

The use of computational fluid dynamics applications to various flow problems

Adrian Bogdan Şimon-Marinică^{1}, Vlad Mihai Păsculescu¹, Florin Manea¹, and Zoltan Vass¹*

¹National Institute for Research and Development in Mine Safety and Protection to Explosion - INSEMEX, Petrosani, Romania.

Abstract. To deal with any such applications, it is a must to understand the basic form and nature of the governing equations of fluid dynamics. It is imperious to fully understand the basic of numerical discretization that can be applied in equations. This paper aims to present some CFD tools that are the starting point in solving problems related to the field of fluid dynamics. During this work, we will note that anyone CFD technique will not be appropriate for all problems and the diverse mathematical nature of partial differential equations will ensure that some algorithms will best work for hyperbolic equations and others will do best for elliptic equations. In addition, this paper examines precisely how CFD techniques can be used to solve various flow problems. In other words, CFD applications requires the simultaneous knowledge of some major aspects, such as the governing flow equations and their mathematical behavior, aspects of numerical discretization of partial differential equations, also known as finite differences or of integral equations, known as finite volumes. Computational fluid dynamics has a major impact on airplane design and soon to be a critical technology for aerodynamic design with the purpose to enhance the design process for any machine that deals with fluid flow.

1 Introduction

Computational fluid dynamic results are directly analogous to wind tunnel results obtained in a laboratory. They both represent sets of data for given flow configurations at different Mach numbers, Reynolds numbers, etc. However, unlike a wind tunnel, which is generally a heavy, unwieldy device, a computer program (say in the form of USB device) is something you can carry around in your hand. Or better yet, a source program in the memory of a given computer can be accessed remotely by people on terminals that can be thousands of miles away from the computer itself. A computer program is, therefore, a readily transportable tool, a "transportable wind tunnel". Carrying this analogy further, a computer program is a tool with which you can carry out numerical experiments [1]. For example, assume that you have a program which calculates the viscous, subsonic, compressible flow over an airfoil. Such a computer program was developed to solves the complete two-dimensional Navier - Stokes equations for viscous flow by means of a finite-

* Corresponding author: bogdan.simon@insemex.ro

difference numerical technique. Now, if we have such a program, we can carry out some interesting experiments with it, experiments which in every sense of the word are analogous to those you could carry out (in principle) in a wind tunnel, except the experiments that we perform with the computer program are numerical experiments, as we can see the example in the pictures below (figs. 1 and 2). For the figure 1 we have instantaneous streamlines over a airfoil for laminar flow. The laminar flow is unsteady. For the picture 2 the streamlines are over the same airfoil with the same conditions except that the flow is turbulent.

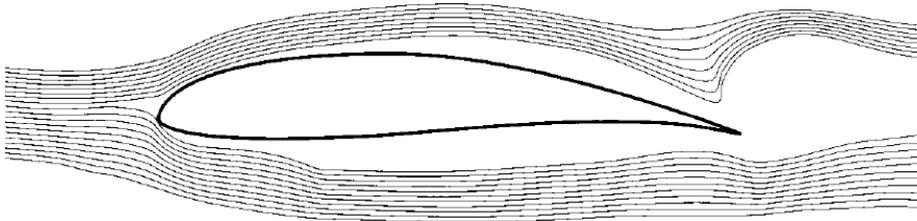


Fig. 1. Laminar flow.

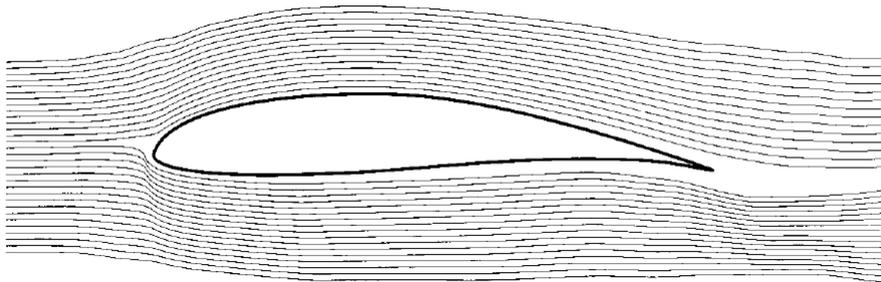


Fig. 2. Turbulent flow.

The numerical technique used to calculate these flows is a time-marching method, using a time-accurate finite-difference solution of the unsteady Navier-Stokes equations. The streamlines shown in figure 1) are simply a snapshot of this unsteady flow at a given instant in time. In contrast, the figure 2) shows the calculated streamlines when a turbulence model is "powered on" within the computer program. As we can see, the calculated turbulent flow is attached flow; moreover, the resulting flow is steady. If we compare the figures above, we can see that the laminar and turbulent flows are quite different; moreover, this CFD numerical experiment give us the possibility to study in detail the physical differences between the laminar and turbulent flows, all other parameters being equal, in a fashion impossible to obtain in an actual laboratory experiment.

2 The blunt nosed body

Let's consider the flow field over a blunt-nosed body moving at supersonic or hypersonic speeds, as sketched in the figure below, fig. 3. The interest in such bodies is driven by the fact that aerodynamic heating to the nose is considerably reduced for blunt bodies compared to sharp-nosed bodies; this is one of the reasons why the space shuttle has a blunt nose and wings with blunt leading edges.

As shown in fig. 3, there is a strong, curved bow shock wave which sits in front of the blunt nose, detached from the nose by the distance, called the shock detachment distance. We can analyze the sketch below and observe that the region of flow behind the nearly normal portion of the shock wave, in the vicinity of the centerline, is locally subsonic, whereas further downstream, behind the weaker, more oblique part of the bow shock, the flow is locally supersonic. The dividing line between the subsonic and supersonic regions is called the sonic line, if the flow is assumed to be invisible, neglecting the dissipative transport processes of viscosity and thermal conduction, the governing flow equations are the Euler. Although these equations are the same no matter whether the flow is locally subsonic or supersonic, their mathematical behavior is different in the two regions.

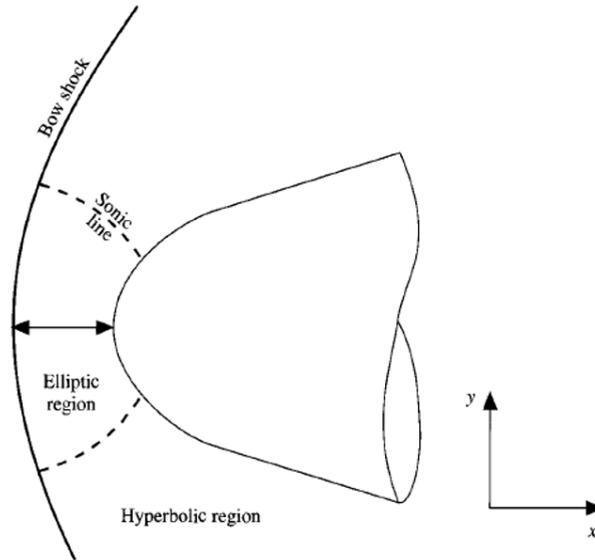


Fig. 3. The flow field over a supersonic blunt-nosed body.

In the steady subsonic region, the Euler equations exhibit a behavior that is associated with elliptic partial differential equations, whereas in the steady supersonic region, the mathematical behavior of the Euler equations is totally different, namely, that of hyperbolic partial differential equations. The change in the mathematical behavior of the governing equations from elliptic in the subsonic region to hyperbolic in the supersonic region made a consistent mathematical analysis which included both regions virtually impossible. Using the developing power of CFD and with a concept of a time dependent approach to the steady state a numerical, finite-difference solution to the supersonic blunt body problem which constituted the first practical engineering solution for this flow was obtained. Here we find an example of the power of CFD combined with an algorithm which properly considers the mathematical behavior of the governing flow equations.

3 Equations of fluid dynamics

All the CFD [2], in one form or another, is based on the fundamental governing equations of fluid dynamics the continuity, momentum, and energy equations. These equations speak physics. They are the mathematical statements of three fundamental physical principles upon which all of fluid dynamics is based:

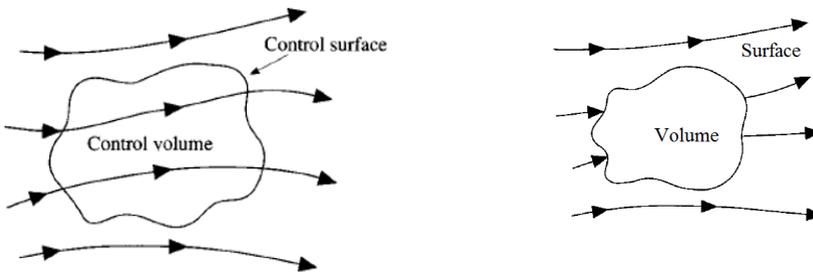
- a) Mass is conserved
- b) Newtons second law, $F = ma$

c) Energy is conserved

The governing equations can be obtained in various forms. For most aerodynamic theory, the form of the equations makes little difference. However, for a given algorithm in CFD, the use of the equations in one form may lead to success, whereas the use of an alternate form may result in oscillations in the numerical results, incorrect results, or even instability. Therefore, in the world of CFD, the various forms of the equations are of vital interest. In tum, it is important to derive these equations to point out their differences and similarities, and to reflect on possible implications in their application to CFD. 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 or hold. 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 do we visualize a moving fluid to apply to it the fundamental physical principles?

The finite control volume model:

We consider a general flow field as represented by streamlines in the figure 4 below. We need to imagine a closed volume drawn within a finite region of the flow. This volume defines a control volume, a control surface defined as the closed surface which bounds the volume. The control volume may be fixed in space with the fluid moving through it. Alternatively, the control volume may be moving with the fluid such that the same fluid particles are always inside it. In either case, the control volume is a reasonably large, finite region of the flow. The fundamental physical principles are applied to the fluid inside the control volume and to the fluid crossing the control surface (if the control volume is fixed in space). Therefore, instead of looking at the whole flow field at once, with the control volume model we limit our attention to just the fluid in the finite region of the volume itself. The fluid-flow equations that we directly obtain by applying the fundamental physical principles to a finite control volume are in integral form.



a) Finite control volume fixed in space with the fluid moving through it.

b) Finite control volume moving with the fluid such that the same fluid particles are always in the same control volume.

Fig. 4. Model of a flow, a) finite control volume approach and b) infinitesimal fluid element approach.

These integral forms of the governing equations can be manipulated to indirectly obtain partial differential equations. The equations so obtained from the finite control volume fixed in space (fig. 4.a), in either integral or partial differential form, are called the conservation form of the governing equations. The equations obtained from the finite control volume moving with the fluid (fig. 4.b), in either integral or partial differential form, are called the non-conservation form of the governing equations.

4 Computer graphic technique used in CFD

Computer graphic techniques are frequently used in the presentation of CFD data and are used as an essential tool by the computational fluid dynamicist to display the results of a CFD calculation. The computational fluid dynamicist usually implements various modes of graphical representation using existing computer graphic software [3], rather than developing the details of new computer graphic programs himself or herself. It is generally not the purview of CFD to be involved with the development details of computer graphic software but rather to simply use this software as a tool. There are many existing software packages used by computational fluid dynamicists today. Tecplot, a software package is used. This is not to be construed as an endorsement of a specific product but rather simply as an example of a standard graphics software approach. Most ways that CFD results that are presented graphically can be classified under six general categories.

4.1 XY Plots

On a two-dimensional graph, they represent the variation of one dependent variable versus another independent variable. XY plots are the simplest and most straightforward category of computer graphical representation of CFD results. Although such graphs are not particularly sophisticated, they remain the most precise quantitative way to present numerical data on a graph; that is, another person can readily read quantitative data from curves on an XY plot without making any mental or arithmetic interpolation.

4.2 Contour Plots

A disadvantage of XY plots as described above is that they usually do not illustrate the global nature of a set of CFD results all in one view. On the other hand, contour plots do provide such a global view. A contour line is a line along which some property is constant. Each line of a contour plot corresponds to a constant value of pressure coefficient. Generally, contours are plotted such that the difference between the quantitative value of the dependent variable from one contour line to an adjacent contour line is held constant. It is clear from examining these contour plots that the global nature of the flow is seen in one single view. To obtain the same global feeling as in the case of XY plots, a short example is the locations of the shock and expansion waves, we would have to examine several XY plots (fig. 5). Contour plots are clearly a superior graphical representation from this point of view. On the other hand, it requires more effort to read precise quantitative data from a contour plot as compared to a curve in an XY plot. Although each contour may be labeled as to the constant numerical value of the property it represents, the obtaining of numerical values between contour lines requires some mental and/or numerical interpolation in space, an imprecise process to say the least. We need to visualize some examples of contour plots from some modern CFD applications, pointing out various nuances and subcategories. We must take in place that these are contour plots of the transverse velocity (the y component of velocity) in the flow field behind a detonation wave propagating through a combustible mixture of H₂, O₂, and argon. Combustion of the hydrogen and oxygen occurs behind the detonation front. Because of the physical presence of slight disturbances in the flow behind the front, the flow field becomes two-dimensional, with transverse waves, along with various slip lines.

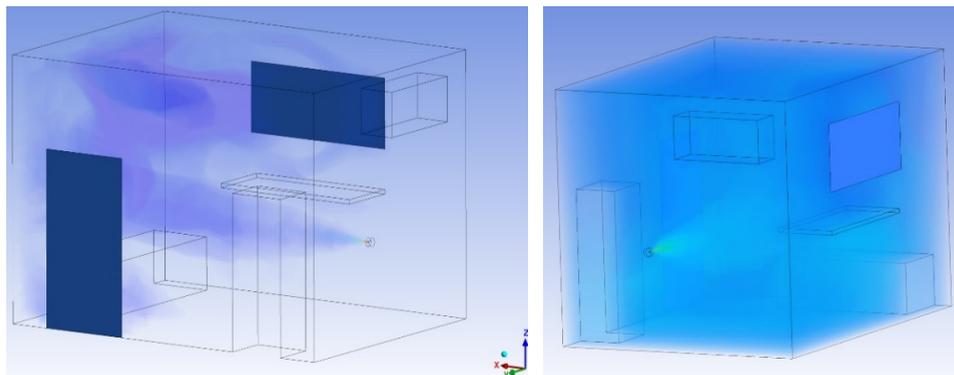


Fig. 5. Velocity contour plot of a gas leak inside a room at a pressure of 24.64 PSI.

5 Conclusions

So, in conclusions, at the end of this paper, we did go through the basics of the numerics necessary for the numerical solution of the governing flow equations and understand that the fundamental aspects of numerical discretization can be put together to form various techniques for the numerical solution of the continuity, momentum, and energy equations. We can conclude that CFD is a growth industry [4], with an unlimited number of new applications and new ideas just waiting in the future. If in the elaborate formulas after some physical experiments were enrolled many empirical factors, the field of computational simulations using CFD techniques will require an increase in these factors. Aware that these factors need to be changed from a case to another, or even over a period within the same simulation, the manufacturers of commercial or less commercial computational fluid dynamics applications, have left within reach of users the possibility to customize a large number of variables, the possibility to define these variables as functions, expressions, or even modifying entire parts of the software by using languages specific to those applications, or dedicated programming languages.

This paper was developed within the Nucleu-Programme, carried out with the support of Romanian Ministry of Research, Innovation and Digitization, project no. PN-19-21-01-05, project title: Fundamental research and computer simulations on the initiation of explosive gas mixtures by potential sources of ignition of a different nature.

References

1. A.B. Simon-Marinica, N.I. Vlasin, F. Manea, G.D. Florea, Finite element method to solve engineering problems using Ansys, *The 9th edition of the International Multidisciplinary Symposium "SIMPRO 2021"*; *MATEC Web of Conferences* **342**, 01015 (2021)
2. C. Tomescu, D. Cioclea, I. Gherghe, E. Chiuzan, M. Morar M., Determination of danger, risk and fire vulnerability parameters. Numerical simulation in fire extinction, *The 8th edition of the International Multidisciplinary Symposium "UNIVERSITARIA SIMPRO 2021"*: Quality-Access to Success, **20** (S1), 55-60 (2019)
3. Z. Vass, A.B. Simon-Marinica, S. Stanila, Defining parameters for numerical modelling of fire, *9th International Symposium on Occupational Health and Safety (SESAM 2019)*, *MATEC Web Conf.* **305**, 00042 (2020)
4. G.D. Florea, N.I. Vlasin, Locating the probable ignition source in fire expertise, *9th International Symposium on Occupational Health and Safety (SESAM 2019)*, *MATEC Web Conf.* **305**, 00042 (2020)