# COBEM-2017-2724
# NUMERICAL ANALYSIS OF AN AERODESIGN WING USING VORTEX LATTICE METHOD AND FINITE VOLUME METHOD

**J. Oliveira Junior**
**L. Ferro**
Universidade Federal Rural do Semi-Árido- UFERSA, Centro de Engenharias, Mossoró RN, Brazil
jakson117@live.com
lferro@ufersa.edu.br

***Abstract.*** *A great effort of aircraft industry has been developed to get more efficient and safer aircrafts. More modern aircraft are requested by airlines in all parts of the globe. Manufacturers, in turn, need increasingly skilled aerospace engineers. The main purpose of SAE Brazil Aerodesign competition is to design an efficient small aircraft. Computational Fluid Dynamics (CFD) can be an important tool on the design of aircraft components minimizing design time and costs. The knowledge flow details about an aircraft are necessary for the design of an efficient aircraft. The three-dimensional flow about an aircraft wing can be computed using a CFD code. A design method, however, must be reliable and fast to allow comparing different geometries easily. In this paper, it is proposed the numerical study of the wing of an Aerodesign in order to predict the aerodynamic behaviour of the wing in flight, analysing the main parameters involved by two different methods, the Vortex Lattice Method and the Finite Volume Method. For the Vortex Lattice Method Athena Vortex Lattice (AVL) code was used. The ANSYS CFX code applied for finite volume method analyses. The domain was discretized by an unstructured grid. A shear stress (SST) turbulence model was chosen. Lift and momentum coefficients evolutions versus the angle of attack are presented and compared. Distribution of pressure and velocity on the surface of the wing computed by the ANSYS code are also presented.*

***Keywords:*** *wing, CFD, Finite Volume Method, Vortex Lattice Method, Aerodesign.*

## 1. INTRODUCTION

A great effort of aircraft industry has been developed to get more efficient and safer aircrafts. The knowledge of the details of the flow about the aircraft is necessary for the design of an efficient aircraft. The Computational Fluid Dynamics (CFD) analyses are an essential tool on the design of an aircraft nowadays.

At SAE Brasil Aerodesign competition each team must design an aircraft with a maximum lift and a minimum weight under a set of restrictions (SAE BRASIL, 2016).

Computational Fluid Dynamics analyses are used in aerodynamic by aircraft manufacturers to minimize costs and design time of their aircrafts. The bigger manufacturers started to use CFD in their projects in the late 1970s with the objective to predict the flow behaviours that until then only can be obtained experimentally. These experiences were very expensive and time-consuming (Johnson *et al.* 2003).

The Vortex Lattice Method (VLM) is an easy and fast converging method. It applies the Prandtl theory to panels scattered throughout the wing area. The final result is obtained by integrating an influence of each panel along the span (Anderson, 2011).

The Finite Volume Method solves the Navier-Stokes equations with a turbulence model. RANS models are robust and introduce more accurate analysis than predecessor methods Spalart and Allmaras (1992), Lauder e Spalding (1972), Wilcox (2007) and Menter (1994). The Shear Stress Transport (SST) model developed by Menter (1994) is widely used in aeronautical analysis (Menter, 2009).

At this paper, the Vortex Lattice Method and Finite Volume Model are applied to compute the flow about a rectangular wing used in an aircraft model of the Aerodesign Brasil competition. Comparisons of spanwise distributions of circulation, lift and momentum coefficients, as well as chordwise distribution of pressure coefficients are presented.

## 2. VORTEX LATTICE METHOD

When a wing have a small aspect ratio and sweep angle or dihedral angle are different of zero the Prandtl's classical lifting line theory do not have good results (Anderson, 2011). An extension of lifting line theory that is applicable for wings with any aspect ratio and with sweep or dihedral angle different from zero and to multicomponent lifting system is Vortex Lattice Method (VLM). Some authors also consider VLM as an extension and variation of panel method (Souza, 2007). On the VLM method the mean line surface of the wing is discretized into trapezoidal panels, as done at panel method. Each panel contains a single horse-show vortex generating a mesh of overlapping horseshoe vortices (Bertin and Cummings, 2009). The bound vortex is located usually at 1/4 chord position with two semi-infinite trailing edge vortex lines. For reasons of simplification, the trailing vortices are considered in the direction of the aircraft axis (Bertin and Cummings, 2009).

The velocity induced by the vortex lines are computed by Biot-Savart law (Brederode, 1997). The intensity of the vortex lines is computed prescribing the boundary condition of zero normal velocity at each control point. The control point at each panel usually is positioned at 3/4 of the chord along the mean line. A matrix of influence coefficients, composed by the velocities induce by the vortex lines at all control points is computed. A linear system of equation must be solved to compute the strength of the vortex lines $\Gamma_i$.at each panel. The right hand side member of the linear system of equations is the symmetric value of the normal component of freestream velocity at each control point

Thus, the total lift coefficient is given by the integral over the entire span, in Eq. (1).

$$C_L = \int_0^1 \frac{C_l c}{c_{av}} d\left(\frac{2y}{b}\right) \tag{1}$$

where $b$ is the wing span, $c$ the chord of aerofoil section of the wing and $y$ the coordinate along the span. The $c_{av}$ is the average chord

$$c_{av} = \frac{1}{b}\int_{-b/2}^{b/2} c\,dy$$

The drag coefficient can also be calculated by the relation presented by Multhopp (1950), in Eq. (2) where $\alpha_i$ is given by Eq. (3).

$$C_{Di} = \frac{1}{S}\int_{-b/2}^{b/2} C_l c \alpha_i dy \tag{2}$$

$$\alpha_i = -\frac{1}{8\pi}\int_{-b/2}^{b/2} \frac{C_l c}{(y-\eta)^2} d\eta \tag{3}$$

## 3. COMPUTATIONAL PROCEDURE

In this paper, for the two finite wing analysis models selected, the same Aerodesign wing was simulated. The results of the methods were then compared, also considering the mesh dependence. The machine configurations used in the simulations are presented in Tab. 1.

Table 1. Computer specifications

| Operacional system | Windows 10 Pro 64 bits |
|---|---|
| Processor | Intel® Core™ i5-4670 @ 3.40 GHz |
| RAM | 4,00 GB |
| GPU | Intel® HD Graphics 4600 |
| HD | 500,00 GB |

The wing that was studied is a rectangular wing, with no sweep angle and no dihedral angle. The wing span is equal to $b$=1.89 m and chord length of the wing section is $c$=0.355 m. The wing section is a modified Selig 1223 aerofoil (Fig. 1). For the Vortex Lattice Method wing analysis, an open source code Athena Vortex Lattice (AVL) was used. The ANSYS CFX was used for finite volume analysis.

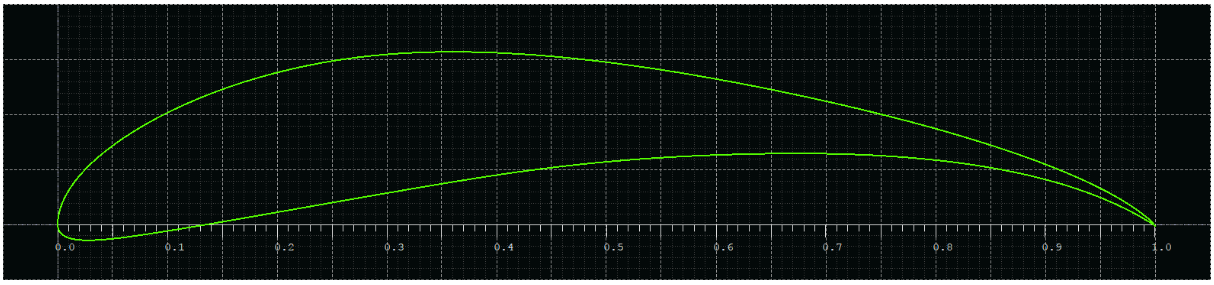


Figure 1. Modified Selig 1223 airfoil

In both analyses, the flow about wing was simulated at predefined attack angles α from 0° to 20 °. The free-stream velocity is $V_\infty = 20$ m/s, density and dynamic viscosity of the air are $\rho = 1.185$ kg/m³ and $\mu = 1.831\times10^{-5}$ Pa·s, respectively. The Reynolds number is Re = $4.6\times10^5$. The AVL software does not perform viscous analyzes, therefore no viscous simulations were performed using the Vortex Lattice Method.

The dimensions of the control volume for the ANSYS CFX analyses are chosen to minimize the influence of external boundary surfaces on the flow close to the wing and at the wake. An elliptical section tunnel (Fig. 2) was chosen to minimize the global number of nodes eliminating unnecessary cells close to the corners that appears at a parallelepipedic control volume. The tunnel length is 6 m and the largest and smallest axes of the ellipse are 6 m and 4 m, respectively.

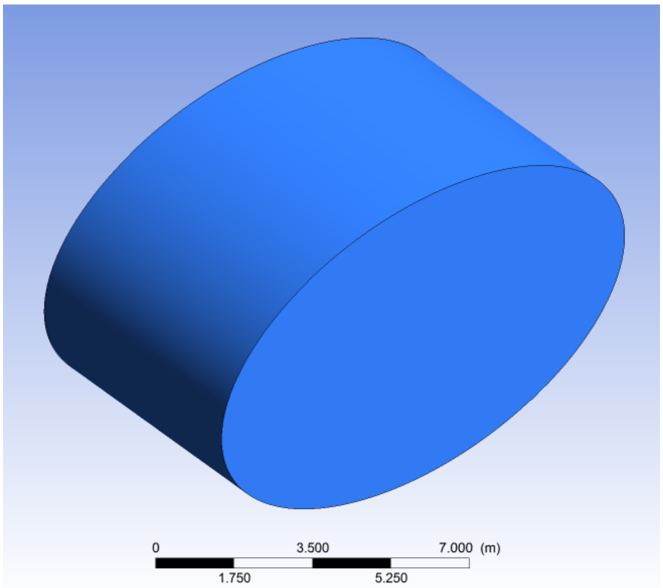


Figure 2. Control volume geometry

It was chosen to use a tetrahedral mesh in the free flow because it is easier to construct, despite the increase in numerical diffusion. Close to the wing, a prismatic mesh was made to improve the calculation of the boundary layer (Fig. 3). A condition of maximum size of element was imposed on the wing wall. This variable parameter was chosen to investigate the dependence of the mesh. Table 2 shows the maximum sizes of elements tested and the number of nodes of the mesh generated.

The mesh of the viscous simulations in the CFX was adjusted in order to approximate the parameter $y^+$ of its ideal value $y^+ \approx 1$ to be used with the SST turbulence model. To do this, the thickness of the first layer of cells close to the surface of the wing should be 1.7 x $10^{-5}$ mm. A twenty-layer inflation condition was applied at wing surface in order to define the thickness of the first layer of elements at the boundary of the wing with a growth rate of 1.1 as recommended by ANSYS (2016).

Table 2. Number of nodes as a function of the maximum size of the wing surface element

| Maximum size of element (m) | Nodes number |
|---|---|

| | |
|---|---|
| $8.5\times 10^{-3}$ | 878625 |
| $8.0 \times 10^{-3}$ | 931027 |
| $7.5\times 10^{-3}$ | 996427 |
| $7.0 \times 10^{-3}$ | 1070034 |
| $6.5 \times 10^{-3}$ | 1165587 |
| $5.5 \text{ x } 10^{-3}$ | 1491838 |

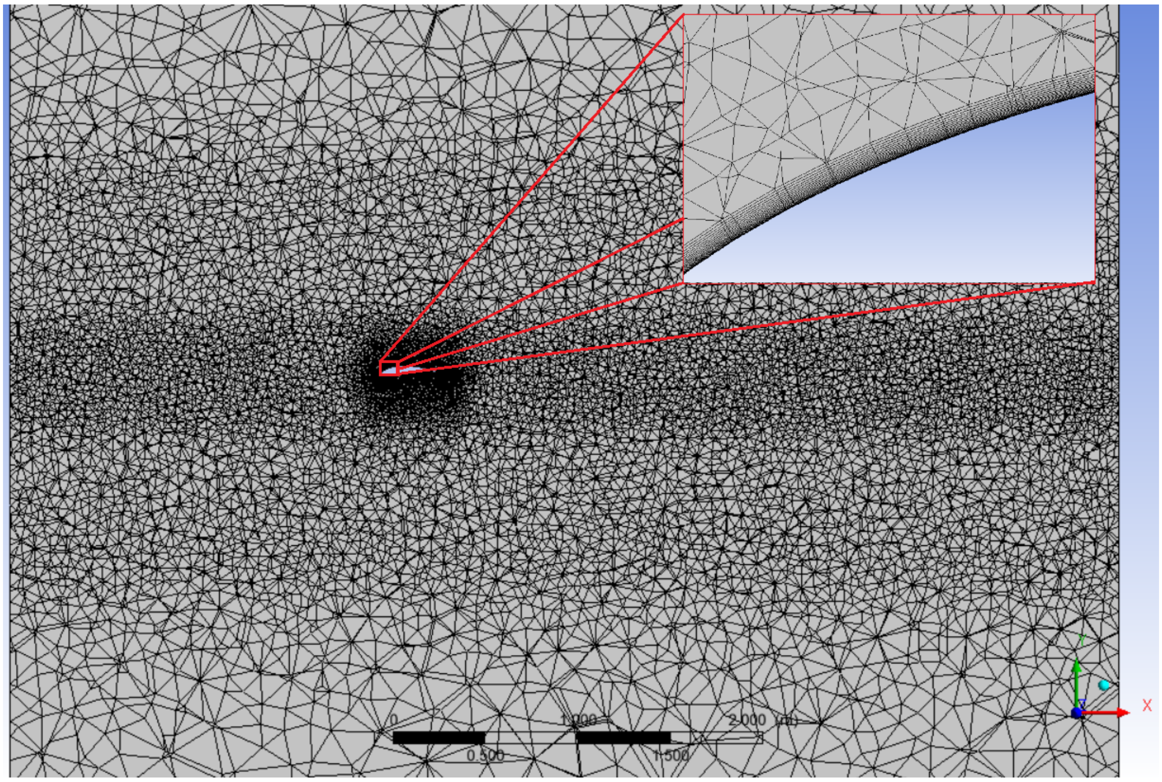


Figure 3. Grid with 1491838 nodes showing boundary layer zone.

The dependence of the mesh was analysed using the $h_i$ parameter, which can be calculated by Equation 4 where $h_1$ is the lowest value of $h_i$, $Vol$ is the total volume of the domain and $N$ is the total number of nodes according to Ferro (2009). Table 3 shows values of $h_i/h_1$ according to the number of nodes of each mesh analysed.

$$h_i = \sqrt[3]{\frac{Vol}{N}} \tag{4}$$

Table 3. $h_i/h_1$ parameter as a function of nodes number

| Nodes number | $h_i/h_1$ | $C_L$ |
|---|---|---|
| 878625 | 1.2497 | 0.837 |
| 931027 | 1.2258 | 0.841 |
| 996427 | 1.1702 | 0.849 |
| 1070034 | 1.1373 | 0.853 |

| | | |
|---|---|---|
| 1165587 | 1.0908 | 0.858 |
| 1491838 | 1.0000 | 0.871 |

The boundary conditions imposed at the boundaries were *Velocity Inlet* at the inlet section, *Opening* at the outlet section and at the walls of the tunnel, *No Slip Wall* for viscous simulations and *Free Slip Wall* for inviscid simulations at the wing boundaries. The model of turbulence chosen was the *SST* because it is the most recommended model for aerodynamic analysis according to Ansys (2016). For the solver, it was chosen to use upwind in Advection Scheme because this option gives greater robustness to the numerical solution. The convergence criterion was chosen for $1\times10^{-7}$ in RMS with 500 maximum iterations.

## 4. RESULTS AND DISCUSSION

The main objective of this work is to do a comparison between the results obtained with the Vortex Lattice Method and the Finite Volume Method in calculating a flow about a wing of an aircraft for the SAE Aerodesign competition.

The dependence of the results on mesh was analysed comparing the values of $C_L$ as a function of $h_i/h_1$. Figure 4 shows that it was not possible to obtain independence of the results with the mesh size. However, the value of $h_i/h_1 = 1$ is the maximum number of elements that the available computer supports. This value was considered as the best result since it was not possible to further refine the mesh on the surface wing.

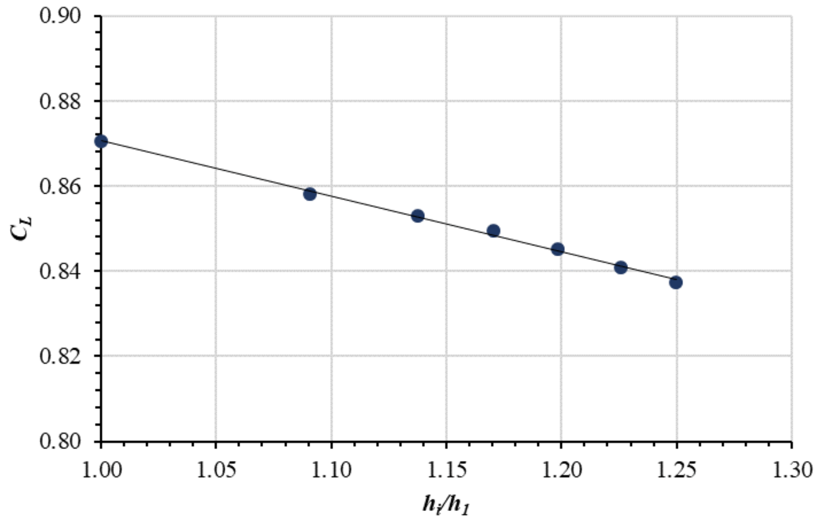


Figure 4. Lift coefficient of the wing as a function of $h_i/h_1$ to an angle of attack $\alpha = 0°$

Figure 5 shows a comparison of lift coefficient $C_L$, plotted versus the angle of attack $\alpha$, computed with the CFX code, viscous and inviscid flows, and the AVL code. The graphs have very similar behavior, with very small differences. The curve of AVL code presents a slightly lower slope than that of the CFX at the linear region.

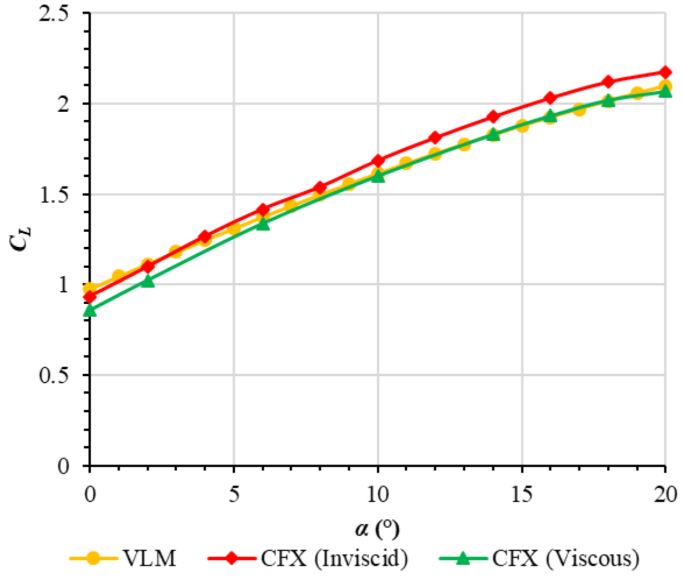


Figure 5. Variation of $C_L$ with the angle of attack $\alpha$

Figure 6 shows the lift distributions along the wingspan for the two methods analysed at an angle $\alpha = 0°$, confirming what is displayed on the $C_L \times \alpha$ curve of Figure 5.

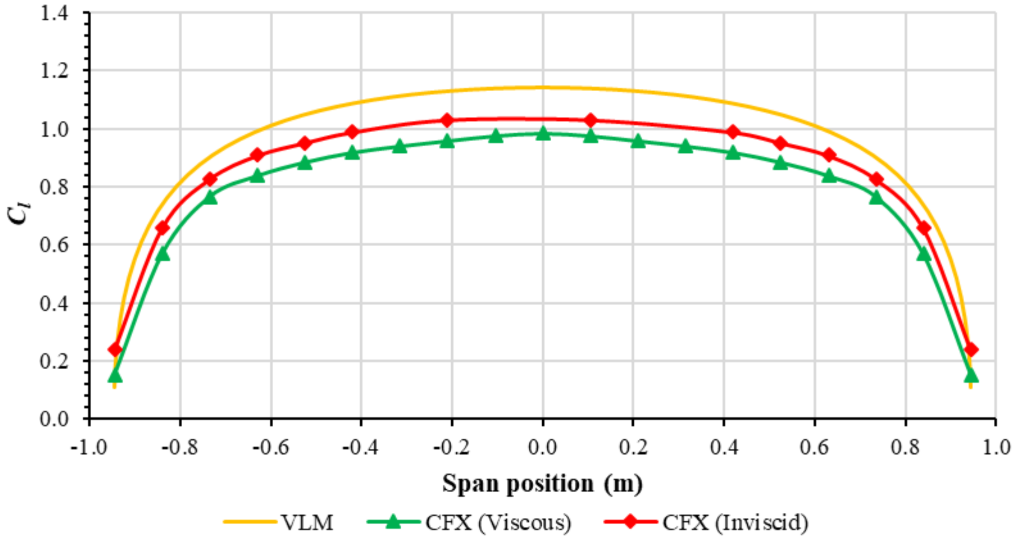


Figure 6. Lift coefficient distributions along the wing span for an angle of attack α = 0°

Figures 7 and 8 shows that the pressure coefficient variation $\Delta C_P$, distributions where $\Delta C_P$ is the difference between lower surface and upper surface pressure coefficients, computed by the CFX and VLM codes have a similar behaviour. The main difference between the curves is close the leading edge, where a different behavior is observed. This difference can be linked to the calculation of the position of the stagnation point, which may have different for each code.

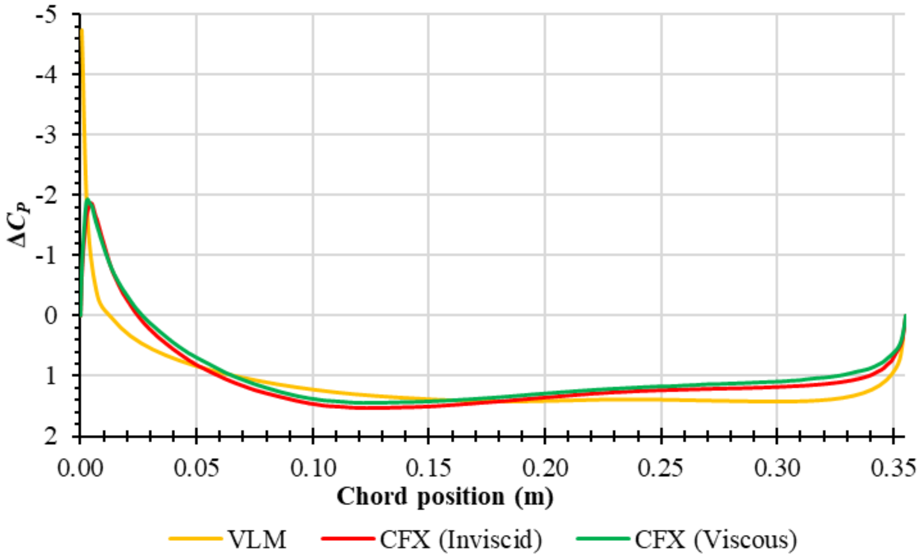


Figure 7. $\Delta C_P$ along the chord at the root of the wing ($y$=0) for an angle of attack $\alpha = 0°$

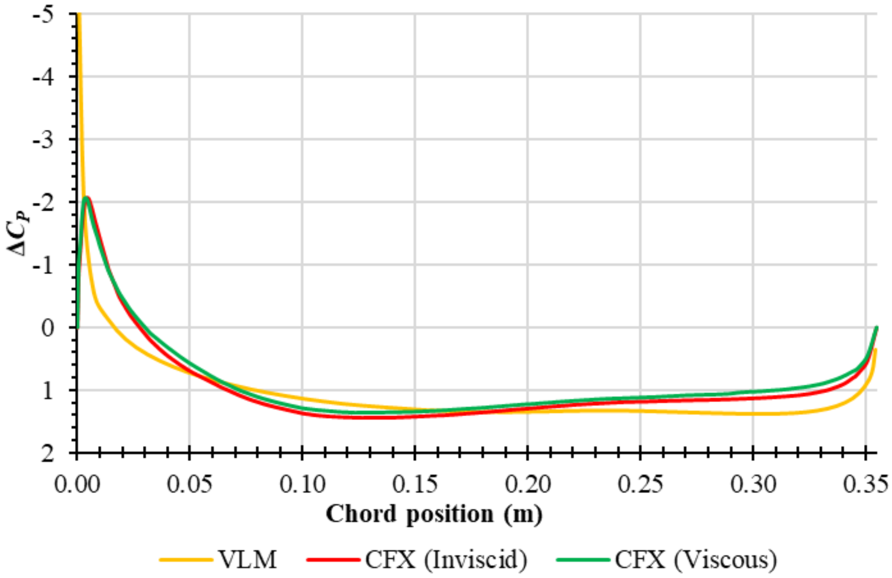


Figure 8. $\Delta C_P$ along the chord at the $y/(b/2) = 0.5$ wing for an angle of attack $\alpha = 0°$

Figure 9 shows computed pressure distributions on the wing surface for an angle of attack $\alpha = 0°$, showing a fairly uniform pressure distribution. Figure 10 shows the distribution of the pressure in section planes for an angle of attack $\alpha = 0°$.

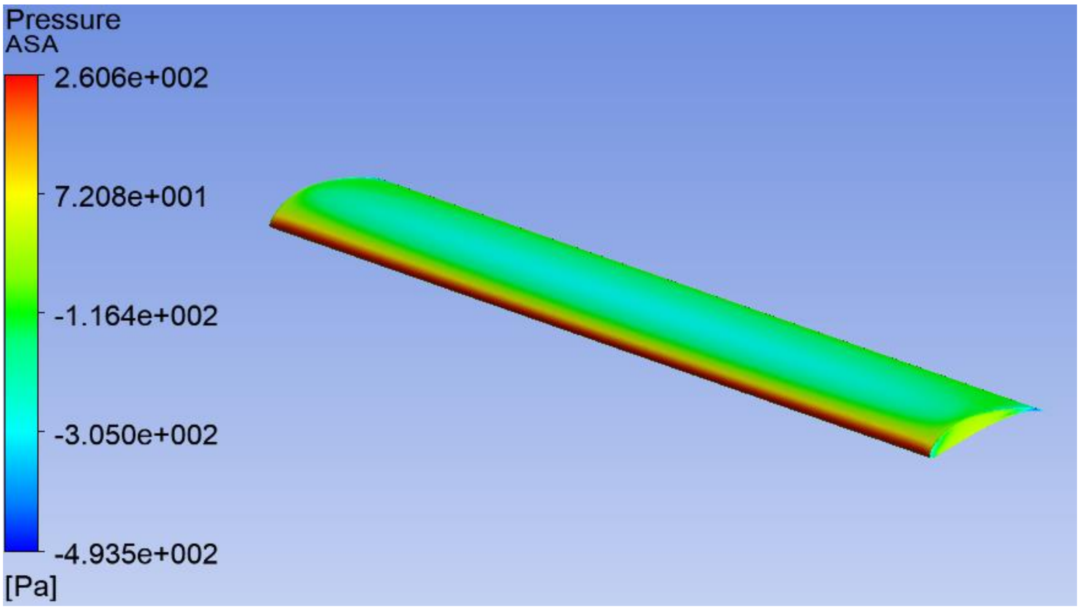


Figure 9. Pressure distribution on the surface of the wing

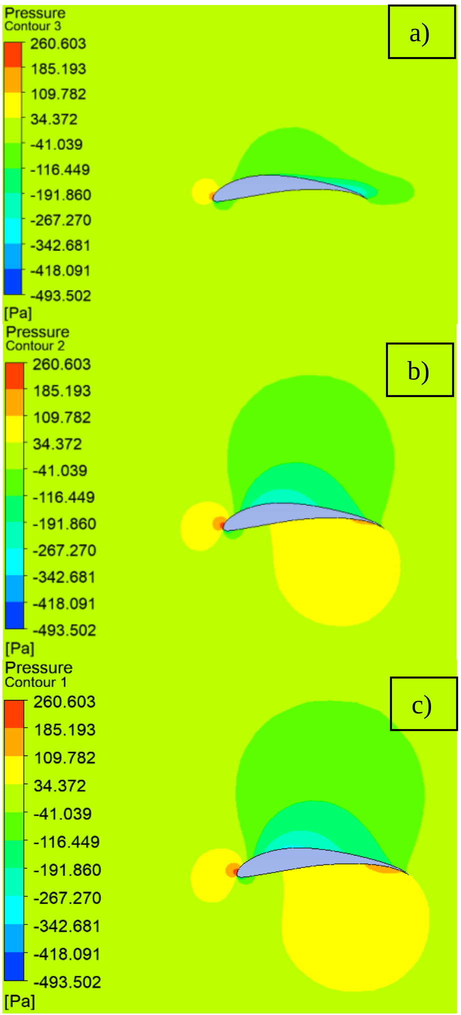


Figure 10. Pressure distribution in section planes at a) $y$ = 0.945 m, b) $y$ = 0.472 m and c)  = 0 m for an angle of attack $\alpha = 0°$

## 5. CONCLUSION

The incompressible three-dimensional flow around a wing of an aircraft destined for the SAE Brasil Aerodesign competition was calculated using Vortex Lattice method and ANSYS CFX Finite Volume code on the solution of inviscid flow and solution of the Navier-Stokes equations and SST turbulence model. The analysis of the dependence of the results of the number of elements used on the discretization of wing surface and contour discretization for the CFX was analysed, and It was noticed that with the machine used in the simulations an excellent convergence of values of the lift coefficient was not possible.

The graphs of $C_L$ as a function of the angle of attack were computed for the inviscid and viscous flow and a very similar behavior was observed in all the curves with a small difference of values.

The Vortex Lattice method proved to be a great tool for simple analysis, with good results and fast convergence. The CFX presented similar results, however the convergence time reaches much more than 1000 times the AVL time, proving to be a good tool for a deeper analysis of a flow on a wing.

## 6. ACKNOWLEDGEMENTS

The authors would like to express their acknowledgements to Federal Rural University of Semi-Árido (UFERSA) and to Engineering Center of UFERSA for the financial support.

## 7. REFERENCES

Anderson, J.D.2011. *Fundamentals of Aerodynamics*, New York., 5th edition.

ANSYS, 2016. *ANSYS Fluent Theory Guide*.

Bertin, J.J. and Cummings, R.M., 2009. *Aerodynamics for Engineers*, Pearson Education International, New York, 5th edition.

Brederode V. de, 1997. *Incompressible Aerodynamic Fundaments (in portuguese)*, Lisbon, Author edition.

Ferro, L.M.C., 2009. *Numerical and experimental study of the flow through an axial hydraulic turbine, in portuguese*. Ph. D. thesis, Technical University of Lisbon, Lisbon

Launder B.E. and Spalding D.B., 1972. *Lectures in Mathematical Models of Turbulence*, Academic Press, London.

Menter F.R., 1994. "Two-equation eddy-viscosity turbulence models for engineering applications", *AIAA Journal*, Vol. 23. No 4, p. 1598-1605.

Johnson, F.T., Tinoco, E.N. and Yu, N.J., 2005. "Thirty years of development and application of CDF at Boeing commercial airplanes, Seattle, *Computer&Fluids*, Vol. 34, No. 10, p. 1115-1151.

Menter F. R., 2009. "Review of the SST turbulence model experience from an industrial perspective". *International Journal of Computational Fluid Dynamics*, Vol.23, No 4, p. 305-316.

SAE BRASIL. Aerodesign. Avaliable at: <http://portal.saebrasil.org.br/programas-estudantis/sae-brasil-aerodesign>, 2016.

Souza, M. S., 2007. "*Further developments in unsteady compressible vortex lattice method in two dimensional motion*" Masters dissertation, Aeronautics Institute of Technology (ITA), São José dos Campos.

Spalart, P.R. and Allmaras S.R., 1992. "A one-equation turbulence model for aerodynamic flows, AIAA Paper 92-0439.

Multhopp H, 1950. *Methods for Calculating the Lift Distribution of Wings*. *Subsonic Surface Lifting Theory*. Aeronautical Research R.&M. 2884, London.

Wilcox D.C., 2007. "Formulation of the $k$-$\omega$ turbulence model revisited", In *Proceedings of 45th AIAA Aerospace Sciences Meeting And Exhibit, Aerospace Sciences Meetings*, Reno, United States of America.

## 8. RESPONSIBILITY NOTICE

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