ISSN 2176-5480

22nd International Congress of Mechanical Engineering (COBEM 2013)
November 3-7, 2013, Ribeirão Preto, SP, Brazil

PROJECT, CONTRUCTION AND ANALYSIS OF AN ALTERNATIVE SUBSONIC HORIZONTAL AERODYNAMIC WIND TUNNEL

Rafael da Silva Alves
Lincoln Nascimento Ribeiro
Universidade Cruzeiro do Sul - UNICSUL, Departamento de Ciências Exatas e Tecnológicas.
Av. Dr. Ussiel Cirilo,225, São Miguel, CEP 08060-070, São Paulo,SP, Brasil
rafael_ferramenteiro@yahoo.com.br, lincoln.ribeiro@cruzeirodosul.edu.br

Victor Barbosa Felix
Universidade Cruzeiro do Sul - UNICSUL, Departamento de Ciências Exatas e Tecnológicas.
Av. Dr. Ussiel Cirilo,225, São Miguel, CEP 08060-070, São Paulo,SP, Brasil
victor.felix@cruzeirodosul.edu.br

Abstract. *This work consists in an alternative way to study, design and analysis of a wind tunnel, the most used instrument in aerodynamics and air flow fields in the aeronautical and automobilist areas. The classical theory of fluid mechanics was used with addition of some simplifications in the construction of the wind tunnel, including the contraction nozzle in the linear form, using these results to orient the construction of wind tunnels to use in a didactic lab. The calculation results are shown with boundary conditions and the adopted methods based on the critical length of the boundary layer. Numerical simulations were performed using a commercial CFD (Computed Fluid Dynamics) code, representing virtually the wind tunnel projected to validate the study to be made in real cases, and taking their results in comparison with the analytical values with already known solution, proving the reliability and effectiveness of this project compared to the classical works in this area.*

Keywords: *Wind Tunnel, Aerodynamics, Project, CFD, Computed Fluid Dynamics*

Nomenclature

Re	Reynolds number
Re_m	Model Reynolds number
Re_p	Prototype Reynolds number
Eu	Euler number
X_{cr}	Critical length
μ	Dynamic viscosity
ρ	Fluid density
v_0	Flow velocity
S	Superior wing area
h	Height from ground
k_n	Similarity equation of "n"
cl	Coefficient of lift
Fy	Sustentation force
ρ_m	Model fluid density
ρ_p	Prototype fluid density
μ_m	Model dynamic viscosity
μ_p	Prototype dynamic viscosity
c_m	Model wing chord
c_p	Prototype wing chord
b_m	Model wingspan
b_p	Prototype wingspan
s_m	Model superior wing area
s_p	Prototype superior wing area
v_m	Model flow velocity
v_p	Prototype flow velocity

ISSN 2176-5480

1. INTRODUCTION

The demand of the globalized world by more efficient and economic machinery has been increased at the last years. Cars, planes and other kind of vehicles each time quicker and simultaneously that demands less fuel to work is a fact. The most common tool to improve these aspects is the wind tunnel. Most used in the aeronautical and automobilist fields, this instrument is used to measure the lift coefficient and the drag coefficient and indicates the flow field around an immersed body. The major difficult in the design of this machine is the capacity to generate and guarantee an adequate air flow through the studied body to simulate the same physical variables in model (usually in reduced scale) as in the prototype in real conditions and so be possible to control and reduce the losses (friction for example) and increase the gains (velocity or capacity of lift in airplanes for example).

As the study in the aerodynamic field is growing up, some engineers and researchers in the world started to study with more details the fluids performance and their influence at the capability to generate sustentation and reduce drags, in most cases based on wind tunnels of complicated project, but in almost all cases without a practical guide or technical information to project an appropriate wind tunnel (specially how to project the contraction nozzle to get a correct velocity profile in the test area).

One of the most pioneering researchers in the area, the engineer Morel (1975) developed an analytical method based on practical charts and some pre-defined shapes analyzed using a CFD code. The subject of study is to analyze the contraction nozzle region in the wind tunnel, to increase the fluid flow velocity in the test area, and to reduce the boundary layer detachment, to avoid the pressure loss. This method was based in a curved geometrical form and a series of design criteria to help the designer in new projects. The shape proposed by the author is shown in Fig. 1

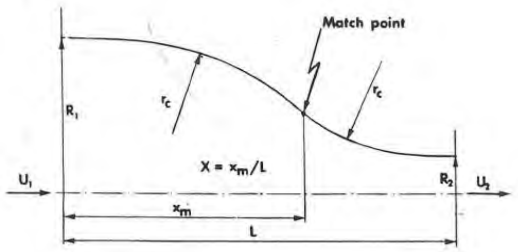

Figure 1. Wall contour construction of two matched cubic arcs (Morel, 1975)

Initially the study of wind tunnels was done based in empirical methods, like the “design by eye” according to Bradshaw et al (1979). This study is about the contractions, with the objective to perform a boundary layer control before the test chamber to avoid the flow separation and to get a uniform and steady stream at test chamber outlet. There is not yet an ideal profile shape design for this situation. Notwithstanding, the wall shape theoretical design has no wholly satisfactory method and calculation of the Stokes-Beltrami equation which is relatively simple to solve for simple geometries and regular flow fields that is not applied for these profiles.

According Bradshaw et al (1979) the “design by eye” that is the most often method used in contractions is an unscientific way. But the authors concluded that the form of a contraction contour it’s not very important to wind tunnel design, except, near the ends. The authors recommendations were based initially on empirical tests and observations.

Albayrak (1991) performed a further study of wind tunnels design. The main objective of this work is the design of axisymmetric contractions nozzle region by the boundary layer control methods, to avoid recirculation and detachment of the flow in the contraction at the test area (section of the wind tunnel where the quality of air flow is more important), ensuring a valid analysis of the air flow. The author method was based in the previous works who approached the shape profile of the contraction by the irrotational and axisymmetric streamlines using the Stokes-Beltrami equation of stream function. This study was an improvement of the previous work in this area.

Pravia et al (2003) presented a didactic alternative and simplified construction and design of a mini wind tunnel (MTV – Mini Túnel de Vento). This work was based on the NASA instructions for a didactic low-cost wind tunnel, but with some modifications and adaptations to construct the wind tunnel with the available resources, maintaining the low costs production. The design and modifications although were made in empirical base, using to start the project, the sheets and instructions in the didactic wind tunnel made by NASA (Baals tunnel).

Recently, Uzueli et al (2008) constructed a wind tunnel for didactic purposes for building and civil engineering analysis using the common and classic methods for project, dimensioning and instrumentation of the machine. The contraction nozzle design was based on the Morel (1975) recommendations with the wall shape modeled in a curve form to get the optimized work as described in other studies.

The work herein cited have the proposition to make an alternative study, different of what was already made at the present moment. For didactic and study purposes, this research have the goal to demonstrate the design procedures for this kind of equipment with a simplified geometry, using the critical length (at the point that the flow takes the characteristic of turbulent) of the development of the boundary layer in forced ducts as controlled parameter of design. This was made thinking in the facility of construction (and consequently the reduced spending) idealizing a straight

ISSN 2176-5480

contraction nozzle unlike the studies made at this moment. To validate the work of this study the commercial CFD code Flow Simulation 2012© (from SolidWorks©) was used at the analysis of the wind tunnel project proposed and to analyses the flow around an airplane wing with known analytical values to compare with the simulated results.

2. STUDY DESCRIPTION

This wind tunnel was mostly based on empirical analysis or constructive factors in most of its parts, however some crucial and specific parts of the machine which makes direct influence in the flow field were studied based on the physicists analytical theories of the fluid flow and more specifically aerodynamics study, being these parts the study object of this work (tunnel length and test section). In Fig. 2 is possible to view the general schema of the wind tunnel proposed.

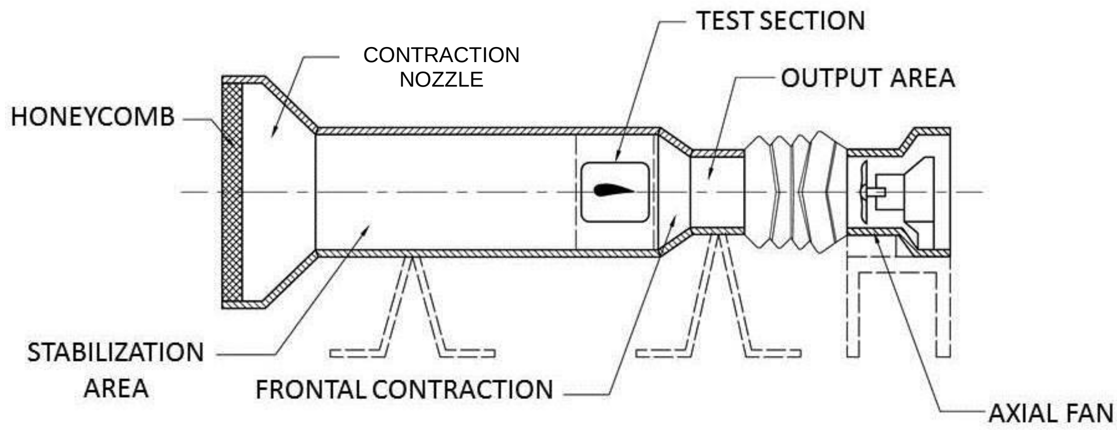

Figure 1. Wind Tunnel main scheme and its parts

Considerations about the tunnel construction

- The honeycomb was installed just in the entrance of the air to stabilize and linearize at the maximum possible the flow and to reduce the local turbulence in the fluid coming from the external ambient;

- Optionally, to improve the flow, it is possible to add a honeycomb right after the contraction;

- The construction of the tunnel was idealized with flanged connections to make the assembly in separated parts, facilitating the installation and uninstallation in modules;

- The connection between the axial fan and the tunnel was chosen to be made with flexible link with the objective to reduce the vibration effects of the fan over the tunnel and to allow the posterior change to a more potent blower;

- The axial fan was chosen based on economic factors and the frontal contraction was made in this form due to the available fan for use which has reduced dimensions in comparison to the transversal section of the test area.

2.1 Stabilization area

The stabilization area is the difference of this project about the others projects presented and herein cited. The subject and functionality of this addition is to compensate the boundary layer detachment and disorder flow caused by the simple geometry of the contraction nozzle, that is in a straight shape (unlike the recommendations of Morel (1975). Morel (1975) has established a criteria and way of calculate the adequate shape format of the contraction ideal curve.

The method used to control the flow generated in the contraction nozzle is based on the boundary layer development in forced ducts study, where the variable in analysis is the length of the stabilization area. In this tunnel, the intended flow on the test section is the turbulence flow; however in a dynamically established steady flow the velocity profile assumes a constant contour after the critical length.

The critical length of the tunnel is calculated by the Reynolds's equation as follow in Eq.1

ISSN 2176-5480

$$x_{cr} = \frac{5 \cdot 10^5 \cdot \mu}{\rho \cdot v_0} \quad (1)$$

Where x_{cr} is equal to the effective length of the stabilization area and the value $5x10^5$ is the Reynolds's number when occur the dynamically established steady flux. From this point, the velocity profile will be turbulent, but constant in from "x" point. This operation condition is acceptable for the test area of the wind tunnel.

2.2 Mathematical modeling

To prove the effectiveness of the simplification proposed in the contraction nozzle with the adequate correction in the stabilization area, it was proposed to analyses a case study that will cover a gamma of aerodynamic profiles of variable capacity and size. The object of study was the Eppler 423 aerodynamic shape. The flying conditions of the prototype (velocity, flying height) were based in the Super-Tucano EMB-314 airplane, being the consideration of a straight wing without tapering with an elongation AR > 4 to calculus simplification. In the (Tab.1) is shown the data for analysis.

Table 1. General information of the study object

CASE STUDY - AIRPLANE WING AERDOYNAMICAL PROFILE	
PLANE STUDIED:	EMB-314 / Super Tucano
WINGSPAN:	11.14 m
MASS:	3200 kg
MAX. VELOCITY (Vp):	163.89 m/s
SUPERIOR WING AREA (S):	19.4 m²
FLYING HEIGHT:	10 665 m
WING CHORD:	2.28 m
AERODYNAMIC SHAPE:	Eppler 423
SCLAE OF REDUCTION:	1:30

To calculate the air density in the flying height of the prototype, with the objective to make more accurate the calculation of dimensional analysis and complete similarity, it was adopted a mathematical modeling created by Barros (2012) and these equations is resumed by the equations (Eq. 2, 3 and 4):

$$\rho = \ln\left(\frac{k^n}{e^e}\right) \quad (2)$$

$$k = (0{,}0002 \cdot h) + 2 \quad (3)$$

$$n = \frac{1}{[(-3{,}478 \cdot 10^{-10} \cdot h^2) + (2{,}98 \cdot 10^{-5} \cdot h) + 0{,}1756]} \quad (4)$$

To determine the exact parameters to be possible to have a complete flow similarity between prototype (real scale) and model (tunnel scale), initially was considered the complete similarity condition by the Reynolds's number (Re) which in briefly form, establishes the relation among model and prototype – similarity equations "k" (eq. 5 / 6).

$$Re_m = Re_p \quad (5)$$

$$k_\mu = k_\rho \cdot k_V \cdot k_c \quad (6)$$

Through the presented relations, is observed that in the geometrical scale of reduction in 1:30, it was necessary a flow velocity of fluid in the wind tunnel approximated in 1638,9 m/s, what is clearly an impossibility. For this case it was analyzed that the Reynolds's numbers for each cases (model and prototype) are greater than 2400, establishing a turbulent flow. Because of this it is possible to consider the viscosity effects negligible in comparison with the pressure forces (greater cause of aerodynamic forces) like said by Oliveira et al (2010).

ISSN 2176-5480

Then can be utilized as an alternative, the Euler similarity scale for dimensioning criteria of velocity, being the results of the equations of similarity the forces generated by the flow, adopting the velocity (V_m) as an input data (based on the blower available). By this means, it is possible to obtain empirically the drag and lift coefficients values that don't depend of the adopted scale. The equation utilized is shown in the Eq. 7 and 8.

$$Eu = \frac{F}{\rho \cdot V^2 \cdot L^2} \tag{7}$$

$$k_F = k_\rho \cdot k_{V^2} \cdot k_{L^2} \tag{8}$$

Taking as initial data already known: ρ_m / ρ_p / V_m / V_p / L_m^2 / L_p^2

To the calculations of drag (cd) and lift (cl) coefficients, is used a variation of Euler's number equation, namely in the equation 9:

$$cl = \frac{Fy}{\frac{1}{2} \cdot \rho \cdot V_0{}^2 \cdot S} \tag{9}$$

For the calculations of the physical variables in the tunnel, was used the Tab. 2 with the values between the model and the prototype:

Table 2. Physical parameter of the analysis

MODEL	***PROTOTYPE***
ρ_m = 1,225 kg/m³	ρ_p = 0.408 kg/m³
c_m = 0,0759 m *	c_p = 2.28 m
b_m = 0,1855 m *	b_p = 5.57 m
s_m = 0,0141 m² *	s_p = 12.7 m²
V_m = 2,7 m/s **	V_p = 163.89 m/s
μ_m = 0,0000174 N.s/m²	μ_p = 0.0000174 N.s/m²

* Geometric recutcion scale of 1:30

** Velocity adopted as inicial value of the calculus

2.3 Computational Simulation

The numerical simulations for validation purposes have been performed using the Flow Simulation 2012 commercial code (CFD) with the goal to prove the effectiveness of the study, making a virtual workbench to simulate the effects of the tunnel idealized in this research and its effects in comparison with the already known values from the classic studies and empirical values taken from other tunnels.

There were made two type of computational analysis, being the first an analysis of the wind tunnel as a whole to prove the function of the stabilization area with the critical length calculated before the test area to guarantee an uniform flow. The second analysis was made focusing in the study object itself and the effects of the flow arising from the tunnel and making a comparison of the lift coefficients (cl) calculated by the CFD representing the tunnel herein designed and the analytical values already calculated and known by the literature.

First Study – Complete Wind Tunnel

For the first study was considered the complete tunnel with the instrumentation and the study object in the bosom of the flow. To make possible the computational analysis, the study model for CFD was simplified taking away from the model insignificant a construction detail that doesn't affect the flow (like bolts and screws). The Fig. 3 shows the model for analysis and the Tab. 3 shows the boundary conditions.

ISSN 2176-5480

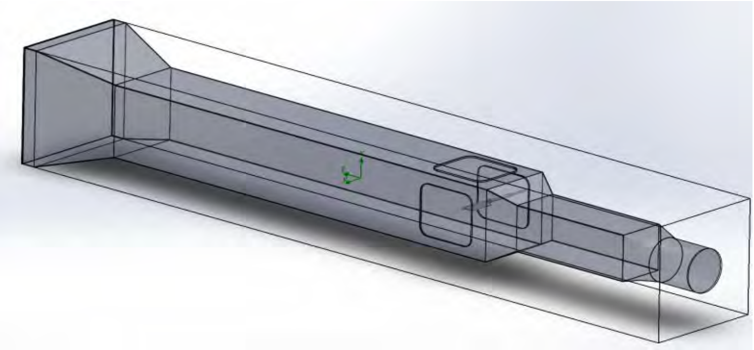

Figure 2. Simplified model for CFD analysis

Table 3. Boundary conditions of the first study

BOUNDARY CONDITIONS - STUDY 1	
EXIT FLOW RATE *	0.675 m^3/s
INLET NOZZLE PRESSURE **	101 325 kPa
WORK TEMPERATURE **	293.2 K
BOUNDARY LAYER TYPE	Turbulent
ANALYSIS TYPE	Internal
WORK FLUID	Air (gas)
WALL CONDITIONS	Adiabatic Walls

* Exit flow rate through the circular nozzle at the right in the fig.3

** Approximated numbers equivalents to the ambient values

Results: First Study

From the analysis made, the result has shown an adequate flow with a velocity profile approximated unidimensional in the test section, occurring fluid perturbation only near the walls of the tunnel what is justified by the boundary layer theory and it is acceptable cause don't makes influence in the object studied. The results of the velocity panorama are shown in the Fig. 4 with the respective color scale.

Through the Fig. 4 is possible to observe that in the test area, the reached velocity was in turn of 2.7 m/s, constantly in all this section, what is inside with the initial requirements. In the Fig. 5 it is possible to view the test section with more detail (with the color scale modified to better viewing), and is possible to view that near the walls of the tunnel the velocity tends to zero in a gradual scale (development of the boundary layer) as already said.

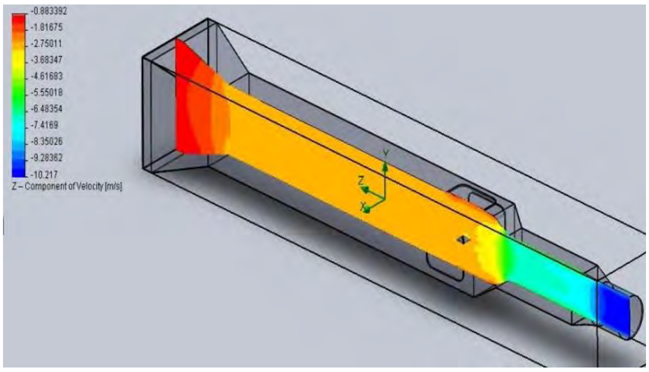

Figure 3. Velocity scale results in the wind tunnel

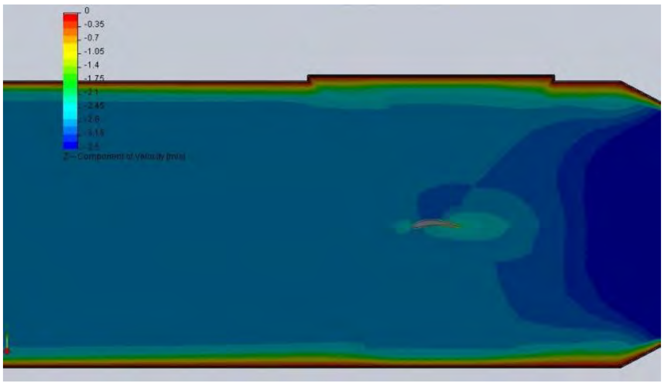

Figure 4. Velocity profule in the test section

ISSN 2176-5480

In the analysis made, by the software resources, was created a graphic that demonstrates the velocity profile (in two different flow rates) in the test area (velocity x tunnel section length). Is verified in the Fig. 6 that the velocity in the middle of the section (place where the study object is located) is constant in the value of ~ 2.7 m/s what validates the study of the stabilization area.

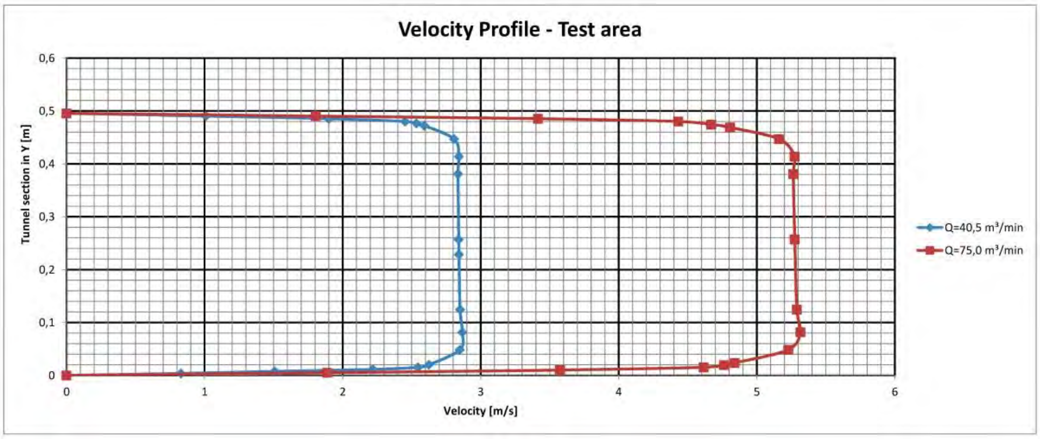

Figure 5. Velocity profile of the test section

Second Study – Airplane Wing in the wind tunnel (detailed)

In the second study, the object of study was focused as a detail of the analysis with the objective to make a comparison between the lift coefficient calculated and the simulated by the CFD code. Through the XFLR 5 aerodynamic software was obtained the *cl* value for the studied shape (Eppler 423) for the model parameters (Re = 14 286), the obtained result is shown in the Fig. 7.

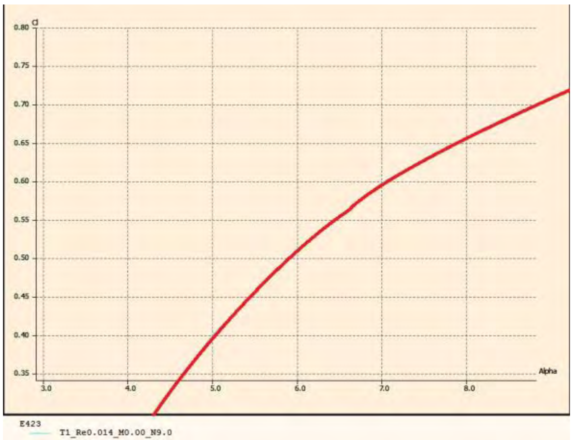

Figure 6. Graphic of cl in function of alpha

The value obtained of the lift coefficient (cl) for a wing attack angle of 7º (angle utilized for the CFD analysis) was in the value of 0.5984. Making the calculations to the lift force results in 0,0378 N. In the next figure (Fig. 8) is shown the CAD model used in CFD analysis and the table 4 shows the boundary conditions.

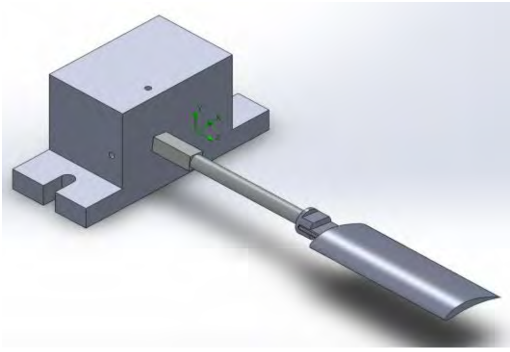

Figure 7. Detailed model for the second study

Table 4. Boundary conditions of the second study

BOUNDARY CONDITIONS - STUDY 2	
VELOCITY INPUT *	2.7 m/s
INLET NOZZLE PRESSURE **	101 325 kPa
WORK TEMPERATURE **	293,2 K
BOUNDARY LAYER TYPE	Turbulent
ANALYSIS TYPE	External
WORK FLUID	Air (gas)
WALL CONDITIONS	Adiabatic wall

* Velocity vector in the X direction

** Approximated numbers equivalents to the ambient values

Results: Second Study

In this analysis made in commercial code CFD, the results obtained have the advantage over the analytical calculus cause in the computational domain is toke in consideration the finite wing (unlike the analytical form that considers the aerodynamic profile – infinity wing) including the wing tip vortex effects, which increase the precision and accuracy of the results.

In the Fig. 9 it is possible to visualize the coherence of the analysis remembering that the relative pressure is directly related with the generated forces in the wing. It is observed that in the soffit there is a greater pressure in comparison with to the extrados (minor pressure) what causes a resultant force upward justifying the lift. Through the Fig. 10 it is possible to visualize the streamlines showing the wing tip vortex.

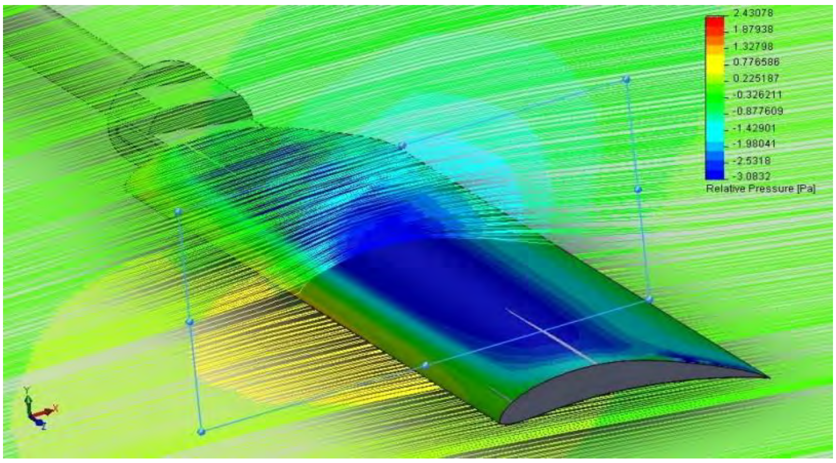

Figure 8. Pressure scale in the middle of wing

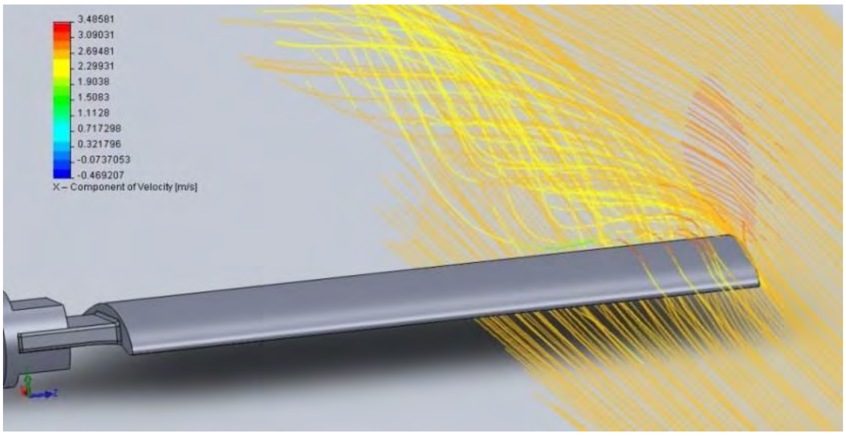

Figure 9. Streamlines in the wing tip

Through these considerations it is possible to make a comparative table with each result, the analytical one and the CFD code. In the table 5 is shown the two values of lift coefficient and the relative error.

Table 5. Comparative table between analytical and CFD results

COMPARATIVE TABLE - ANALYTICAL / CFD VALUES	
Fy (analytical)	0.0378 N
Fy (CFD)	0.0368 N
Relative error (abs)	0,001 N
Relative error (%)	2.645 %

2.4 Conclusions

The present work discusses the efficiency of an alternative method and economic to the design and development of wind tunnels accessible and with easy construction, compensating the simplification in the geometric shape of the contraction nozzle adding a stabilization length to stabilize the air flow to test area to make possible this kind of machine in a didactic context.

Through a case study specific it was concluded that served with excellence the established objectives, resulting in a relative error between the analytical values already consecrated and the CFD simulation of the tunnel negligible (2,645%) what for didactic purposes is very acceptable.

As a suggestion for future studies about this design, the same simulation made could be done in a real constructed wind tunnel with these specifications, consecrating the values herein shown.

3. ACKNOWLEDGEMENTS

The authors would like to acknowledge the University Cruzeiro do Sul – UNICSUL by the theoretical support and practical, and its professors who helped the development of this work and the University of Taubaté – UNITAU in special the Engineer Coordinator Evandro Luis Honara by the contribution with the technical visit in their aerodynamic installations providing great know about the practical phenomena that occurs in a wind tunnel.

4. REFERENCES

OLIVEIRA, L.A.; LOPES, 2010. *Mecânica dos fluídos*. 3rd ed. Lisboa. Publisher ETEP

MOREL, T., "*Comprehensive Design of Axisymmetric Wind Tunnel Contractions". ***Journal of Fluids Engineering**. June 1975.

BRADSHAW, P.; MEHTA, R.D., "*Technical Notes – Design Rules for small low speed wind tunnels". ***The Aeronautical Journal of the Royal Aeronautical Society**. November 1979.

BRADSHAW, P.; PUNKHURST, R. C., 1979. "*The Design of Low-Speed Wind Tunnels".* National Physical Laboratory, Teddington.

ALBAYRAK, K, "*Design of a Low-speed axissymetric wind tunnel contraction". ***Journal of Wind Engineering and Industrial Aerodynamics**. v.37, 1991.

ISSN 2176-5480

PRAVIA, Z.M.C. et al, 2003. "Um protótipo de um mini túnel de vento (MTV) para ensino de graduação". In *Proceedings of the XXI Brasilian Congress of Education in Engineering – COBENGE 2003*. Rio Grande do Sul, Brazil.

UZUELI, A.G., 2008. "*Túnel de Vento da FAUUSP - Instrumento para ensaios de modelos físicos em conforto ambiental".* Universidade de São Paulo - Faculdade de Arquitetura e Urbanismo, São Paulo.

BARROS, C. S. "Densidade do ar em função da altitude" YouTube . 20 June 2012
<http: //www.youtube.com/watch?v=udBZ-WhsucE >.

5. RESPONSIBILITY NOTICE

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