International Journal of Scientific & Engineering Research, Volume 8, Issue 2, February-2017
ISSN 2229-5518

A study to understand differential equations applied to aerodynamics using CFD Technique

Zoya. A. Rizvi

Abstract— The purpose of this paper is to study the methodology involved in Computational Fluid Dynamics (CFD) technique of fluid flow.Computational fluid dynamic results are directly analogous to wind tunnel results obtained in a laboratory they both represent sets of data for given flow configuration at different Mach numbers, Reynold numbers, etc. The cornerstone of computational fluid dynamics is the fundamental governing equations of fluid dynamics i.e. the continuity, momentum and energy equations. These equations speak physics. The purpose of this study is to discuss these equations for aerodynamics. In CFD the fundamental physical principles applied to a model of flow can be expressed in terms of basic mathematical equation which are either Integral equation or partial differential equation. In the world of CFD, the various forms of the equations are of vital interest. In turn, it is important to derive these equations in order to point out their differences and similarities, and to reflect on possible implications in their application to CFD. An example of application of CFD to aerodynamics is discussed in this paper. Finally, post-processing and interpreting the result completes the methodology.

Index Terms— Partial Differential equations, Fluid flow, Finite control volume, Substantial derivative, Aerodynamics, CFD , Navier-stokes equations

1 INTRODUCTION

COMPUTATIONAL fluid dynamics (CFD) is the branch of fluid dynamics providing a cost-effective means of simulating real flows by the numerical solution of the governing equations.The governing equations for Newtonian fluid dynamics, namely the Navier- Stokes equations, have been known for over 150 years. However, the development of reduced forms of these equations is still an active area of research, in particular, the turbulent closure problem of Reynolds averaged Navier-Stokes equations [2].

2 FLUID FLOW EQUATIONS

In obtaining the basic equations of fluid motion, the following philosophy is always followed:

(1) Choose the appropriate fundamental physical principles from the laws of physics, such as

(a) Mass is conserved

(b) F = ma (Newtons 2nd Law).

(c) Energy is conserved.

(2) Apply these physical principles to a suitable model of the flow.

(3) From this application, extract the mathematical equations which embody such physical principles.

A solid body is rather easy to see and define; on the other hand, a fluid is a squishy substance that is hard to grab hold of. If a solid body is in translational motion, the velocity of each part of the body is the same; on the other hand, if a fluid is in motion the velocity may be different at each location in the fluid. How then do we visualize a moving fluid so as to apply to it the fundamental physical principles?

For a continumm fluid, the answer is to construct the following models:

1. Finite Control Volume & Infinitesimal Fluid Element
2. Substantial Derivative & Physical meaning of ∇.V
3. Navier-Stokes Equation

2.1 Finite Control Volume & Infinitesimal Fluid Element

Consider a general flow field as represented by the streamlines[1] in Fig. 1. The control volume (C.V) is represented by V and control surface is represented by S. The C.V may be fixed in space with the fluid moving through it as shown in Fig. 1a. Alternatively the C.V may be moving with the fluid such that the same fluid particles are always inside it as in Fig. 1b.

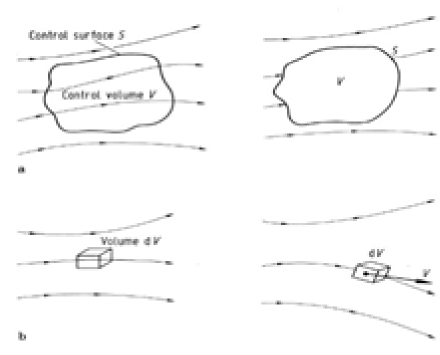


Fig. 2. Fluid element moving in the flow fieldillustration for thesubstantial derivative

The equations are obtained from the finite control volume fixed in space in either integral or partial differential form and are called the conservation form of the governing equations. The equations obtained from the finite control volume moving with the fluid in either integral or partial differential form are

- *Zoya A Rizvi is currently working as Assistant Professor in MPSTME(Mechanical Department), NMIMS University, Vile Parle, Mumbai, Maharashtra, India, PH – 02249330534. E-mail: zoya.rizvi@nmims.edu* called the non-conservation form of the governing equations.

Let us imagine an infinitesimally small element in the flow, with a differential volume, dV. The fluid may be fixed in space with the fluid moving through it and alternatively it may be moving along a streamline with a vector velocity v equal to the flow of velocity at each point. Again, instead of looking at the whole flow field at once the fundamental equation are applied to just the fluid element itself. This application leads directly to the fundamental equations in partial differential equation form. So in general we will be dealing with conservation type of partial differential equations in this study.

2.2. Substantial Derivative & physical meaning of ∇ V

Before deriving the governing equations, we need to establish a notation which is common in aerodynamics that of the substantial derivative[2]. Let us consider a small fluid element moving with the flow as shown in Fig. 2.

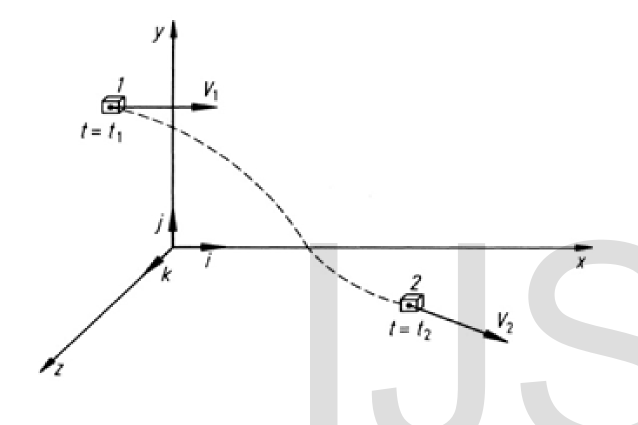


Fig. 2. Fluid element moving in the flow fieldillustration for thesubstantial derivative

Here, the fluid element is moving through Cartesian space. The unit vectors along x,y and z axes are i , j and k respectively. The vector velocity field in this Cartesian space is given by equation (1)

$$\vec{V} = u\vec{\imath} + v\vec{\jmath} + w\vec{k} \tag{1}$$

Note that we are considering in general an unsteady flow, where u, v, and w are functions of both space and time, t. In addition, the scalar density field is given by

$$\rho = \rho(x, y, z, t)$$

Furthermore, in Cartesian coordinate the vector operator is defined as in equation (2)

$$\nabla = \vec{\imath}\,\frac{\partial}{\partial x} + \vec{\jmath}\frac{\partial}{\partial y} + \vec{k}\,\frac{\partial}{\partial z} \tag{2}$$

Taylor series expansion from point 1 to point 2 can be given as in equation (3)

$$\rho_2 = \rho_1 + \left(\frac{\partial \rho}{\partial x}\right)_1 (x_2 - x_1) + \left(\frac{\partial \rho}{\partial y}\right)_1 (y_2 - y_1) + \left(\frac{\partial \rho}{\partial z}\right)_1 (z_2 - z_1) + \left(\frac{\partial \rho}{\partial t}\right)_1 (t_2 - t_1) + higher\ order\ terms \tag{3}$$

Dividing by $t_2 - t_1$ and ignoring higher order terms we get

$$\frac{\rho_2 - \rho_1}{t_2 - t_1} = \left(\frac{\partial \rho}{\partial x}\right)_1 \frac{(x_2 - x_1)}{t_2 - t_1} + \left(\frac{\partial \rho}{\partial y}\right)_1 \frac{(y_2 - y_1)}{t_2 - t_1} + \left(\frac{\partial \rho}{\partial z}\right)_1 \frac{(z_2 - z_1)}{t_2 - t_1} + \left(\frac{\partial \rho}{\partial t}\right)_1 \tag{4}$$

The left side of equation (4) is the average time-rate of change in density from point 1 to point 2. By taking limits as t_2 approaches t_1 this term becomes,

$$\lim_{t_1 \to t_2} \frac{\rho_2 - \rho_1}{t_2 - t_1} = \frac{\partial \rho}{\partial t} \tag{5}$$

If we take the derivative of equation (4) we get the following :

$$\frac{D\rho}{Dt} = u\frac{\partial \rho}{\partial x} + v\frac{\partial \rho}{\partial y} + w\,\frac{\partial \rho}{\partial z} + \frac{\partial \rho}{\partial t} \tag{6}$$

The above equation can be written in cartesian cordinate form which is the substantial derivative form:

$$\frac{D}{Dt} = u\frac{\partial}{\partial x} + v\frac{\partial}{\partial y} + w\,\frac{\partial}{\partial z} + \frac{\partial}{\partial t} \tag{7}$$

From equation (2) & (7) we get the substantial derivative

$$\frac{D}{Dt} = \frac{\partial}{\partial t} + \vec{V}.\nabla \tag{8}$$

We once again we once again emphasize that D/Dt is the substantial derivative, which is physically the time rate of change following a moving fluid element; $\frac{\partial}{\partial t}$ is called the local derivative, which is physically the time rate of change at a fixed point; $\vec{V}.\nabla$ is called the convective derivative, which is physically the time rate of change due to the movement of the fluid element from one location to another in the flow field where the flow properties are spatially different. The substantial derivative applies to any flow-field variable, for example, Dp/Dt, DT/Dt, Du/Dt, etc., where p and T are the static pressure and temperature respectively[3]. For example:

$$\frac{DT}{Dt} = \frac{\partial T}{\partial t} + \vec{V}.\nabla \equiv \frac{\partial T}{\partial t} + u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y} + w\,\frac{\partial T}{\partial z} \tag{9}$$

2.3 Navier-Stokes Equation

The Navier-Stokes equations, developed by Claude-Louis Navier and George Gabriel Stokes in 1822, are equations which can be used to determine the velocity vector feild that applies to a fluid given some initial conditions[3]. They arise from the application of Newton's second law in combination with a fluid stress (due to viscosity) and a pressure term.

$$\rho\left[\frac{du}{dt} + u\nabla u\right] = \nabla\sigma + f \tag{10}$$

where ρ denotes the density of the fluid and is equivalent to mass, $du/dt+u\cdot\nabla u$ is the acceleration and u is velocity, and $\nabla\cdot\sigma+f$ is the total force, with $\nabla\cdot\sigma$ being the shear stress and f being all other forces. We may also write this as

$$\rho[du/dt+u.\nabla u] = -\nabla p + \mu\,\nabla 2u + f \tag{11}$$

The term on the LHS are often referred as inertial terms, and arise from the momentum changes. These are countered by the pressure gradient, viscous forces and body forces. μ is dynamic viscosity.

The Navier–Stokes equations dictate not position but rather velocity. A solution of the Navier–Stokes equations is called a velocity field or flow field, which is a description of the velocity of the fluid at a given point in space and time as in equation (6), if velocity is considered in place of density.

Finally, by dividing out ρ and subtracting $u{\cdot}\nabla u$, we obtain the traditional form of the Navier-Stokes equation below

$$\frac{du}{dt} = -(u.\nabla)u - \frac{1}{\rho}\nabla p + \gamma\nabla^2 u + f \quad (12)$$

3 CFD Equations

Computational fluid dynamics (CFD) is a branch of fluid mechanics that uses numerical analysis and algorithms to solve and analyze problems that involve fluid flows. The fundamental basis of almost all CFD problems is the Navier–Stokes equations. The Navier-Stokes equations consists of a time-dependent continuity equation for conservation of mass, three time-dependent conservation of momentum equations and a time-dependent conservation of energy equation. There are four independent variables in the problem, the x, y, and z spatial coordinates of some domain, and the time t. There are six dependent variables; the pressure p, density r, and temperature T (which is contained in the energy equation through the total energy Et) and three components of the velocity vector; the u component is in the x direction, the v component is in the y direction, and the w component is in the z direction, All of the dependent variables are functions of all four independent variables. The differential equations are therefore partial differential equations and not the ordinary differential equations that you study in a beginning calculus class.
You will notice that the differential symbol is different than the usual "d /dt" or "d /dx" that you see for ordinary differential equations. The symbol "partial" is is used to indicate partial derivatives. The symbol indicates that we are to hold all of the independent variables fixed, except the variable next to symbol, when computing a derivative. The set of equations are[6]:

Continuity:

$$\frac{\partial\rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0$$

X-Momentum

$$\frac{\partial(\rho u)}{\partial t} + \frac{\partial(\rho u^2)}{\partial x} + \frac{\partial(\rho uv)}{\partial y} + \frac{\partial(\rho uw)}{\partial z} = -\frac{\partial p}{\partial x} + \frac{1}{Re_r}\left[\frac{\partial\tau_{xx}}{\partial x} + \frac{\partial\tau_{xy}}{\partial y} + \frac{\partial\tau_{xz}}{\partial z}\right]$$

Y-Momentum

$$\frac{\partial(\rho v)}{\partial t} + \frac{\partial(\rho uv)}{\partial x} + \frac{\partial(\rho v^2)}{\partial y} + \frac{\partial(\rho vw)}{\partial z} = -\frac{\partial p}{\partial y} + \frac{1}{Re_r}\left[\frac{\partial\tau_{xy}}{\partial x} + \frac{\partial\tau_{yy}}{\partial y} + \frac{\partial\tau_{yz}}{\partial z}\right]$$

Z-Momentum

$$\frac{\partial(\rho w)}{\partial t} + \frac{\partial(\rho uw)}{\partial x} + \frac{\partial(\rho vw)}{\partial y} + \frac{\partial(\rho w^2)}{\partial z} = -\frac{\partial p}{\partial z} + \frac{1}{Re_r}\left[\frac{\partial\tau_{xz}}{\partial x} + \frac{\partial\tau_{yz}}{\partial y} + \frac{\partial\tau_{zz}}{\partial z}\right]$$

Energy:

$$\frac{\partial(E_T)}{\partial t} + \frac{\partial(uE_T)}{\partial x} + \frac{\partial(vE_T)}{\partial y} + \frac{\partial(wE_T)}{\partial z} = -\frac{\partial(up)}{\partial x} - \frac{\partial(vp)}{\partial y} - \frac{\partial(wp)}{\partial z} - \frac{1}{Re_r Pr_r}\left[\frac{\partial q_x}{\partial x} + \frac{\partial q_y}{\partial y} + \frac{\partial q_z}{\partial z}\right]$$
$$+ \frac{1}{Re_r}\left[\frac{\partial}{\partial x}(u\,\tau_{xx} + v\,\tau_{xy} + w\,\tau_{xz}) + \frac{\partial}{\partial y}(u\,\tau_{xy} + v\,\tau_{yy} + w\,\tau_{yz}) + \frac{\partial}{\partial z}(u\,\tau_{xz} + v\,\tau_{yz} + w\,\tau_{zz})\right]$$

where Re is the Reynolds number which is a similarity parameter that is the ratio of the scaling of the inertia of the flow to the viscous forces in the flow. The q variables are the heat flux components and Pr is the Prandtl number which is a similarity parameter that is the ratio of the viscous stresses to the thermal stresses. The τ variables are components of the stress tensor. A tensor is generated when you multiply two vectors in a certain way. Our velocity vector has three components; the stress tensor has nine components. Each component of the stress tensor is itself a second derivative of the velocity components.

The terms on the left hand side of the momentum equations are called the convection terms of the equations. Convection is a physical process that occurs in a flow of gas in which some property is transported by the ordered motion of the flow. The terms on the right hand side of the momentum equations that are multiplied by the inverse Reynolds number are called the diffusion terms. Diffusion is a physical process that occurs in a flow of gas in which some property is transported by the random motion of the molecules of the gas. Diffusion is related to the stress tensor and to the viscosity of the gas. Turbulence, and the generation of boundary layers, are the result of diffusion in the flow. The Euler equations contain only the convection terms of the Navier-Stokes equations and can not, therefore, model boundary layers. There is a special simplification of the Navier-Stokes equations that describe boundary layer flows.
Notice that all of the dependent variables appear in each equation. To solve a flow problem, you have to solve all five equations simultaneously; that is why we call this a coupled system of equations. There are actually some other equation that are required to solve this system. We only show five equations for six unknowns. An equation of state relates the pressure, temperature, and density of the gas. And we need to specify all of the terms of the stress tensor. In CFD the stress tensor terms are often approximated by a turbulence model[7].

3.1 CFD Simulation

CFD simulates fluid (either liquid or gas) passing through or around an object. The analysis can be very complex—for example, containing in one calculation heat transfer, mixing, and unsteady and compressible flows. The ability to predict the impact of such flows on your product performance is time consuming and costly without some form of simulation tool.
The simulation is done in three steps[5]:

1. Preprocessing : Define geometry
2. Solver: Once the problem is set-up defining the boundary conditions we solve it with the software on the computer.
3. Postprocessing: Once we get the results as values at our probe points we analyse them by means of color plots, contour plots, appropriate graphical representations can generate reports.

3.1.1 Airfoil Analysis using ANSYS

Three airfoil shapes with high camber shown in Fig. 3 were chosen from UIUC website [4] for designing wing of a UAV(Subsonic). A preliminary parameter study was performed manually to find the most suitable chord length for obtaining high L/D ratio.

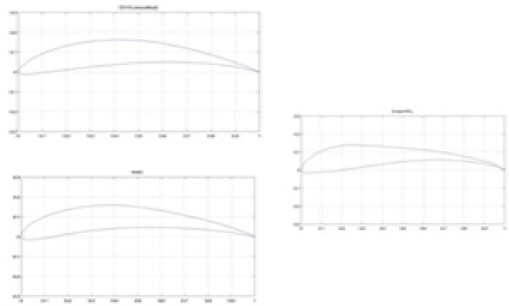


Fig. 3 Airfoil shapes. a) CH10 , b) E243 c) S1223

A grid was generated as shown in Fig. 4. for processing in solver. The various boundary conditions were analysed before solving like Inlet, Outlet and walls for input to the solver.

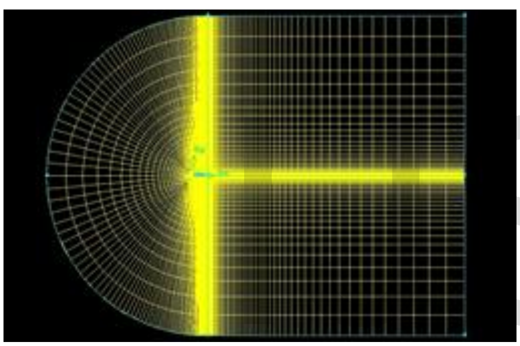


Fig. 4.Airfoil Grid generated in Gambit for processing in ANSYS

The analysis gave the following results as shown in Fig5. Below:

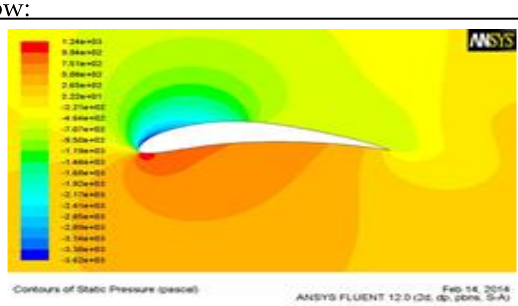


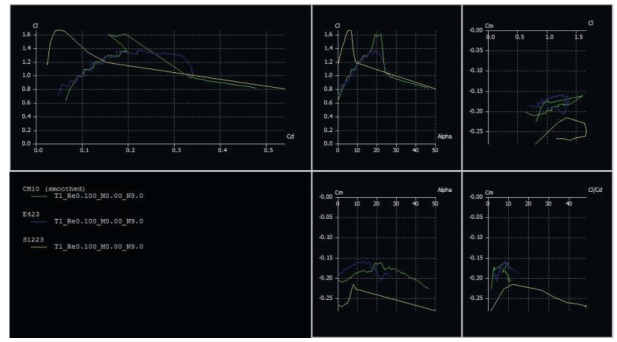
Fig. 5.CFD Analysis showing pressure contour for airfoil E423

A similar analysis as shown in Fig. 5 were performed on CH10 and S1223 and its results were interpreted which showed that best L/D ratio was given by E423 and hence it was chosen for the final design of UAV.

Airfoil	L/D Ratio
S1223	5.128
CH10	6.29
E423	7.26

3.1.2 Airfoil Analysis using XFLR

For validation of results a another analysis technique was used to compare the results. This technique is software based called XFLR Analysis. It is highly used software in aerodynamic airfoil designing.



As one can interpret from the above graph that E423 is giving the most stable CL/CD ratio. A better L/D ratio leads to better fuel economy, climb performance and glide ratio. These factors are important to obtain high aerodynamic efficiency.

4 Conclusion

The fluid flow equations of aerodynamics which form the basis of CFD were discussed in this paper. Navier stokes equation was derived by considering a small fluid element for a unsteady flow and its various parts were discussed for interpretation of its physical meaning in real life. Finally after interpreting the meaning of fluid flow equations an experimentation was conducted on airfoil for analyzing its Lift in ANSYS software and its results were obtained by postprocessing which were compared to XFLR results for validation. The comparison has shown that E423 airfoil gives the desired high L/D ratio and hence it was chosen for final design of wing of the UAV.

References

[1] John D. Anderson. JR., W. Daly Patrick, "Computational Fluid Dynamics The Basics with Applications", published by McGraw-Hill Inc (1995), ISBN 0-07-113210-4.

[2] Praveen Kumar kanti, Shyam chandran , "A review paper on basics of CFD and its applications", National conference on advances in mechanical engineering science(NCAMES-2016).

[3] Anderson, John D., Jr., "Fundamentals of Aerodynamics", 2nd Edition McGraw-Hill,New York, 1991.

[4] Prof. Michael Selig, David Ledniser (1995). UIUC Airfoil Coordinates Database, http://aerospace.illinois.edu/m-selig/ads/coorddatabase html.

[5] Tutorial written by Rajesh Bhaskaran, Flow over airfoil : Gambit CFD Processing, Aerodynamic Tutorial from BARC LAB

[6] Navier-stoke-equations, https://www.grc.nasa.gov/www/k12/airplane/nseqs.html

[7] Abdul Nasser Sayma "Computatinal Fluid Dynamics", Ventus Pub lishing, 2009, ISBN – 978-87-7681-430-4