^2}$ *are three normal stresses*

$\rho\overline{u'v'}$, $\rho\overline{u'w'}$, $\rho\overline{v'w'}$ *are three apparent stresses.*

u, v, w: velocity components in (x, y, z) coordinates

u', v', w': fluctuation velocities

ρ *: density*

p: pressure

μ*: flow viscosity*

(g_x, g_y, g_z)*: gravitational components in the (x, y, z) coordinates.*

1.Introduction.

Numerical methods for an incompressible viscous flow is a major part of the rapidly growing field of computational fluid dynamics (CFD). At present CFD is emerging as an operative tool in many parts of industry and science. However, CFD is not a mature field either from a natural scientist's or an application engineer's point of view; robust methods are still very much under development, many different numerical tracks are still competing, and reliable computations of complex multi-fluid flows are still (almost) beyond reach with today's methods and computers.

Computational fluid dynamic expensive simulation codes based on mathematical models appropriate to the system, are in wide-spread use throughout the engineering industry. For example, in the field of computational fluid dynamics (CFD) a single evaluation of the model may take several hours of computer run time. With the help of modern computer facilities, numerous difficult and mathematically intensive problems are now readily solved. The aircraft design community is using computational fluid dynamics (CFD) and computational structural mechanics tools to the place of traditional approaches centering on simplified theories and wind tunnel testing [1].

The Naiver-Stokes equations are the governing equations for fluid flow. Theyare a system of nonlinear partial differential equations. There are few exact solutions of the (NS) equations, and even for simple geometries, the equations have to be solved numerically. With increasing the Reynolds number, the accuratesolution of the equations becomes more difficult. A number of techniques and algorithms have been developed to obtain an accurate solution of the equation for high Reynolds numbers". The solutions of the Reynolds-averaged Naiver-Stokes (RANS) equations in the application of computational fluid dynamics (CFD) to aircraft design, involve challenges such as Computational Efficiency, Human Efficiency and Accuracy of Physical Modelling.

"The flow fields from a turbulent channel simulation are used to compute the budgets for the turbulent kinetic energy (k) and its dissipation rate (epsilon). Data from boundary layer simulations are used to analyze the dependence of the eddy-viscosity damping-function on the Reynolds number and the distance from the wall. The computed budgets are used to test existing near-wall turbulence models of the k-epsilon type"[2].

"The k-epsilon (k-ϵ) model for turbulence is the most common to simulate the mean flow characteristics for turbulent flow conditions"[3]. The k-ϵ model is shown to be applicable for free-shear flows, such as the ones with relatively small pressure gradients. [4]

Formulation

1- Governing Equations

Computational Fluid Dynamics (CFD) is a means of finding a solution to the equations that govern the way in which fluid flow. Three equations form the basis of any statement or calculation of fluid flow. These are the conservation of mass, conservation of energy and Newton’s second law. The equations derived from Newton’s second law, known as Navier-Stokes equations. Navier Stokes equations form the basis of most CFD problems. These are used to determine outcomes from any fluid flow that occurs in the single-phase. Consequently, the equations for governing incompressible turbulent flow can be represented in vector form, as follows:

$$\nabla \cdot \mathrm{V} = 0 \tag{1}$$

$$\rho \frac{D\mathrm{V}}{Dt} = \rho g - \nabla p + \nabla \cdot \tau_{ij} \tag{2}$$

Where;

$$\tau_{ij} = \underbrace{\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)}_{\text{Laminar}} - \underbrace{\rho\overline{\acute{u}_i\acute{u}_j}}_{\text{Turbulet}}$$

The body force is ignored (flow of air).

Where there is significant fluctuation in both pressure and velocity, as

$$u = \bar{u} + u' \qquad v = \bar{v} + v'$$
$$w = \bar{w} + w' \qquad p = \bar{p} + p' \qquad (3)$$

The three-dimensional expression for steady state incompressible viscous turbulent fluids can be expressed as follows [7]:

$$\rho\left(u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}\right) = -\frac{\partial p}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right) - \rho\overline{\acute{u}^2} - \rho\overline{\acute{u}\acute{v}} - \rho\overline{\acute{u}\acute{w}}$$

$$\rho\left(u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z}\right) = -\frac{\partial p}{\partial y} + \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2}\right) - \rho\overline{\acute{u}\acute{v}} - \rho\overline{\acute{v}^2} - \rho\overline{\acute{v}\acute{w}}$$

$$\rho\left(u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z}\right) = -\frac{\partial p}{\partial z} + \mu\left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2}\right) - \rho\overline{\acute{u}\acute{w}} - \rho\overline{\acute{v}\acute{w}} - \rho\overline{\acute{w}^2} \qquad (4)$$

2- Flow Equation Solver

The Fluent software package is used to solve CFD problems. This software operates using a method of finite-volume, which allows for many different types of physical models to be used. For this paper, a two-equation turbulent model was used to solve the governing equations. Here, the Navier-Stokes equations were used in their integral form, with the assumption of constant viscosity and incompressible flow. The realizable k-ε model for a single transport equation was used. This allowed for the individual determination of the turbulent kinetic energy and the rate at which it dissipated. This model was particularly relevant for this study, as it made use of an enhanced wall treatment. An incompressible steady state two-dimensional flow was assumed for this paper which allowed for variables to be solved using a Semi-Implicit Method for Pressure-Linked Equation as the algorithm. The interpolating scheme used in this study was a first order upwind method. This allows for the results to be quite close to experimental results.

Methodology: -

The base geometry of a NACA 23012 airfoil with a 23012 external airfoil flap was used (0.2 C_w) as shown in Figure below.

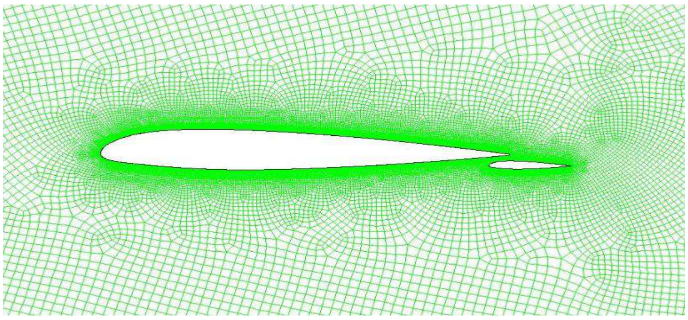

Figure 1. Structured domain mesh for NACA23012 with flap

The boundary specification is an important aspect of generating a model, and this is crucial both for external and internal boundaries. As shown in Figure 2.

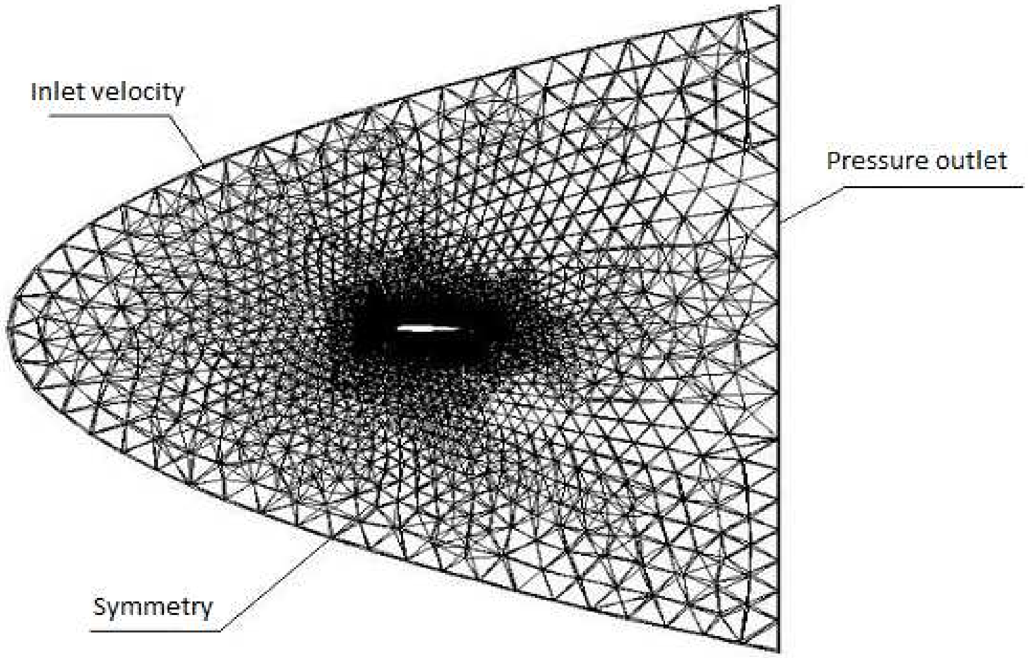

Figure 2. Specification zones for the model

Results: -

The flow velocity contours around NACA23012 with an external flap airfoil is depicted in Figure 3. The velocity scale depicts a maximum value of 45 m/s for $\alpha=0^o$, which is depicted by the red colors, while the minimum, which is zero illustrated by the use of dark blue colors.

The figure also illustrates that airfoil surfaces have high velocities while stagnation point has low velocities at leading and trailing edges.

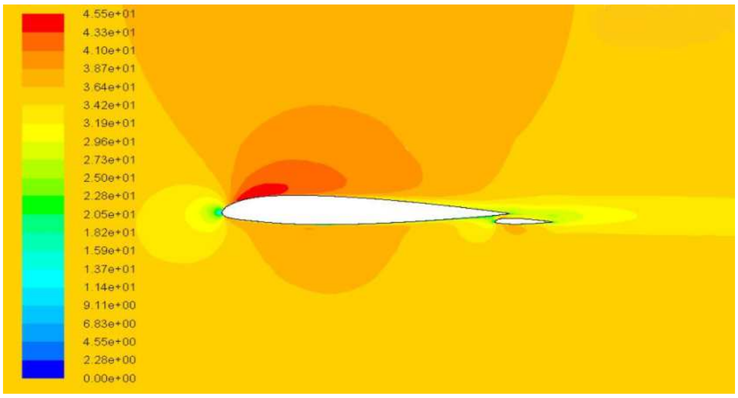

Figure 3 Contour Velocity magnitude (m/s) at 0 degree

The study of aerodynamic performance of various sections of an airfoil is effectively studied with reference to the pressure distribution over the airfoil. The expression of this pressure distribution is in terms of the coefficients of pressure. Figure 4 illustrates C_p versus chord length at 0 attack angles. As shown in figure 5 the chord length varies from 0 to 0.25 m from leading to trailing edge respectively for main airfoil, while to 0.3 m for the flap. At the leading edge, a stagnation point is obtained. This point is regarded as the section where V= 0 for incompressible flow and C_p=1.0 at the stagnation point. The Figures 4 and 5 depict that $C_p$begins at a stagnation point of 1.0 near to the leading edge, increases rapidly (pressure decreases) on both lower and upper surfaces, and eventually attains a minimal $C_p$as a positive value close to the trailing edge.

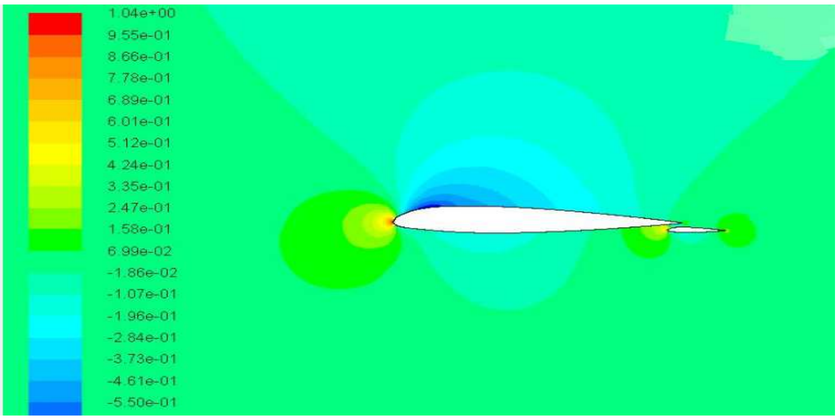

Figure 4 Contour of Pressure Coefficient at 0 degree

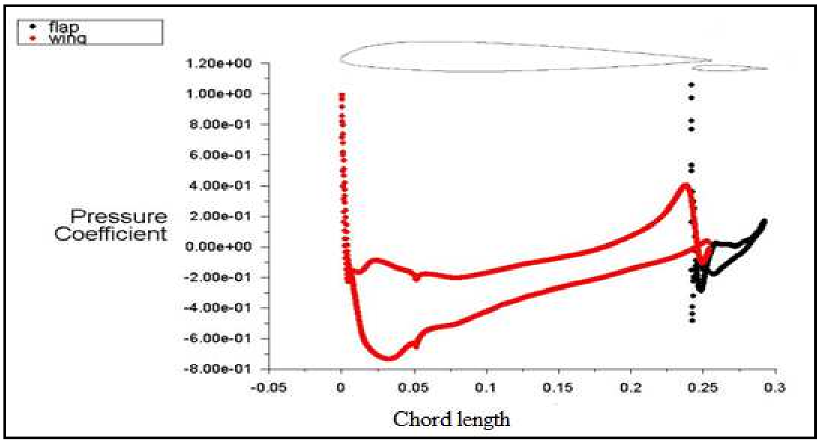

Figure 5 Pressure Coefficient at 0 degree

Comparisons with Expermental Data

By using k-epsilon model and SIMPLE algorithm turbulent, steady-state, incompressible flow, the NACA23012 experimental wind tunnel results [6] are compared with computation results for the purpose of validation. Wind tunnel and CFD results plotted on the same plot for the comparison purposes. The experimental and computed drag and lift coefficients for NACA23012 at a constant free stream velocity and realizable k-ε turbulent model are compared as indicated in Figures 6 and 7.

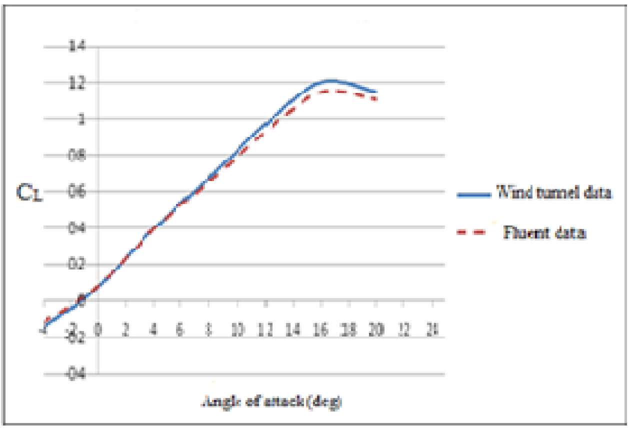

Figure 6. Comparisons CL between CFD and Wind Tunnel Data

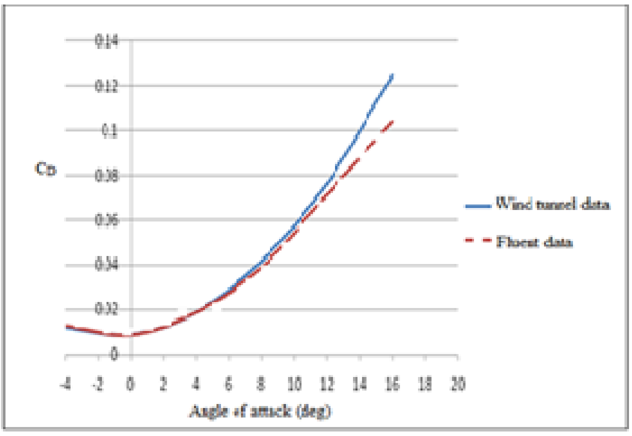

Figure 7. Comparison CD between CFD and Wind Tunnel Data

Conclusion

An incompressible viscous flow over 2D multi-element airfoil was numerically investigated. Computational fluid dynamic (fluent) was used as a solver for solution of Naiver-Stocks equations. The results obtained depict that there is agreement between CFD and wind tunnel results. Both data sets depict an identical drag and lift coefficient at different degree attack angle. The k-e model gavesignificant results, and was much easier to use, particularly for multi-element flows. This approach is capable of producing accurate solutions. Accurate pressure prediction was shown for geometry with two airfoils.

References

1. *J. Martins, J. Alonso and J. Reuther,(2005), "A Coupled-Adjoint Sensitivity Analysis Method for High-Fidelity Aero-Structural Design".Optimization and Engineering, 6, 33–62, 2005*
2. *Mansour, N. N.; Kim, J.; Moin, P.(1987), " Near-wall k-epsilon turbulence modeling" NASA Technical Reports Server (NTRS).*
3. *https://www.science.gov/topicpages/k/k-epsilon+turbulence+models.*
4. *Bardina, J.E., Huang, P.G., Coakley, T.J. (1997), "Turbulence Modeling Validation,Testing,and Development", NASATechnical Memorandum 110446*
5. *. Nielson, Eric J., (December 1998), "Aerodynamic design sensitivities on an unstructured mesh using the Naiver-Stokes equation and a discrete adjoint formulation", Ph.D. dissertation, Department of Aerospace and Ocean Engineering, Virginia Polytechnic Institute, Virginia.*
6. *Platt, R. and Abbott, I., "Aerodynamic characteristics of NACA. 23012 and 23021 airfoils with 20-percent-chord external- airfoil*

flaps of NACA 23012 section", National Advisory Committee for Aeronautics, 1937.

7. *Versteeg, H., Malalasekera, W., "An introduction to computational fluid dynamics, The Finite Volume Method", Second edition, pp 63-64, 2007.*
8. *White, F., "Viscous Fluid Flow", Third Edition, 2006.*