DEVELOPMENT OF AN ANALYSIS METHODOLOGY OF AERODYNAMIC PROFILES APPLIED TO DESIGN OF UNMANNED AERIAL VEHICLES

Paulo Marçal Fernandes Filho, paulofilho_fernandes@hotmail.com
Lucas da Silva Vieira, lucasvieira10@hotmail.com
Welberth Douglas Pereira do Nascimento, welberthdouglas@gmail.com
Luciano Golçalves Noleto, lucianonoleto@unb.br
Manuel Barcelos, manuelbarcelos@gmail.com
University of Brasília, Campus Universitário Darcy Ribeiro, Gleba A, Bloco G, Faculdade de Tecnologia.

Abstract. *This work has as goal the development of a methodology to analyze and determine aerodynamic parameters for flow over different airfoils. Numerical simulation by using CFD programs of commercial and opensource origin are executed for the sake of comparison, and also to create robust and reliable tools. These tools have great applicability to the design of small model aircrafts and unmanned aerial vehicles (UAVs), and also to the traditional aeronautic industry. The proposed study consists of building meshes to analyze airfoils by using the exporting module of the ANSYS ICEM-CFD program tailored for different CFD codes, and being the simulation of those airfoils executed by three distinct programs: ANSYS CFX, OpenFOAM and CORDEL. The whole methodology proposed in this work considers laminar flows of low Reynolds numbers. The choice of the ANSYS CFX is due its efficiency and reliability as a commercial code already tested by the market. The OpenFOAM is chosen as CFD tool because it has an open source code of great acceptance in the academic field, becoming possible its modification and adaptation to meet specific characteristics of each simulation. As for the CORDEL, its choice is due to the need to analyze its robustness and reliability, once this is developed by researchers of the Mechanical Engineering Department of the University of Brasilia. The code is written in MatLab language and uses the finite element projection method to solve the Navier-Stokes equations coupled to several turbulence models. The results obtained by the simulation in this work are still compared to experimental and numerical data presented in the literature in order to verify their quality.*

Keywords*: finite elements, finite volume, low Reynolds, UAV, Airfoil*

1. INTRODUCTION

The design concept of an unmanned aerial vehicle was first developed in the American Civil War and latter was also applied in World War II by Japan and United States. Since then, this kind of tool has been improved, being employed not only for military or commercial applications, but also for the public and government sectors in air traffic control, agriculture, safety, among others (LIST,2003). It is expected that the worldwide market of UAV's will reach the mark of US$ 13.6 billion by 2014 (Forecast,2005).

Several projects in the aeronautical field including radio controlled aircraft models and more specifically UAV's have a growing demand for computational determination of aerodynamic parameters, such as lift, drag and pressure coefficients. The computational simulation is necessary for the reduction of wind tunnel use and destructive tests, avoiding the waste of time and money.

The search for computational models that better represent the reality is already a demanding task in the academic community. Several works aim to compare CFD packages on simulation of airfoils (Naveen and Vidyadhar, 2010), especially high-lift airfoils, once it is observed an increasing application of these kind of airfoils, in both aeronautical and aerospace industry, as for designs of UAV's (Ribeiro,2002).

The main goal of this work is to compare CFD packages and to create a reliable set of tools for the determination of airfoil aerodynamic characteristics. The study, here proposed, intend to construct different meshes in the ICEM-CFD software for simulation of flow over airfoils by using three CFD tools: CFX, OpenFOAM and CORDEL. These tools will be used to make the process of obtaining the aerodynamic parameters more convenient and fast, as well as more reliable.

For this work, the choice of the CFX software had many reasons. The first and most important was the great confidence that this software has achieved in the field of commercial CFD tools. Another motivation was a good user interface, allowing simulations in different conditions without major difficulties. Also, CFX employs a finite volume numerical method for the discretization and solution of physical problem, which provides greater accuracy to the results obtained by the numerical simulation (Assumpção et al, 2010).

CORDEL is a finite element based program developed by researchers from the Department of Mechanical Engineering of the University of Brasília. The program is written in MATLAB language, a widely used language in engineering. CORDEL uses a projection method with possibility of coupling different turbulence models. Works by Noleto *et al*. (2009) and Pereira (2004) showed results obtained by CORDEL for several simulations of laminar flows on different geometries, as well as of turbulent flows over airfoils.

An open-source CFD package was chosen as a viable alternative for commercial codes. There are some packages available; among them, the most complete choice was OpenFOAM, which consists of a robust and reliable CFD package of interest in academia. Moreover, its source code is open and developed in C++ language which allows for a better understanding of what is behind the modeling and numerical implementation. The OpenFOAM is already employed successfully in simulations of flows over airfoils (Naveen and Vidyadhar, 2010).

In summary, this work describes the numerical simulation of three different airfoils under laminar flow and with low Reynolds number. The CFD software ANSYS CFX, OpenFOAM and CORDEL were used in the simulations. Aerodynamic coefficients were computed and compared to experimental and numerical data available in the literature.

2. METHODOLOGY

2.1. ANSYS CFX, OpenFOAM and CORDEL

After the mesh construction, the CFX software prepares, executes and analyzes a simulation in three main steps. The first step is the CFX-Pre where the mesh is loaded and the boundary conditions and material properties are defined. At this point, one can define the flow velocity for the three directions of the coordinate system. Therefore, the change of the angle of attack to match the characteristics of different simulations is achieved without major difficulties.

The second step is the CFX-Solver which is an equation solver responsible for solving different types of ordinary and partial differential equations. For instance, in this step the partial differential equations based on conservation of mass and momentum are integrated over a region of interest. By employing the method of finite volumes these differential equations are converted to an algebraic linear system of equations which is solved by an iterative process (ANSYS CFX, 2009).

The CFX-Post is the last step where the results are post processed and the flow field and its characteristics are visualized. At this process, graphics of the velocity and pressure fields are plotted, and forces that act over the airfoil are integrated, and then aerodynamic coefficients are computed.

The software CORDEL solves the Navier-Stokes equations by using a finite element discretization based on the CBS (Characteristic-Based Split) method, Pereira e Júnior (2005). For the discretization and solution of the CFD problems, the CORDEL code employs two-dimensional triangular mesh elements.

The CBS methodology is basically divided three steps. Initially, the problem of convection-diffusion is solved explicitly for the velocity field. Later, the algorithm solves the pressure field implicitly, and finally it corrects the velocity field computed in the first step using the new pressure field. If the time step employed is short enough this solution methodology is stable.

The main advantage of the CBS algorithm is use of only one mesh for pressure and velocity discretization. This is possible because the method applies the same order of interpolation for pressure and velocity. Another advantage is the reduction of computational cost, due to the use of a semi implicit numerical scheme which results in solving only a system of linear equations per iteration of the pressure and velocity coupled problem solution.

The OpenFOAM software is basically a set of C++ libraries that uses the finite volume method for solving coupled partial differential equations. Among its features are the simulation of flows over 3D geometries, the use of unstructured meshes with an arbitrary number of faces and the selection of specific libraries for each flow condition (Souza et al, 2009). The simulations in OpenFOAM are performed by executable files called solvers, which are appropriately selected according to the characteristics of the flow and geometry which allow for the solution of specific problems of continuum mechanics. OpenFOAM offers, through specific settings, the option of using two-dimensional meshes. Thus, the software proves to be versatile and robust because its solution features cover a large set of flow conditions and geometries, ranging from two-dimensional simple problems to three-dimensional more complex ones.

2.2. Mathematical Equations

By considering a time dependent viscous laminar and incompressible flow problem, the flow governing equations are described as:

$$\nabla \, . \, u = 0 \tag{1}$$

$$\frac{\partial^2 u}{\partial t} + (\nabla u)\, u = -\frac{1}{\rho_f} \nabla p + \nabla \, . \, (\nu_f \nabla u) + f_f \tag{2}$$

where $u(x,t)$ e $p(x,t)$, respectively, the velocity and the pressure fields of the flow are function of the mesh position and the time, ρ_f and ν_f are in this order the density and the viscosity of the fluid problem, and f_f is a given force function. The fluid boundary conditions are defined on $\Gamma_f = \Gamma_d \cup \Gamma_n \cup \Gamma_p$ by:

$$u(x,t) = u_{\Gamma_d} \text{ on } \Gamma_d \tag{3}$$

$$u(x,t) = u_{\Gamma_n} \text{ on } \Gamma_n \tag{4}$$

$$p(x,t) = p_{ref} \text{ on } \Gamma_p \tag{5}$$

The subset Γ_d represents the boundary values that are constant where the Dirichlet boundary condition for the velocity field u_d is prescribed. The subset Γ_n represents the boundary values where the Neumann boundary condition for the velocity field u_n is prescribed. Γ_p is the outlet boundary condition where a reference pressure p_{ref} is prescribed.

2.3. Geometry and Boundary Conditions

Three airfoils were chosen to compose base of the data comparison, they are: Naca0012, Selig1223 and Eppler423. The Naca0012 airfoil was chosen due to the large amount of experimental and numerical flow data available. The last two airfoils were chosen due to their extensive application to the design of radio controlled aircraft models and UAV's, because they are suitable to missions that require high lift at low Reynolds numbers (Ribeiro,2002). Each airfoil has a chord of 1 m, and their geometries are presented in the Figure 1.

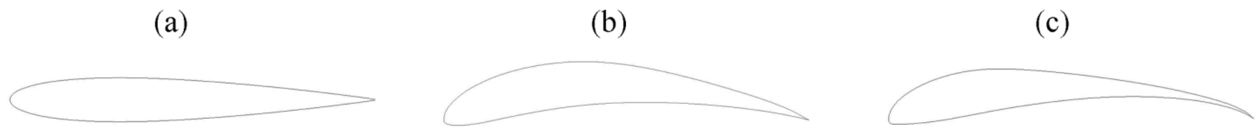

Figure 1. (a) Naca0012, (b) Eppler423 and (c) Selig1223.

The domain that encloses each airfoil was devised based on their geometry. The domain size is defined by the length of the airfoil chord. After, the domain mesh was built using the software ICEM-CFD. The boundary conditions applied to the domain limits are: "no slip wall' on the border of the airfoil and "inlet or outlet" on the frontier of domain. A "symmetry" boundary condition has to be applied to the lateral surface if a quasi-2D model is employed instead of a 2D model. A quasi-2D model is used when the numerical solver does not have 2D special discretization capabilities, the case of the ANSYS CFX software. The quasi-2D model is obtained extruding the symmetry surface over the perpendicular coordinate, and one layer of 3D elements is use to discretize the resulting gap. A 2D representation of the domain and its boundary conditions are showed in the Figure 2.

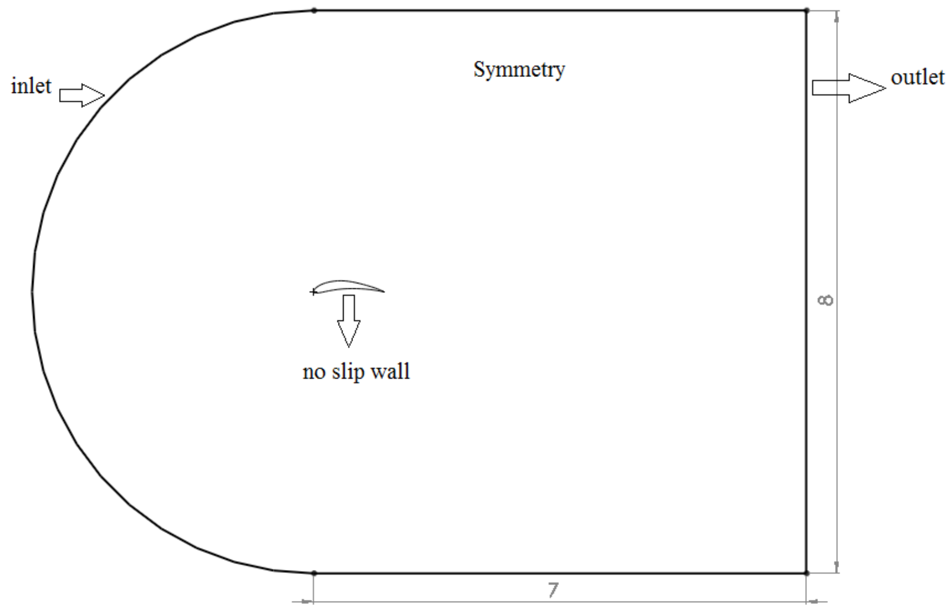

Figure 2. Domain and boundary conditions – xy plane

In the case of the OpenFOAM software, only a quasi-2D model of Naca0012 airfoil was employed and the mesh used has one layer of 43,066 hexahedrons elements and 21,812 nodes. The OpenFOAM software works with 3D meshes, but it also has the possibility of using two-dimensional meshes, being only necessary for this case to set a specific control flag. The mesh used in the OpenFOAM software was automatically generated by a mesh tool which is part of its package. The automatic mesh generation capability allowed the study of the flow simulation over several slice thicknesses, where the best results were obtained over a thickness of 0.05m. The mesh used in the OpenFOAM software was not generated using the ICEM-CFD software as the ones employed by the CFX software and the CORDEL code, mainly because of to the lack of a reliable conversion procedure to the format the is read by the OpenFOAM software.

For the CORDEL code only a 2D mesh of the Naca0012 airfoil with 28,574 triangular elements and 14,995 nodes was used. Due to the impossibility of solving 2D models in the CFX software, it was necessary to develop for every airfoil test case a quasi-2D model extruding a 2D symmetry mesh over the z direction, resulting into a 3D slice geometry with unit thickness and discretized by one layer of prismatic elements. Also, because of the several steps necessary for the procedure of mesh generation in the ICEM-CFD software and to facilitate the computation of aerodynamic parameters in the post-processing tool of the CFX software, a unit thickness was chosen for the quasi-2D models employed.

The boundary conditions applied to the numerical simulations fit the common UAV mission requirements. At the end of every simulation, the flow results were post-processed and aerodynamic parameters were calculated and presented in tables and graphics. For the sake of comparison, curves of lift versus different angles of attack obtained over different airfoils and software are analyzed. Also, the pressure fields of different airfoils at certain angles of attack are commented. Besides, experimental data were used to validate some numerical results.

3. RESULTS

For every test case, a density value of 1.225 kg/m^3 and a dynamic viscosity value of 10^{-5} kg/ms were at the free stream. In addition, a laminar, isothermal and incompressible fluid model was used to simulate the flow over the airfoils.

3.1. Test Case 1

The airfoil studied in this case was the Epperl423 for a flow with Reynolds number equal to 299,500. The mesh used has 18,221 elements and 18,584 nodes (Fig. 3). The maximum number of iterations was set to 10,000 and the residual target for the continuity equation was set 10^{-6}. The goal is to compare the results obtained by the CFX software and the experimental data available in the literature (UIUC, 1997).

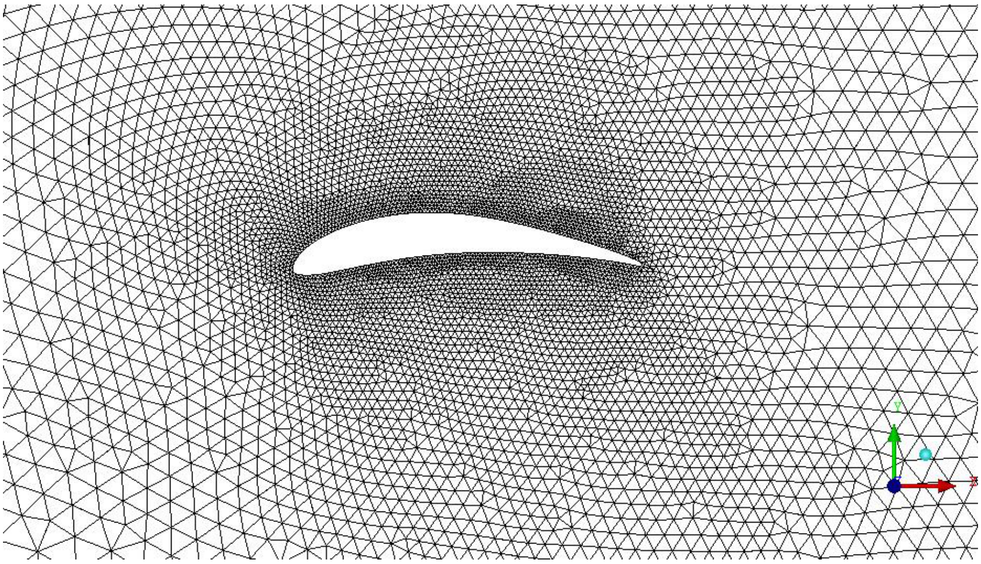

Figure 3. Mesh for the Eppler423 airfoil.

The lift coefficients of the airfoil Eppler423 were computed for five angles of attack by postprocessing the flow results obtained with the CFX software and compared to experimental lift coefficients, as showed in the Table 1. Also, by taking advantage of the same set of data, it was built the graphic of the lift coefficients versus the angles of attack, as showed in the Fig. 4.

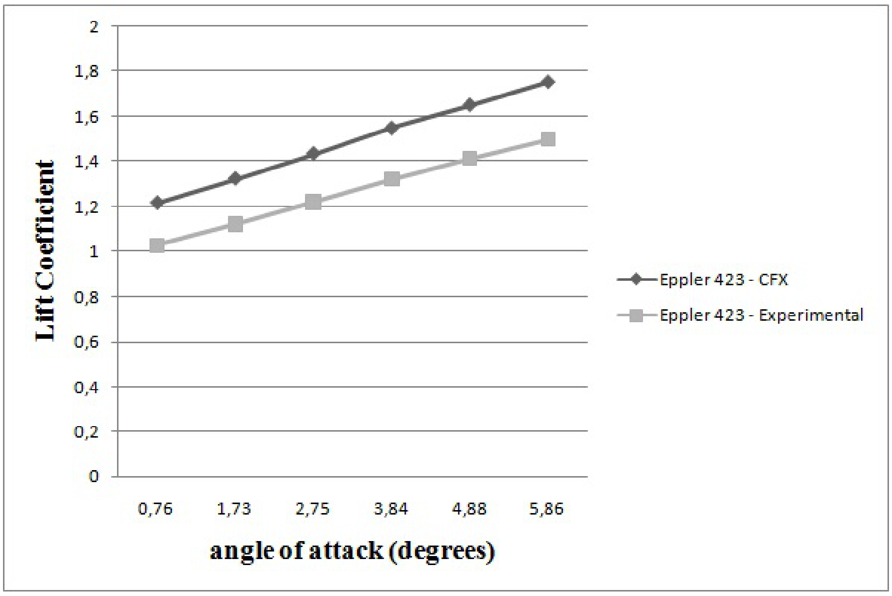

Figure 4. Lift curve for the Eppler423 airfoil.

The difference between the two curves presented in graphic of the Figure 4 is explained in part by the model chosen. The difference between the experimental and the numerical data happens because the laminar flow model with Reynolds of the order of 10^5 does not fully represent the reality. Turbulent structures are neglected in a laminar model with a high Reynolds number, and if they were taken into account, they could better represent the reality. Further, despite the efforts of having a refined mesh over the airfoil, the phenomena that happen in the boundary layer are not completely determined because of the lack of an appropriate mesh layer structure.

Table 1. Lift coefficients for the Eppler423 airfoil.

Angle of Attack	Lift Coefficient	
Alpha (degrees)	CFX	Experimental
0.76	1.216	1.027
1.73	1.324	1.121
2.75	1.435	1.219
3.84	1.548	1.319
4.88	1.652	1.413
5.86	1.754	1.497

The postprocessed flow fields provided by the CFX software are quite satisfactory when someone looks at the properties of the simulated laminar flow from an aerodynamic point of view. The pressure field is showed in the Fig. 5. One can see that even for alpha equal to zero, there is a difference of pressure between the upper and lower surfaces, an expected characteristic of an asymmetrical profile.

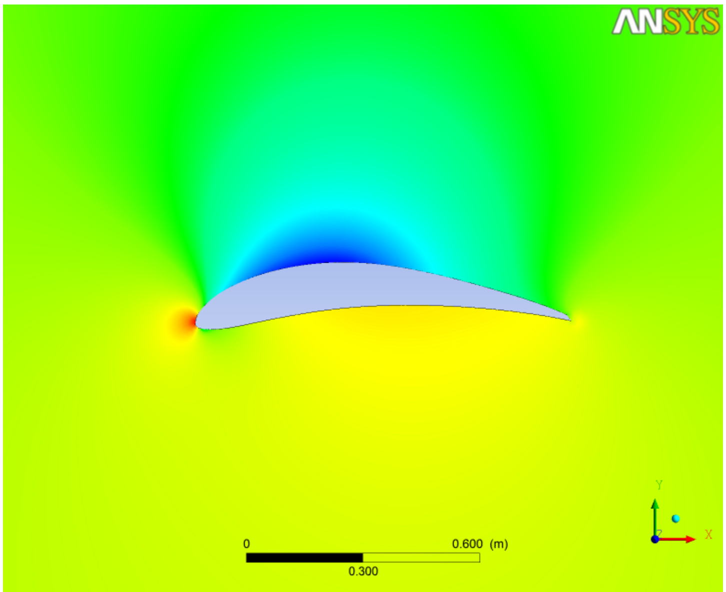

Figure 5. Pressure field for the Eppler423 at zero angle of attack.

3.2. Test Case 2

In this case, the profile chosen was the Selig1223 for a flow with Reynolds number of 302,200. The mesh used has 19,173 elements and 19,532 nodes (Fig. 6). The maximum number of iterations was set to 10,000 and the residual target for the continuity equation was set to 10^{-6}. Similarly to the test case 1, the goal is to compare the results obtained by the CFX software and the experimental data available in the literature (UIUC, 1997).

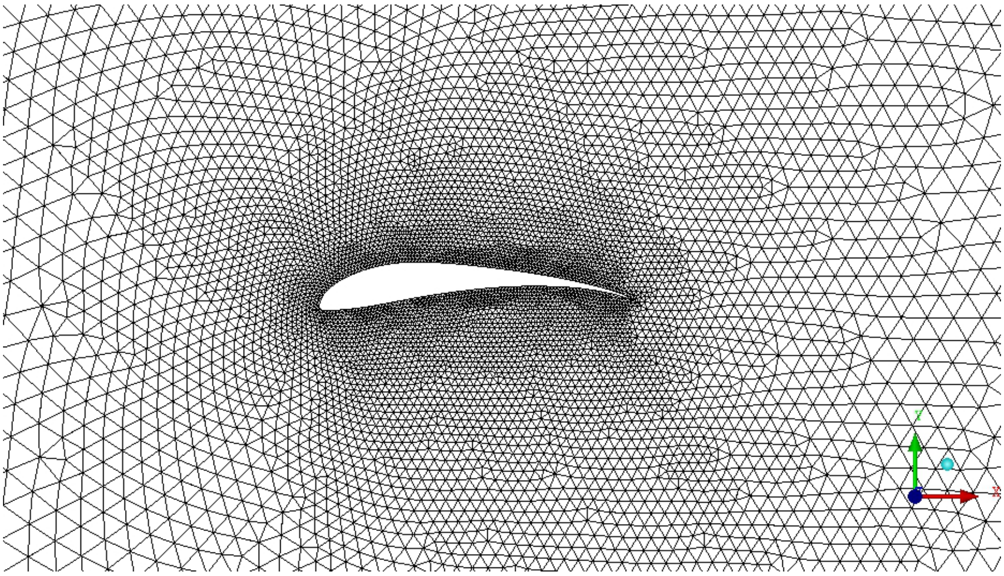

Figure 6. Mesh for the Selig1223 airfoil.

The test case now considered is the Selig1223. Again, the figures were built by using the postprocessing package of the CFX software and the lift coefficients for six different angles of attack were compared to experimental data, as showed in the Table 2. From the data of the Table 2, the graphic of the lift coefficients versus the angles of attack was plotted, as showed in the Fig. 7.

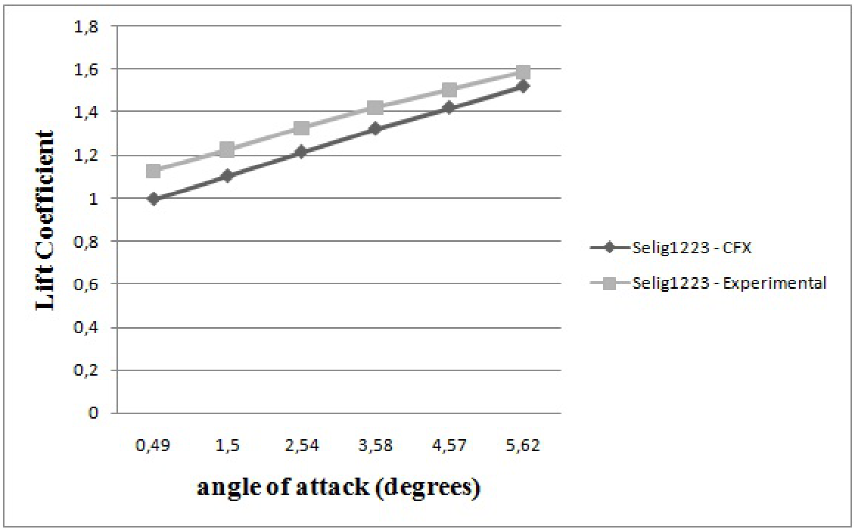

Figure 7. Lift curve for the Selig1223 airfoil.

In the Figure 7, there is also a consistent gap, though less pronounced, between the curve of the data extracted from the CFX and the curve of experimental data. Possible reasons are also that the laminar model does not represent well the real flow on an airfoil with the Reynolds number of 302,200, and the lack of a proper boundary layer mesh structure.

Table 2. Lift coefficient for the Selig1223 airfoil.

Angle of Attack	Lift Coefficient	
Alpha (degrees)	CFX	Experimental
0.49	0.994	1.129
1.50	1.102	1.227
2.54	1.215	1.327
3.58	1.322	1.424
4.57	1.421	1.506
5.62	1.520	1.588

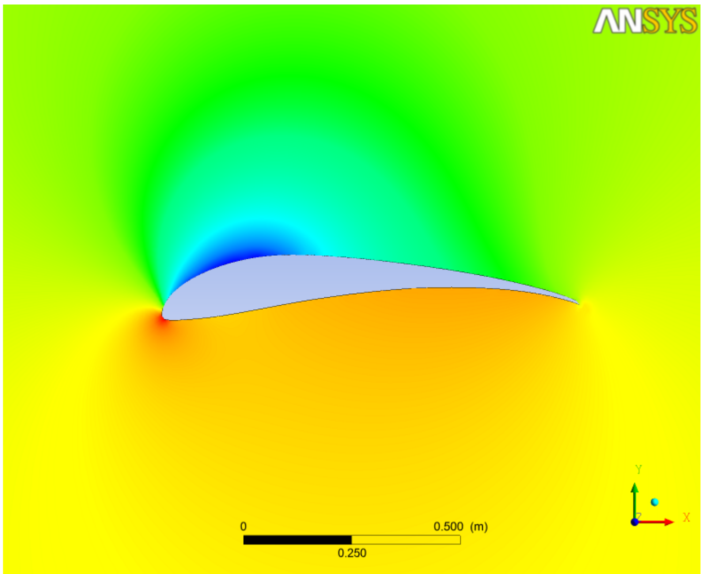

Figure 8. Pressure field for the Selig1223 airfoil at 5.62 angle of attack.

The Figure 8 represents the pressure field for Selig1223 airfoil at an angle of attack equal to 5.62.

3.3. Test Case 3

In this case the NACA0012 airfoil was simulated in the CFX software and in the CORDEL code. For the CFX software, the mesh used has 28,579 elements and 28,998 nodes. The convergence strategy is identical to the one employed in the test cases 1 and 2. The mesh used for the COEDEL code has 28,574 elements and 14,995 nodes. In both cases the Reynolds number used was 10^3.

The Reynolds number used in this comparison was kept low due to the long time demanded for the convergence of the CORDEL laminar flow solver for high Reynolds numbers. The graphic of the lift coefficient as a function of angle of attack is presented in the Fig. 10.

.

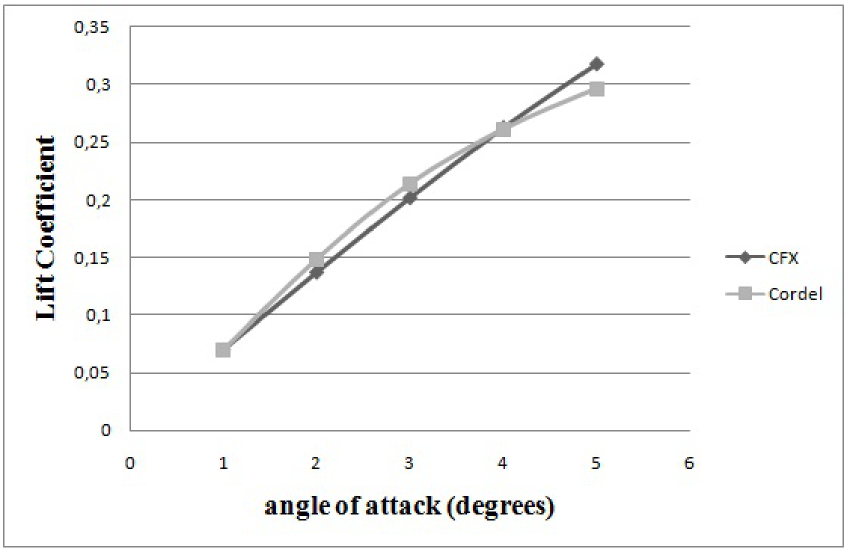

Figure 10. Lift curve for the NACA0012 airfoil with Reynolds number of 10^3.

Despite the Reynolds number shortcomings, the results of the CFX and CORDEL programs are close. From a numerical perspective, these results validate the academic solver and show its potential of use.

3.4. Test Case 4

In this case, the NACA0012 airfoil was simulated in the CFX and OpenFOAM software. For the CFX software, the mesh used has 28,579 elements and 28,998 nodes. The convergence strategy is identical to one of the test cases 1, 2 and 3. For the OpenFOAM software the mesh used has 28,574 elements and 14,995 nodes. In both cases the Reynolds number used was 360,000. The data postprocessed by using results of the simulations are showed in the Table 3.

Table 3. Lift coefficient for the NACA0012 airfoil.

Angle of attack	Lift Coefficient	
Alpha (degrees)	CFX	OpenFOAM
0	0.0709	0.09471
1	0.2018	0.23254
2	0.3355	0.35411
3	0.4651	0.45140
4	0.5940	0.56965
5	0.7163	0.74874

From the data of the Table 3, the graphic of the lift coefficients versus the angles of attack obtained from the simulations of the both software is plotted. The curves are showed in the Fig. 11. The results of the OpenFOAM software were very close to the ones of the CFX software. From a numerical point of view, these results validate both the model and the solvers used in each program.

5. CONCLUSION

The results computed by using the software CFX, OpenFOAM and CORDEL are consistent and satisfactory. Although, not exact, if compared to experimental data, the case of the CFX software. The choice of the laminar flow model over a turbulent one, and the lack of a proper boundary layer mesh lead to a difference between the numerical results and the reality. However, the results also show that from a numerical perspective, for each program, the physics model and solvers were validated. Specially, the CORDEL code proved quite useful regarding the prediction of the airfoil lift coefficient.

The objective of this work was achieved. The study and the development of a solution procedure to tackle problems of low Reynolds number were carried out. Existent numerical tools were verified and their numerical models and solvers were validated. For future work, the implementation of a turbulence model and the improvement of the boundary layer mesh are considered to become the solution procedure as close as possible to the reality.

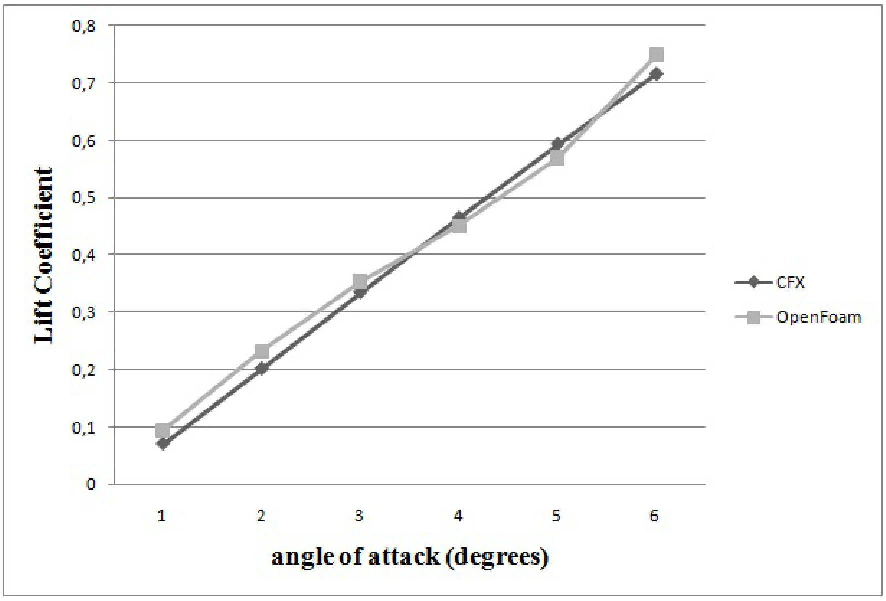

Figure 11. Lift curve for the NACA0012 airfoil with Reynolds number of 360,000.

4. REFERENCES

Ansys CFX, 2009. "CFD Methodology", CFX International, AEA Technology, UK.

Assumpção, M.E., Souza, D.A.C., Junior, L.A., César, L.M., Cristoforo, A.L, 2010 "Comparação Entre O Coeficiente De Arrasto Da Asa De Uma Aeronave Rádio-Controlada Através Dos Métodos Dos Volumes Finitos E Multhopp", I Encontro Regional De Matemática Aplicada eComputacional,Minas Gerais, Brasil.

Forecast International,2005 "UAV Market to Top $13 Billion by 2014" Forecast International. 21 Oct 2005 <Http://Www.Forecastinternational.Com/Press/Release.Cfm?Article=80 (2005)>.

LIST, 2003. "Brief history of UAVs" Laboratory for Information Systems & Telecommunications, 17 Jan 2003 <http://www.list.ufl.edu/uav/UAVHstry.htm>.

Noleto, L.G., Junior, A.C.P. and Júnior, M.N., 2009. "A finite element numerical simulation of the laminar flow over an oscillating airfoil", Proceedings of the 20th Brazilian Congress of Mechanical Engineering, Gramado, Brazil.

Pereira,R.M., 2004. "Simulação numérica de escoamentos laminares e turbulentos através do método de elementos finitoscom estabilização CBS".Dissertação de mestrado em Engenharia Mecânica, Brasília-DF, Brasil.

Pereira, R.M.and Júnior, M.N.D., 2005. "Laminar Flow Numerical Study Using Finite Elements Method And Characteristic-Based Split Algorithm For A Same Pressure And Velocity Interpolation Order", Proceedings of the 18th Brazilian Congress of Mechanical Engineering, OuroPreto, Brazil.

Ribeiro, D.E., 2002. "Simulação Numérica De Aerofólios De Alta Sustentação",IX Congresso Nacional de Estudantes de Engenharia Mecânica, Itajubá, Brasil.

Souza, D.B., Alves, U.F. and Murata, V.V. 2009. "Avaliação Do Escoamento De Fluidos Incompressíveis Em Tubulações Usando CFD". VIII Congresso Brasileiro de Engenharia Química em Iniciação Científica, Uberlandia, Brasil.

UIUC, 1997."UIUC Wind tunnel data on the web" UIUC Applied aerodynamics group.19 Dec.1997 < http://www.ae.illinois.edu/>.

6. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.