Determination of the Pressure and Velocity of the flow around an airfoil using Finite Element Method

Dr. Anwar jaafar Mohammed jawad                      Dr. Ali Yousif Abdulla

Abstract
Numerical study of 2-dimensional, steady state and subsonic flow around the airfoil has been achieved. The airfoil was placed in a regular duct of the airflow, which is the physical domain of the problem under consideration. The basic governing equations of the flow were rebuilt from Cartesian coordinates (x, y) to local coordinates (ζ, η).

Linear (quadrilateral isoperimetric) element has been used in the flow problem around the airfoil and solved by finite element technique using the primitive variables formulation. A wide range of angle of attack was covered. The results were presented in terms of pressures and velocity vectors showing the behavior of the flow around the airfoil. The stagnation point and the circulation are shown at the airfoil surfaces, which are caused by the relatively high angle of attack and viscous effects. Angles of attack 2, 8 and 13 degrees were selected at the aircraft takeoff.

Numerical simulation of the flow around the airfoil was performed using a finite element program “ANSYS package Release 5.4”. Results were in good agreement.

1. Introduction
Numerical simulation of the flow around airfoils is important in the aerodynamic design of aircraft wings and turbo machinery components. These lifting devices often attain optimum performance at the onset of separation. Thus separation phenomena must be dealt with if the analysis is aimed at practical application. When airfoils operate at small angles of attack, the viscous effects introduce only minor modification to the inviscid flow. In this case, an ordinary boundary layer analysis is adequate to describe the viscous flow behavior within the framework of weak viscid-inviscid interaction. However, when the angle of attack is increased, separation usually occurs which significantly alters the inviscid flow field and the effective way to deal with this latter situation is to carry out a large-scale numerical calculation on the basis of the governing equations.

The problem of solving non-linear partial differential equations is simplified to that of solving a set of linear algebraic equations. The solution of linear algebraic equations can be found by different methods. Gaussian elimination method can be used to solve such problems. Incompressible viscous flow phenomena arise in numerous disciplines in science and engineering. The governing equations consist in this case of the incompressible Navier-Stokes equations, and the equation of continuity. Most flows in nature and technological devices are turbulent. The transition from laminar to turbulent flow is governed by the Reynolds number, \( \text{Re} = \frac{U C}{\nu} \), where \( U \) is a characteristic velocity of the flow, and \( C \) is a characteristic length (e. g the chord of the airfoil), \( \nu \) is a kinematics viscosity. The basic Navier-stokes equations describe both laminar and turbulent flow [1].

Many numerical methods can be used to solve the equations of incompressible fluid, such as finite difference method and finite element method. Several advantageous properties of the F.E.M. have contributed to its extensive use. Irregularly shaped boundaries can be approximated using elements with straight sides or matched exactly using elements.
with curved boundaries. The number of the elements can be varied. This property allows the element grid to be expanded or refined as the need exists. Boundary conditions such as discontinuous surface loading present no difficulties for the method. Mixed boundary conditions can be easily handled [2]. Therefore, F.E.M. is preferred in applications.

The two-dimensional viscous subsonic flow field around an airfoil for determination of the shapes of the flow at different angles of attack is computed. Incompressible and turbulent flow, steady state is assumed. The governing equations of the fluid flow can be formulated with finite element method to get the solution of the unknown variables (velocity and pressure) around these types of the airfoils throughout the domain $\Omega$. Tinsley J. et al. [3], derived general finite element models of compressible and incompressible fluid flow. These involve local approximations of the velocity field, the density, and the temperature for compressible fluids as well as the velocity, temperature, and pressure for incompressible fluids. Kadhim S. J. Al-Tornachi et al. [4], developed a numerical method to analyze steady incompressible flow around airfoils and wings with turbulent separation. The method combines the classical Prandtl lifting-line theory with a two-dimensional airfoil method, which includes boundary layer calculation and separation model. The model used is a direct viscous-inviscid interaction scheme based on a vortex panel method. Some example calculations were discussed, and a good agreement between calculations and experimental results was obtained. Ahmed [5], studied two-dimensional laminar flow over airfoil NACA 2412 with different angles of attack. Stream function vorticity method was used in body fitted coordinates to overcome the complicated shape of the airfoil. Different Reynolds was tested from 500 to 2000. Full Navier-Stokes laminar flow and predicting the onset of separation at the airfoil surface and circulation, which is caused by the high angle of attack. Angle of attack was increased gradually from zero to 50 degrees in a step of 10 degrees. The separation onset seems to appear between 10 and 20 degrees.

2. The mathematical model:

A set of mathematical equations governing the flow of a viscous fluid was developed well-known Navier-Stokes equations. These equations, in their most general form, together with the equation of conservation of mass (continuity equation) are purported to interconnect the pertinent dependent variables, which describe the flow of a viscous fluid. Since that time the basic form of the equations has remained unchanged even when turbulence phenomena are introduced. Indeed, most present day developments in research where turbulence is significant are based on the Navier-Stokes equations [8].

These equations are expressed in terms of partial differential equations in fluid dynamics [9].

2.1 Continuity Equation
(or Conservation of Mass)

For an incompressible fluid, ($\rho = $ constant), and steady state flow, then:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \quad \text{...(1)}$$

or

$$\nabla \cdot \vec{V} = 0. \quad \text{...(2)}$$

This is called the continuity equation of steady state incompressible flow [6], [7], [8].

2.2 Conservation of Linear Momentum
(or Newton’s second law of motion)

The equation for the conservation of linear momentum is obtained by applying Newton’s second law: The net force acting on a fluid particle is equal to the
time rate of change of the linear momentum of the fluid particle.
The general steady state equations of motion for the x– and y–directions form of the Navier-Stokes equations assuming constant viscosity is \[8\],

\[
\begin{align*}
    u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} &= \frac{1}{\rho} \frac{\partial p}{\partial x} - \frac{1}{\rho} \frac{\partial \rho}{\partial x} + v \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right) \ldots (3) \\
    u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} &= \frac{1}{\rho} \frac{\partial p}{\partial y} - \frac{1}{\rho} \frac{\partial \rho}{\partial y} + v \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right) \ldots (4)
\end{align*}
\]

these are the Navier-Stokes equations for a viscous an incompressible fluid which are general and apply to turbulent as well as non-turbulent cases. Along with the equation of continuity, it is the basic equations, which govern the flow of Newtonian fluids. Which furnishes three equations for \( u, v, \) and \( p \). In which,

\[
v = \frac{\mu}{\rho} \ldots (5)
\]
is called the kinematics viscosity. For Newtonian fluids, the kinematics viscosity is considered constant.

**3. Finite Element Method**

The solution of the set of equations for viscous incompressible flow is attempted using the F. E. M. In this method, a continuum is divided into many small elements of convenient shapes (namely triangular, quadrilateral, tetrahedral, hexahedral, etc.) choosing suitable points called “nodes” within the elements. In fluid dynamics the weighted residual method is often considered the most convenient tool for formulating finite element models. Because it is distributed the error over the domain is in a way that produces an average error equal to zero in order to obtain the linearly independent equation of the shape function [10].

The governing equations of viscous incompressible flow can be expressed as functions of velocities and pressure [11].

Galerkin’s method is a means of obtaining an approximate solution to a differential equation. It does this by requiring that the error between the true solution and the approximate solution is orthogonal to the functions used in the approximation [2].

**3.1 ANSYS Package.**

The ANSYS package 3rd Edition Release 5.4 September 1997 developed by SWANSON ANALYSIS SYSTEM INC. It has many finite element analysis capabilities, ranging from a simple, linear, static analysis to a complex, nonlinear, transient, dynamic analysis.

**3.2 Discretization of the Domain**

The discretization of the domain into sub-regions is the first of a series of the steps that must be performed when solving a problem. The flow domain is to subdivide into a number of sub-domains called elements as in Figure(1). Each element is associated with a number of discrete points or nodes located within or on the element boundary. Fundamentally, the number of elements must be increased in the region where accurate results are desired and reduced in regions where results are not so important [2], [8]. The type of finite element to be introduced in this work is a quadrilateral element for two-dimensional problems.

**3.3. Airfoil specifications**

An airfoil profile NACA 632415 would be used[12]. The subsonic flow is considered in a (2.74) meters high duct consisting an airfoil located at (3.05) meters from the inlet and outlet flow which have (1.16), (1.4163) and (1.73) meters for chord lines [12]. The distance from the trailing edges of airfoil to outlet flow equals (3.05) meters as shown in Figure(2), to achieve the fully developed flow. The direction of flow is from left to right and rotation of the airfoil varies the angle of attack.

The velocity component \( u \) of 28.45 m/s is applied to the inlet flow. The component \( v \) is set to zero m/s, and the outlet boundary
condition is the relative pressure $P$ set to zero Pa. the airfoil could be placed in a symmetrical position with respect to the duct wall. No-slip conditions were assumed on the duct wall and the airfoil surfaces. Hence the velocity at the duct walls and the airfoil surfaces are zero m/s, the dynamic viscosity (in 20°) equals $1.8E-5$ Pa. s and the mass density equals 1.2 kg/m³. In order to calculate the unknown variables near the airfoil, a fine mapped mesh (using linear quadrilateral isoparametric element) must be selected as shown in Figure(3). The grids represent an attempt to achieve the best compromise between the desired flow detail near the airfoil and the specified conditions at the far away straight edges of the duct. The finite boundaries of the duct are sufficiently far away from the airfoil to approximate the free field conditions.

4. Results

The ANSYS results presentations offer fantastic graphical layouts both by vector and contour plots. The results for determination the flow around the airfoil depends on the angle of attack ($\alpha$) as parameter. Different angles of attack are taken, 2, 8 and 13 degrees, at free stream Reynolds number $2.2E+6$.

The maximum value of the pressure at the lower surface is more than that at the upper surface of the airfoils; thus lift is created. The amount depends on the angle of attack. The chord line of the airfoil is 1.4163 meters. The maximum thickness is 15 percent of the chord and it is located at the wing middle approximately. Incidence the subsonic flow around this airfoil is introduced through rotating the surface of the airfoil to vary the angle of attack.

Figure(4) represents the shape of the velocity vectors as quantity and direction of the flow field around the airfoil. Figure(4.a) shows that at an angle of attack of 2 degrees, there are a stagnation point and no circulation within velocity vector around the airfoil. The flow moves smoothly around the airfoil and is attached around the surface as shown in the streamline picture. The distribution of the velocity vectors in the uniform direction indicates a good view of the velocity distribution around the airfoil.

Figure(4.b) shows that at an angle of attack that equals 8 degrees, the stagnation point appears in the lower surface, while it appears in the trailing edge of the airfoil at an angle of attack that equals 13 degrees as shown in Figure(4.c). This indicates that the Kutta condition is imposed. In general the velocity increases above the upper surface with increasing angle of attack from 2 degrees to 13 degrees.

Figure(4) shows the velocity in contour type around isothermal airfoil at angles of attack, 2, 8 and 13 degrees. The result of the velocity distribution shows that the maximum velocity at the upper surface of the airfoil is increasing and the contour appears at the upper surface in different shapes as the angle of attack is increasing. The stagnation point is very clear at lower surface of the airfoil as shown in Figure(5.b). The stagnation point was observed at the lower surface and the trailing edge of the airfoil at an angle of attack that equals 13 degrees as shown in Figure(5.c).

Figure(6) shows the change in pressure distribution around the airfoil with changing angles of attack of 2, 8 and 13 degrees. The pressure value at an angle of attack that equals 8 degrees is greater than the pressure value at an angle of attack that equals 2 and 13 degrees. The maximum value of the pressure is at the lower surface of the airfoil. This implies lifting will be created as expected.
5. REFERENCES


تحديد الضغط والسرعة للجريان حول مطيار من جناح طائرة باستخدام
طريقة العناصر المحدودة

الدكتور المهندس أنور جعفر محمد جواد*

الدكتور علي يوسف عبد الله

ملخص البحث

تم إنجاز دراسة عدبة بعدين لجريان مستقر تحت صوتي للهواء حول مطيار من جناح طائرة، موضوعة في
مجرى هوائي منتظم والذي يعتبر المجال الفزيائي للمساعدة في دراسة المعادلات الأساسية الحاكمة للجريان قد أعيد
بنائها من الإحداثيات الكرزية (x, y) إلى الإحداثيات المحلية (η, ζ). هذه المعادلات هي معادلات الحركة لتأثير
(Navier-Stokes equations) يستخدم عناصر خطيّة (Continuity equation) رياضية (Isoparametric) واستخدمت صيغة المتغيرات الأساسيّة (Finite Elements) في حلها. تم تطغية مجال واسع
لزوايا الهجوم وكانت النتائج على شكل مموجات ضغط وسرعة والتي تُعبر عن تيار الهواء حول المقاطع
المختلفة للجناح. ظهرت نقطة الركود والدورة على سطوح المقاطع، وحدث الدوران عادة نتيجة لزوايا الهجوم العالية
نسبةً وتأثيرات الزوايا. لقد تم اختيار مدار زوايا الهجوم عند إقلاع الطائرة لتكون 2 و 13 درجة، وقد ظهر الدوران
عدد 13 درجة على سطح أحد المقاطع.

(ANSYS Package Release 5.4) لقد تم المحاكاة العدبية باستخدام برنامج العناصر المحدودة المسمى (*
وكان النتائج متوافقة بصورة جيدة.

* د. أنور جعفر محمد جواد ....

- دكتوراه في الرياضيات التطبيقية
- عضو هيئة تدريسية في جامعة البصيرة
- عنوان السكن: سوريا/ حمص / قرب الملعب البلدي
- هاتف الأرضي: 09546994967 / جوال: 0312474495