Wind Tunnel Test of a Three-Element Airfoil Configuration: Influence of the Flow Direction Generated by a Blower, Used to Minimize Three-Dimensional Flow Along the Model Span

Roberto da Mota Girardi
Instituto Tecnológico de Aeronáutica (ITA) – Praça Mal. Eduardo Gomes, 50 - 12.228-900 - S. J. dos Campos, SP, Brazil
girardi@ita.br

Marcelo Assato
Instituto Tecnológico de Aeronáutica (ITA) – Praça Mal. Eduardo Gomes, 50 - 12.228-900 - S. J. dos Campos, SP, Brazil
assato@.ita.br

Dimitrios van der Werff
Delft Technical University, Delft, The Netherlands
dsvdwerff@hotmail.com

Abstract: *In order to minimize the three dimensional flow on a two dimensional model placed inside a wind tunnel, blowers are installed at the wind tunnel walls where the model extremities are fixed. The basic idea is to increase the momentum of the wind tunnel boundary layer and avoid migration of fluid particles from the wind tunnel surface to the upper surface of a two dimensional model. With such kind of procedure the flow along the model span is minimized and true two dimensional characteristics can be obtained. The objective of the present paper is to present the first experimental results of a research line, whose main goal is to develop a methodology for obtaining true two-dimensional flow over three element airfoils, used as hyper lift devices at wings of the aircrafts designed by the EMBRAER staff. The flow behavior is verified through two sets of pressure taps, which are distributed along the chord, at two stations along the model span. Comparing the two pressure distributions is possible to verify if the flow is two-dimensional. Additional pressure taps distributed along the span, on the model upper surface, provides information about the flow field in the region between both stations mentioned above. In this paper two configurations to the blower system are compared and their basic difference is the compressed air jet direction, relative to the wind tunnel undisturbed flow. For one of the configurations both air flows are aligned and this pattern is kept constant while the airfoil angle of attack is changed. In the second configuration, the jet direction follows the airfoil angle of attack variations and, therefore, the jet flow direction is no longer aligned to the undisturbed flow. Almost all experiment is controlled by a computer and a great effort has been done to obtain reliable experimental results. The airfoil is tested in a range of angle of attack and the pressure inside the blower settling chamber has to be varied in order to minimize the three-dimensional flow along the model span. The differences between both configurations are small but the research objective was achieved and reliable two dimensional tests can be performed with the experimental apparatus proposed in this paper.*

Keywords: .Wind Tunnel, Airfoil Testing, Boundary-Layer Blowing, Experimental Procedure.

1. Introduction

In aerodynamic experiments for obtaining the flow behavior over an airfoil (two dimensional flow), the model extremities are fixed at the walls of the wind tunnel test section. Three dimensional flow caused by the pressure difference between the lower and upper airfoil surfaces (similar to the flow at a finite wing tip region) is avoided by such configuration. On the other hand, variations on the flow along the model span are caused by the interaction of the test section boundary layer and the model extremity and two-dimensional tests are compromised. This kind of problem is very strong for three-element airfoil and it is necessary to develop some measurement technique in order to obtain reliable results.

Before proposing solutions it is important to understand the problem associated to the interaction between the boundary-layer at the test section wall and the flow over the airfoil: In an infinite wing a true two-dimensional flow is established. Considering a fluid particle flowing along a streamline, one can realize that there is an equilibrium along the transverse direction, between the centrifugal force, associated to the fluid particle mass, and the centripetal force, associated to the pressure gradient caused by the airfoil shape and angle of attack. In the region at the airfoil model extremity, which is immersed in the boundary-layer of the test section wall, the force equilibrium mentioned above no longer exist, because the centrifugal force acting on a fluid particle is diminished due a velocity reduction, caused by the friction force. On the other hand, the pressure gradient inside the boundary-layer region is established by the flow immediately above its border, where the friction forces are very small. Therefore, considering the airfoil upper surface, in the boundary-layer region the fluid particles are subjected to a resultant force in the transverse direction, pointing toward the airfoil surface. Due to this resultant force the fluid particle is directed to the airfoil surface and a flow between the test section wall and the airfoil upper surface is established, causing a premature flow separation at the model extremities. Using flow visualization this behavior can be observed (see Pessoa and Girardi, 2004).

In order to minimize the three-dimensional behavior of the flow along the model span there are three different kinds of solutions: (i) compressed air can be blown in order to increment the wind tunnel boundary-layer momentum in the region near the model extremities (see Elsenaar, 1983), (ii) suction can be performed to withdraw the fluid particles with low values of momentum inside the boundary-layer at test section walls, where the model is fixed and (iii) end plates or fences (passive elements) can be installed at the model extremities regions, in order to filter the perturbation caused by the interaction discussed above and avoid its propagation to the central region of the model.

The solution of the problem treated in this paper is part of a great research project, supported by FAPESP and EMBRAER, whose main objective is to increment the productivity and reliability of aerodynamic test. The blower technique and the use of passive elements have been developed at the ITA's new wind tunnel, the suction technique has been tested at the USP-São Carlos wind tunnel and at the CTA – IAE larger wind tunnel, one of these techniques will be implemented.

The comparison of two blower system configurations is the objective of the present paper. The basic difference between the tested configurations is the plane jet (generated by the blowers) direction, relative to the undisturbed flow direction in the wind tunnel test section. In a configuration tested previously, both directions are coincident and do not vary while the model angle of attack is changed. In the second configuration, the blowers and the airfoil are fixed on a turn table and their relative positions are kept constant while the angle of attack is varied. In such case, the pane jet and the wind tunnel undisturbed flow directions are not coincident.

After a brief description of the ITA's new wind tunnel, the three-element airfoil used as model is characterized through its geometrical shape and dimensions, as well as, the number and positions of the pressure taps required to measure the pressure coefficient distribution along the chord. The experimental apparatus will be described, as well as, the procedure to obtain reliable results, which is almost completely performed automatically by a computer.

The results generated in this work allow to conclude that both configurations can be used to perform tests on two dimensional airfoils, where three dimensional flow is minimized. Up to the airfoil stall angle, both configurations produced the same pressure coefficient distribution at the model middle section station, showing that the experimental apparatus and procedure are capable to generate very reliable results. The pressure inside the blowers is the only difference observed and this parameter is associated to the experiment feasibility and cost, as will be discussed later.

2. Experimental Apparatus

The Aeronautical Institute of Technology (ITA) wind tunnel is an open circuit type, as can be seen in the figure 1. The atmospheric air is admitted in the wind tunnel by the entrance nozzle. In the settling chamber, air flows through a honeycomb and three screens. After the settling chamber there is a nozzle, with contraction ratio of 10:1. The test section is 1.0 m high and 4.0 m length. The width varies from 1.20 m to 1.36 m along the test section length and the maximum flow velocity is 80 m/s (Re = 4.7 x 10^6 per meter).

The wind tunnel diffuser is divided in two parts by the 200 hp power fan (see figure 1). The test section flow velocity can be controlled by a frequency inverter, capable to vary the fan rotation in steps of 1 rpm.

After the diffuser second part, there exist a fast expansion diffuser, used to reduce the velocity of the flow entering in a chamber, designed to damp the effects of atmospheric wind variations in the test section flow. In such chamber the wind tunnel flow is deflected to upward direction, which appear to be the most appropriate configuration to minimize the effect of the atmospheric wind variations. More detailed information is reported by Girardi et all (2002, 2003).

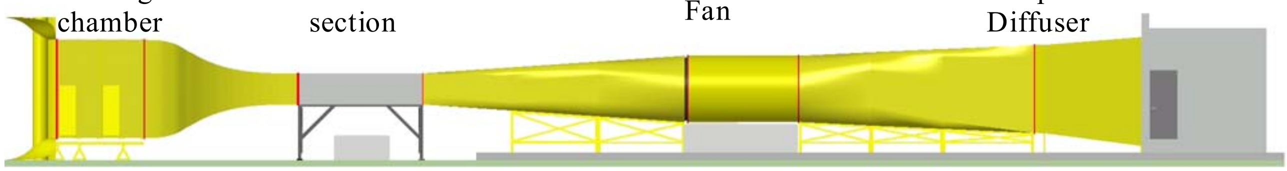
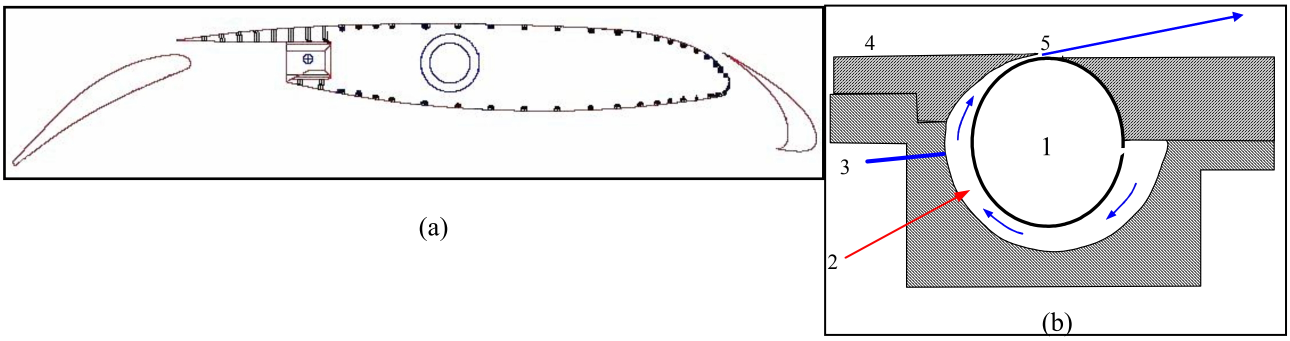


Figure 1: General view of the ITA's teaching and research wind tunnel and its elements.

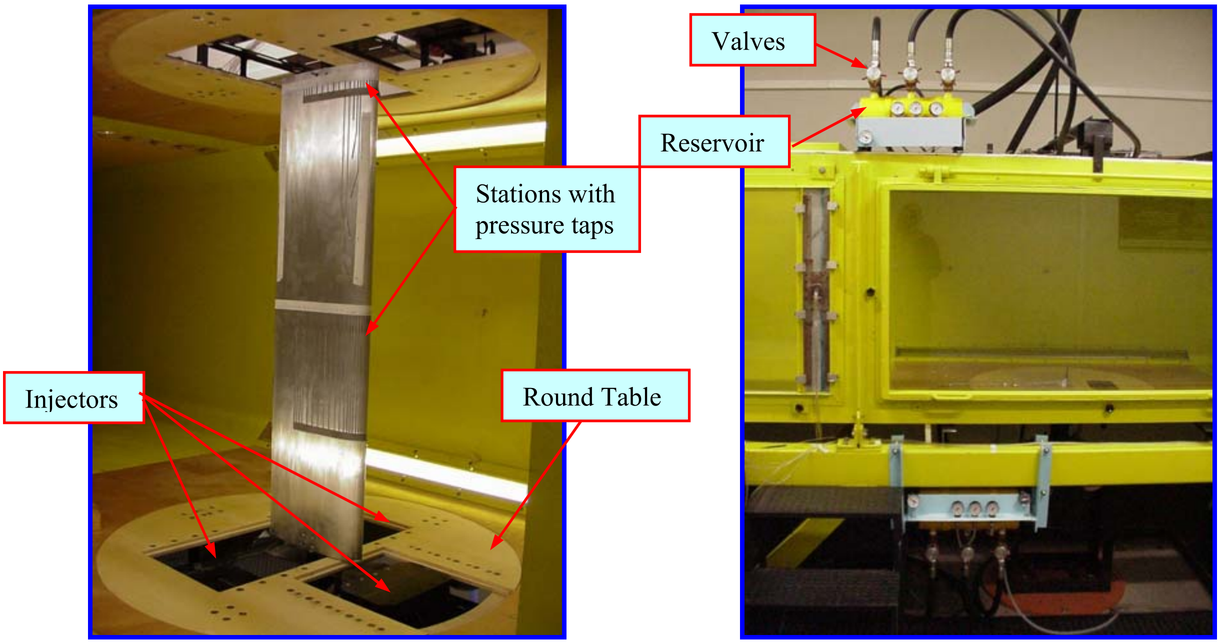
The model used in this research is the three-element airfoil, constituted by a slat, a single slotted flap and the main element, studied by the EMBRAER personnel during conception development phase. Such airfoil was chosen because a strong three-dimensional flow was observed during the experiments conducted in the large facility of IAE-CTA. The airfoil shape can be seen in the figure 2(a) in the configuration used in the present paper, where the flap is positioned at an angle of 35 degrees and the slat is deflected with an angle of 23 degrees. The airfoil, in the deflected configuration, has a 0.36 m chord and approximately 1.0 m width. Such model is mounted in the vertical position inside the wind tunnel test section. A set of 50 pressure taps is distributed along the main element chord, at two stations along the airfoil span (total of 100 pressure taps). One of stations is located at the model middle section and the other is positioned near the upper wall, at a distance of 40 mm. The pressure distributions at these two stations are used to verify the flow behavior along the span, that is, two-dimensional flow is established when the two pressure distributions have a very good correlation. Two additional sets of 7 pressure taps each are distributed at the upper surface, along the model span: one of them was placed near the airfoil leading edge and the other was located in a region near the trailing edge of the main element. The pressure distribution along the span has to be constant if the flow is two-dimensional.

Compressed air injectors are part of a blowing system used to increment the momentum of the boundary-layer on the test section walls, where the model extremities are fixed. In figure 2(b) it is shown the configuration adopted in the present research. Through a lateral duct the high-pressure air enters a cilindrical chamber (lable 1), responsible for the fluid distribution along the device width. After this chamber, air flows to the entrance region of a curved convergent nozzle (2), where the flow velocity is incresead up to the desired value at the exit section of the injector. The injector can be modified by exchanging element labeled 4, resulting in six different plane jets, which interacts with the boundary-layer flow and transfer momentum. The plane jet flow has a 200 mm width. The other jet characteristics can be modifyed and they are a function of: (i) the jet slit, which may be set at 1.0, 1.5 or 2.0 mm, (ii) the jet angle with respect to the tunnel wall surface, which may be adjusted with 10 or 20 degrees and (iii) the compressed air pressure, measured at point labeled 3, which is used to control the velocity of the jet flow generated by the blowing system.



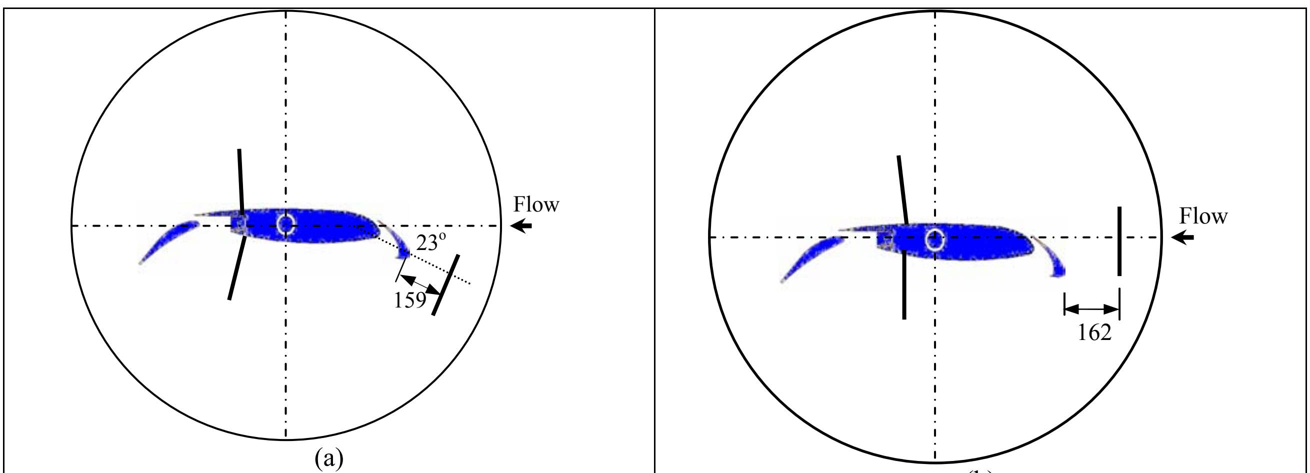
(a) (b)

Figure 2 : (a) Three element airfoil used as a model and its pressure taps. (b) Injector cross section.



(a) (b)

Figure 3: (a) Airfoil mounted inside the test section and (b) part of the blowing system.



(a) (b)

Figure 4: Airfoil and the injector system, assembled in a round table: (a) Conf. # 2 (b) Conf. # 1. (dimensions in mm)

The compressed air is supplied by the following system: the first element is a two stage alternative compressor, with 154 hp power. The compressed air pass through an air dryer, that works based on a refrigeration cicle, and is stored in a 10 m^3 reservoir, whose maximum working pressure is 400 psi (28 atm). After the reservoir, the air flows through a pressure valve, capable to maintain the downstream pressure constant, which works in the range betweem 30 and 10 atm. After that valve, there is a T connected with two flexible tubes, used to distribute the air flux to the injector systems located at the upper and lower walls of the wind tunnel test section, where the model extremities are fixed. Each injector system is constituted by a pressure valve (works in the range between 17 and 1 atm), which is connected to a small reservoir (see Fig. 3b), designed to distribute the compressed air to three injectors. Such air distribution is performed through three flux valves (flow rate is controlled), which is used to control the pressure inside each injector in an independent way and can be visualized in figure 3 (b). Model and the injector system are assembled at two turn tables, located at the upper and lower test section walls. Configurations 1 and 2 are shown in the figure 4.

The pressure measurements are performed by using electronic pressure scanner, containing 32 independent pressure transducers (ESP-32), which are controlled by a personal computer through an interface specially designed to the present research. The analog output of each pressure transducer is connected to a multiplexer (responsible by the scanning process), which is controlled by using a six bit binary code. The multiplexer output is connected to an internal amplifier responsible to provide a full scale output of ±5 VDC. A data acquisition board, capable to make measurements of analog signals, as well as, to send binary codes to digital instruments is used in this research.

The dynamic and static pressures of the undisturbed flow at the wind tunnel test section are measured through a set of pressure taps, connecting to each other forming two rings (wind tunnel anemometric system): one placed at the test section and the other at the settling chamber. In previous work (see Assato at all, 2003) such measurement setup were calibrated by using a standard Pitot tube placed inside the test section and the calibration curves are used in the present work, to correct the dynamic and static pressures. The pressure measurement of each ring is performed by using special pressure transducers, which are connected to signal conditioners and them sent to the data acquisition board.

3. Experimental procedure

Accurate results determination is a function of two aspects: (i) good quality equipment and (ii) well developed experimental procedure. In the present work, modern pressure transducers are used and such equipment was carefully calibrated, by using a Betz manometer. Data acquisition and experiment control is performed automatically by using a computer. Almost all the experimental procedure is programmed, guarantying the repeatability of the results.

In the present investigation computer codes, written in the LabView ambient, were developed to perform the following tasks: (i) calibration of the pressure transducers, (ii) data aquisition during the experiments and (iii) determination of the pressure coefficient (Cp) at each pressure tap and results presentation.

As mentioned before, the static pressure of 110 taps, distributed on the model surface, have to be measured (using 4 ESPs equipment), as well as, the dynamic and static pressures of the wind tunnel flow. Using these parameters the pressure coefficient of each pressure tap can be calculated and the pressure distributions, at each station along the model span, can be obtained and compared.

In the acquisition computer code, the measurements are performed in the following order: (i) the dynamic and static pressures of the wind tunnel flow are the first ones and then (ii) the static pressure connected to each channel of the ESP. This task is accomplished through a loop, where the following sequence of events is performed: (a) the computer sends to the ESP a binary code, responsible to define one of the 32 pressure transducers, (b) a settling time (Δt_t =0.05s) is fixed in order to avoid uncertanties due to voltage transient of the multiplexer operation (responsible to define one of the transducers) and (c) the pressure transducer output is registered during the sampling time (Δt_s) and its value is digitalyzed by using the analog/digital (A/D) converter of the data acquisition board. The above loop is repeated N times ($N \leq 32$) in order to make the measurements of the first N pressure transducers of each ESP used in the experiment. Each loop cicle takes a time equal to $\Delta t_t + \Delta t_s$ and, therefore, all the measurement procedure described above is performed in $\Delta t_s + N(\Delta t_t + \Delta t_s)$.

Once the wind tunnel fluctuations, due to the atmospheric wind variations, cannot be avoided, it is very important to repeat the measurements of the above parameters, in order to obtain an average value for each tap pressure coefficient.

All the parameters, mentioned above, used to control the computer code developed for this investigation were studied before (see Girardi et all, 2005) in order to guarantee results reliability. Considering the better set of these parameters, the scanning process, performed to obtain the average pressure distributions (20 repetitions), takes approximately 100 seconds. At first sight, this value seems very short, but considering the time available to maintain a steady state flow in the injector (approximately 4 minutes) it would be desirable to make measurements in a shorter time, in order to minimize the experiment cost.

The computer code developed to pos-processing the data obtained in the activities described above has two main objectives: (i) by using the data generated by the acquisition and calibration codes, it calculates the average and RMS values of the pressure coefficient of each pressure tap distributed on the model surface and (ii) organize and present data in order to make easy and fast the comparison of the pressure distributions obtained at the two stations along the model span. The airfoil flow behaviour can them be verified with these pressure distributions.

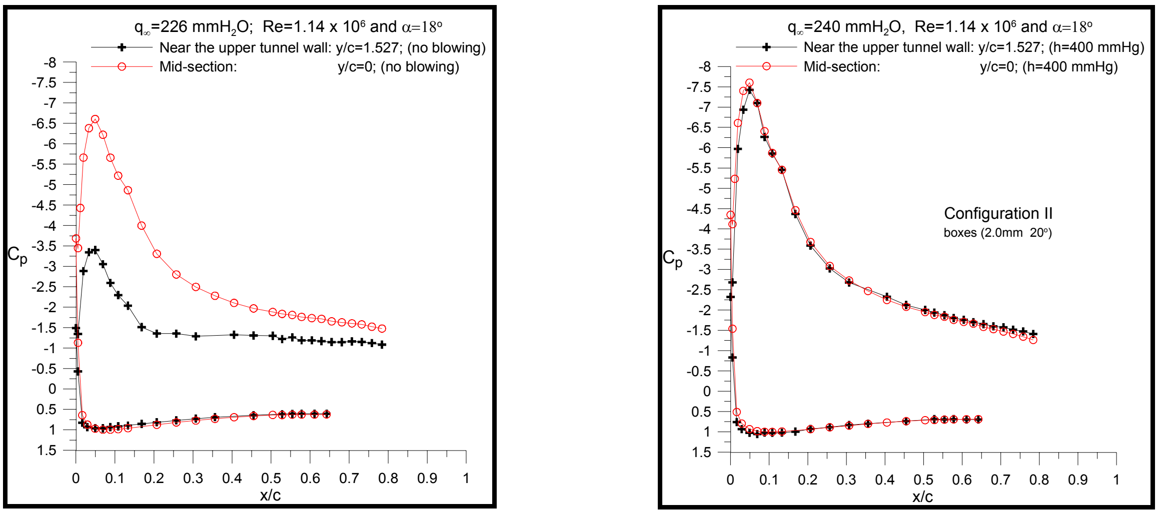
4. Analysis of Results

The average pressure distributions (Cp) along the model chord, at the two stations along the span, are presented in the figure 5. Results obtained with and without blowing are placed side by side in order to allow comparison. The experiments with blowing were performed in an iterative way. In each iteraction, the blower settling chamber pressure (P_{SC}) was set with a specific value and the pressure distributions for the above mentioned stations were compared in order to verify the correlation between them. During iterative procedure P_{SC} is increased up to the value associated to the best correlation, but greater values are always tested to check if better results could be achieved.

For the airfoil positioned with 18^{o} angle of attack (see Fig. 5a), a strong lack of correlation between the pressure distributions measured at the model middle section (Y/C = 0.0) and at the model upper extremity (Y/C =1.527) can be observed when experiments are performed without blowing. At this angle, a very high suction peak in the region of the airfoil leading edge causes a resultant force on the fluid particles inside the wind tunnel boundary-layer, as discussed in the introduction. Due to this force the fluid particles are directed to the airfoil upper surface, causing flow separation at the model extremity regions. Such separation can be made clear by observing the constant pressure coefficient on the upper surface for the station located at the extremity region (Fig. 5a). Very good correlation of the pressure distributions can be verified in the figure 5(b) and these results were obtained when P_{SC} = 400 mm Hg.

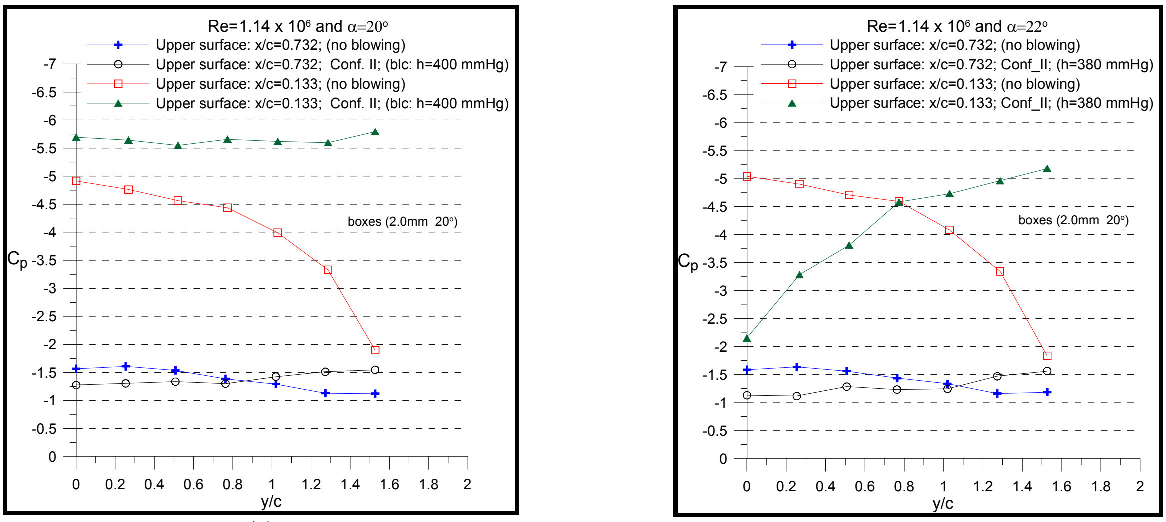
Airfoil middle section is part of the test section symmetric plane and in this case the flow could be considered two dimensional, because there is no velocity component along the model span. Meanwhile, even at such station the Cp distributions are different, depending if the blower is turned on or off, as can be observed by comparing the results presented in figures 5(a) and 5(b). Such differences can be explained in the following way: when the experiment is performed without blowing, the separated regions near the model extremities cause a flow field distortion, and the streamlines are deflected in direction to the middle section plane. The situation becomes similar to an airfoil being tested in a convergent nozzle, where the undisturbed flow velocity is increased along the model chord. The effective angle of attack at each station along the model span is other way to explain the difference pointed above. For the case where blowing is used and two dimensional flow is established, the effective angle of attack is constant along the model span and has the same value of the geometric one. On the other hand, when blowing is not used, the lift coefficient (Cl) along the model span varies and a three dimensional flow, similar to the one observed in a finite wing, is established. As in a finite wing, the effective angle of attack is lower than the geometric one and its value change along the model span. The more negative value for the suction peak observed in figure 5(b), when compared with the figure 5(a), is in accordance to the above explanation.

The results obtained with and without blowing, for the Cp distribution along the model span are presented in the figure 6. The experimental apparatus implemented in this work was capable to achieve the expected results up to the stall angle of attack (20^{o}), as can be observed in the figure 6(a). Without blowing, the Cp becomes less negative while pressure taps are followed from the middle section to the model extremity. Such result is correlated to the Cl variation, as discussed above. When the correct value of P_{SC} is found, and two dimensional flow is established, a uniform Cp distribution can be observed (see Fig. 6a). Although a great amount of effort has been done to obtain two dimensional behavior for α = 22^{o}, this goal could not be achieve, as can be seen in the figure 6(b). The Cp distributions have opposite behavior, when blowers are turned on and off. The flow in the model extremity region is kept attached due to the injected air, while the middle section flow is separated.



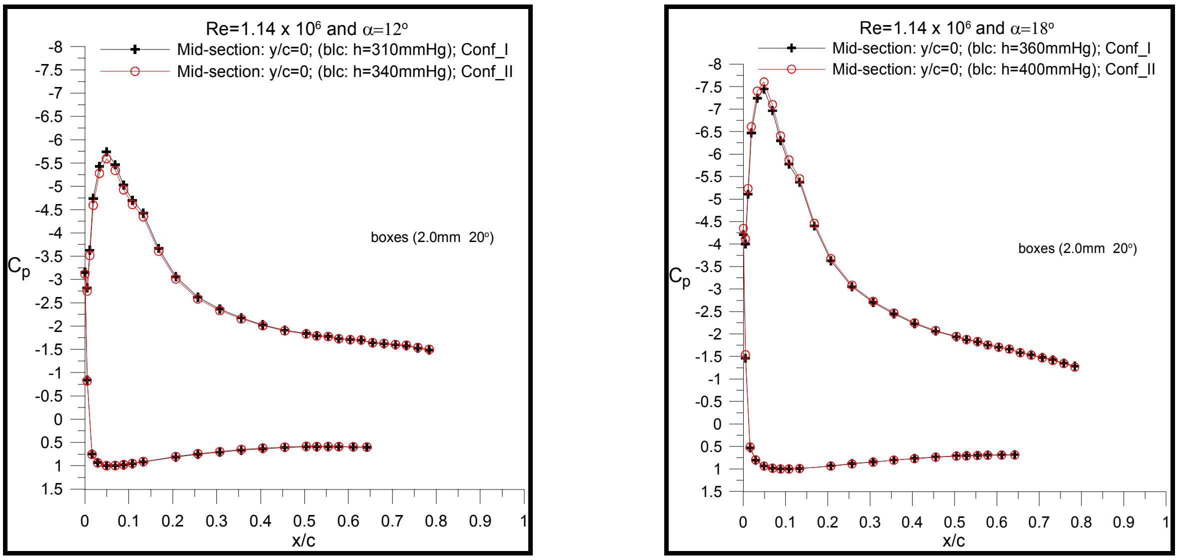
(a) (b)

Figure 5: Cp along the chord at stations Y/C=0.0 and 1.53, for α = 18 degrees: (a) blowing off; (b) blowing on.



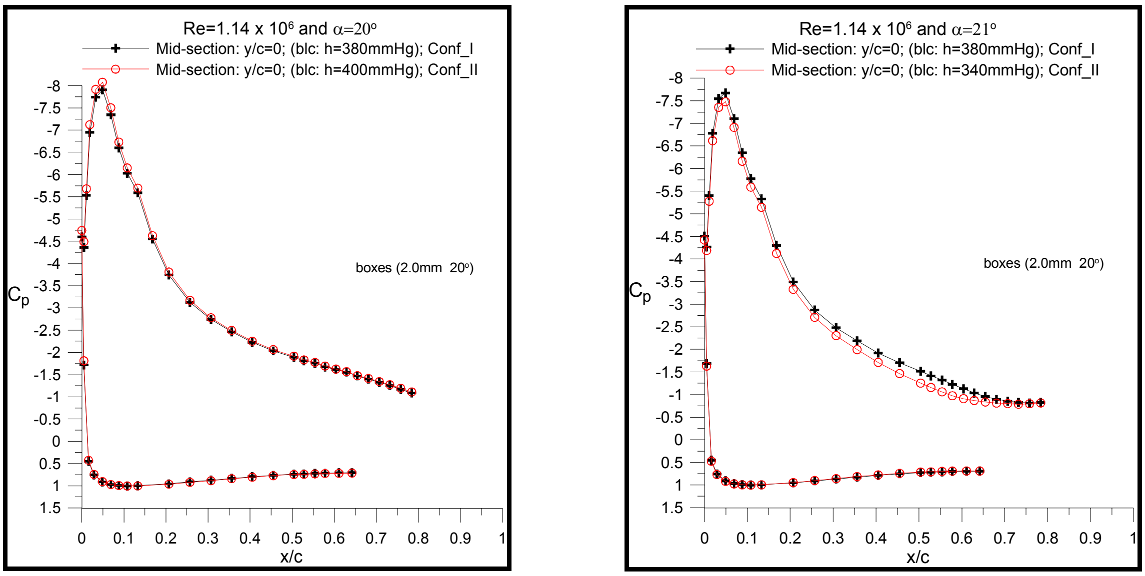
(a) (b)

Figure 6: Effect of blowing (Config. #2) on the Cp distribution along the airfoil span: (a) α = 20^{o}; (b) α = 22^{o}.



(a) (b)

Figure 7: Cp along the chord at station Y/C=0. Comparison between configurations #1 and 2: (a) α = 12^{o} (b) α = 18^{o}.

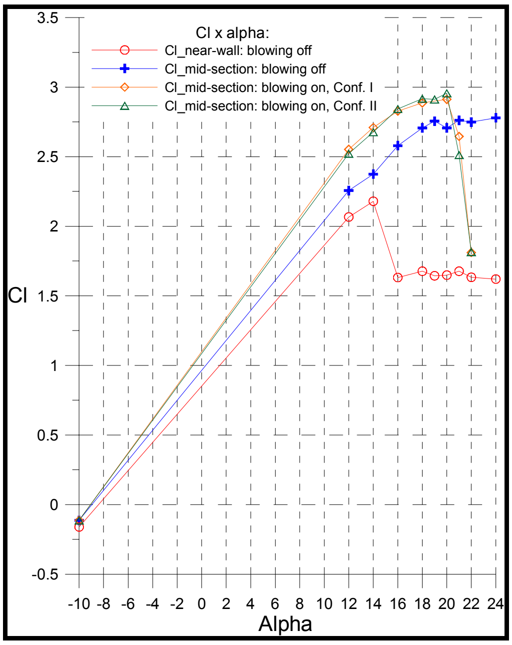


(a) (b)

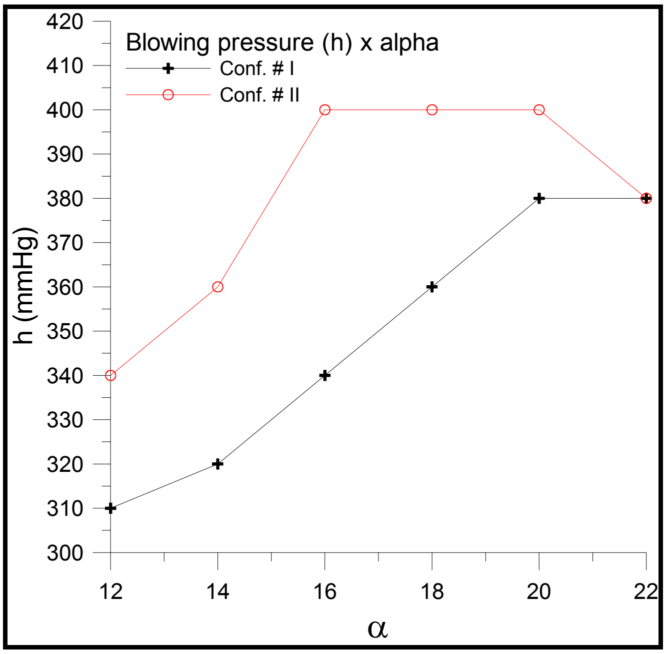
Figure 8: Cp along the chord at station Y/C=0. Comparison between configurations #1 and 2: (a) α = 20^{o} (b) α = 21^{o}.

Comparison between configurations 1 and 2 are presented in the figures 7 and 8. For the airfoil angle of attack up to the stall ($\alpha = 20^o$), no differences are observed in the Cp distribution along the chord for the middle section station, but the injector settling chamber pressure (P_{sc}) has a specific value for each configuration, as shown in the figure 9(b). For airfoil positioned at 21^o angle of attack a small difference can be observed in the figure 8(b). A larger separated region can be observed to the configuration #2. Such difference can not be explained by the authors and it is difficult to choose one of them as the correct one. To solve this problem more research has to be done and probably other parameters have to be measured as well as flow visualization techniques have to be developed (PIV methodology could be useful).

The lift coefficient (Cl) was obtained by integrating the Cp distributions of the model main element, considering the airfoil shape and the pressure taps positions, showed in the figure 2(a). Due to the three dimensional flow, the values obtained to the two stations are different, when the experiments are performed without blowing, as can be seen in the figure 9(a). At the model extremity station, the airfoil stall occurs for $\alpha=14^o$, one the other hand, the stall angle is very difficult to be defined to the middle section station. The correct stall angle is 20^o, as can be observed in the curves obtained with blowing. Comparing results obtained with configurations 1 and 2, at the model middle section, it is possible to verify a very good correlation to angles of attack up to stall. At 21^o, an appreciated difference is observed for the Cl results, which is related to the Cp distribution discrepancies observed in the figure 8(b). The pressure inside the injector settling chamber has to be increased while the model angle of attack is incremented, as shown in the figure 9(b). This result seems to be associated to the pressure gradient increment established in the region near the airfoil upper surface (more negative suction peak is observed for greater α values). In order to compensate the pressure force, the momentum furnished by the blowers, to the fluid particles inside the wind tunnel boundary-layer, has to be increment and this is accomplished by increasing the above mentioned pressure. The results, presented in the figure 9(b), show configuration # 1 is less expensive, because lower values of P_{sc} are required in order to minimize the three dimensional flow along the model span. Minimization of the P_{sc} values is one of the goals of the present research program, because the time available to execute the experiment is inversely proportional to the value required of the P_{sc}.



(a)



(b)

Figure 9: Comparison between configurations 1 and 2: (a) Lift coefficient (Cl) of the model main element and (b) Injector settling chamber pressure as a function of airfoil angle of attack.

5. Final Remarks

In the experimental apparatus developed to this research program, a large number of parameters can be varied with the objective to obtain true two dimensional flow over an airfoil. In the present paper, the results obtained to two configurations are compared. The basic difference between them is the compressed air jet direction, relative to the wind tunnel undisturbed flow. For configurations # 1, both air flows are aligned and this pattern is kept constant while the airfoil angle of attack is changed. In the configuration # 2, the jet direction follows the airfoil angle of attack variations and, therefore, the jet flow direction is no longer aligned to the undisturbed flow. The injector used in both configurations is the same and it is characterized with the following parameters: (i) localized at some distance upstream from the airfoil leading edge, at each test section wall, (ii) the injector exit section is 2.0 mm thick and (iii) the jet produced by the injector has an inclination of 20^o, related to the tunnel wall.

Up to the airfoil stall angle, a perfect correlation for both configurations is observed to the pressure coefficient distribution at the model middle section. On the other hand, in order to obtain two dimensional flow over the model, the injector settling chamber pressure is lower for the configuration # 1. Therefore, considering the experiment cost, such configuration seems to be better. It is worth to remember that greater values of the blower pressure is related to greater mass flux rate and, as a consequence, lower time to run the experiment, because the compressed air resources (associated to the reservoir volume and maximum pressure it resists) are limited. During the experiments conducted in this work a maximum run time of approximately 4 minutes was possible and this is the time required to perform the experiment to just one angle of attack.

As mentioned in the introduction, the present research results will be transferred to a larger wind tunnel, where EMBRAER airfoils will be tested and developed. Due to the larger dimensions and velocity a greater amount of compressed air resources will be necessary. In this case, experiment feasibility is compromised and cost could reach high values. Due to the above discussion, the research program is now oriented to find a configuration capable to generate good flow characteristics for two dimensional tests of airfoils and to minimize the cost associated to the experiment. Such goal will be accomplished by testing other configurations resulting from the above parameters variations or with the introduction of passive elements, such end plates or fences, that could work together with some blowing.

6. Acknowledgement

To the Fundação de Amparo a Pesquisa do Estado de São Paulo (FAPESP), for supporting part of the resources used to built the wind tunnel (project: FAPESP 00/13769-0). To EMBRAER, through Eng. Paulo Komatsu and his team, for supporting the construction of model and blowing system. To Prof. Fernando Catalano, from USP-SC, responsible for the model design. To the staff of the Prof. K.L. Feng Aeronautical Engineering Laboratory: Carlos Guedes Neto, Luis Zambrano Lara, Vitor Valentim Betti and Mario Correia.

7. References

Assato, M; Fico Jr, N.G.C.R.; Girardi, R.M. (2003) "Test Section Pressure Measurements at the Research Wind Tunnel of the Aeronautical Institute of Technology". In: 17TH International Congress of Mechanical Engineering, (COBEM 2003), São Paulo.

Assato, M., Fico Jr, N.G.C.R & Girardi, R.M., 2005a, "*Calibration of a Boundary-Layer Control System for use in a Low-Speed Wind Tunnel*", Paper AIAA-2005-4977, 23st AIAA - Applied Aerodynamics Conference, 6-9 Jun, Toronto, Canada.

Assato, M., Girardi, R.M. & Fico Jr, N.G.C.R, 2005b, "*Evaluation of a Boundary-Layer Control System by Air Blowing*", 18th COBEM – International Congress of Mechanical Engineering, November 6-11, 2005, Ouro Preto/MG, Brazil (to appear).

Elsenaar, A., (1983) "Wall blowing requirements for 2-D high lift testing in pessurized wind tunnels", National Aerospace Laboratory, NLR, The Netherlands, NLR Memorandum AC 83-005 U".

Girardi,R.M.; Fico Jr., N.G.C.R.; Mello, O.A.F. & Komatsu, P. (2002) "Design of the Research Subsonic Wind Tunnel of the aeronautical Institute of Technology", Proceedings of the Mechanical Engineering National Congress (CONEM 2002), João Pessoa, P.

Girardi, R.M.; Fico Jr, N.G.C.R; Mello, O.A.F; Komatsu, P. (2003) "Research Wind Tunnel of the Aeronautical Institute of Technology: Manufactoring Aspects and Design Details". In: 17TH International Congress of Mechanical Engineering, (COBEM 2003), São Paulo.

Girardi, R.M.; Assato, M.; van der Werff, D. (2005) "Wind Tunnel test of a three-element airfoil configuration: Development of Experimental Apparatus and Procedure required for obtaining reliable results". 18th COBEM – International Congress of Mechanical Engineering, November 6-11, 2005, Ouro Preto/MG, Brazil (to appear).

Labrujère, Th.E., (1984), "Correction for wall interference by means of measured boundary condition method", , National Aerospace Laboratory, NLR, The Netherlands, NLR TR 84114 U.

Pessoa, G. A & Girardi, R.M. (2004) "Escoamento Tridimensional sobre Modelos Bidimensionais em Túnel de Vento". Congresso Nacional de Engenharia Mecânica (CONEM 2004), Belém, PA, Ago

Rae Jr., W.H. & Pope, A., (1984), "Low-Speed Wind Tunnel Testing", John Wiley & Sons, Second Edition, USA.

7. Responsibility notice