Mecánica Computacional Vol. XXII
M. B. Rosales, V. H. Cortínez y D. V. Bambill (Editores)
Bahía Blanca, Argentina, Noviembre 2003.

NUMERICAL SIMULATION OF A BLOWING BOUNDARY-LAYER CONTROL SYSTEM

Marcos Soares[*], Nide G C R Fico Jr[†], Marcelo Assato[†] and Roberto M. Girardi[†]

[*] Empresa Brasileira de Aeronática S. A. - EMBRAER
Av. Brigadeiro Faria Lima, 2170, 12227-901, São José dos Campos – SP – Brasil
TE: 55-12-3927-1000 Fax: 55-12-3921-2394
e-mail: marcos.soares@embraer.com.br, web page: http://www.embraer.com.br/

[†]Divisão de Engenharia Aeronáutica
Instituto Tecnológico de Aeronáutica
Praça Marechal Eduardo Gomes 50, Vila das Acácias, 12228-900, São José dos Campos, SP, Brasil
TE: 55-12-39475823 FAX: 55-12-39475024
e-mail: nide@aer.ita.br, web page: http://www.ita.br

Key words: Wind Tunnel, Boundary-layer control, Numerical Methods.

Abstract. *The Instituto Tecnológico de Aeronáutica, ITA, in a joint with EMBRAER and the Instituto de Aeronáutica e Espaço (IAE-CTA) has just finished the construction of a low-speed open circuit wind tunnel. The new tunnel is inserted into a Project of Technological Innovation supported by FAPESP (www. fapesp.br). The tunnel is now under calibration. Just after its calibration the new wind tunnel will be furnished with a boundary-layer control system. It will be a blowing system. This system usually has more than one air jet blowing into the model region. Its main function is to keep the flow two-dimensional over airfoil models. There are many variables involved in such a problem. There is need, for example, to determine the position of the air jets in respect to the airfoil and their angle. In order to access all the relevant parameters an initial two-dimensional numerical simulation is under way. The numerical simulation of the flow, inside the wind tunnel, in the blowing region will be performed using the commercial software FLUENT that has a history of great success in several aeronautical applications at EMBRAER. As a result of such an effort it will be possible to fine tune the design of the boundary-layer control system*

1 INTRODUCTION

The Instituto Tecnológico de Aeronáutica (ITA) designed and built [1] a new Research Open-Circuit Low-Speed Wind Tunnel, see Figs 1 and 2. ITA's wind tunnel conceptual design was reported by Girardi et all [2], where the solutions adopted for this project are discussed in detail. The new wind tunnel is part of a greater project whose main objective is to increment the productivity and reliability of aerodynamic testing. The Instituto de Aeronáutica e Espaço (IAE-CTA) and the University of São Paulo at São Carlos (USP-SC) as well as Empresa Brasileira de Aeronáutica (EMBRAER) are also involved in this effort. IAE-CTA has closed-circuit subsonic wind tunnel with a 7x10 feet square test section. While the USP-SC wind tunnel has a square test section with a 0.6 m side and 1.5 meters long, the maximum velocity is 50m/s. Both ITA's wind tunnel and the one at USP-SC are important tools in the development of new experimental procedures, which later can be applied to the bigger IAE-CTA's facility. FAPESP and EMBRAER are partners in the sponsorship of the project.

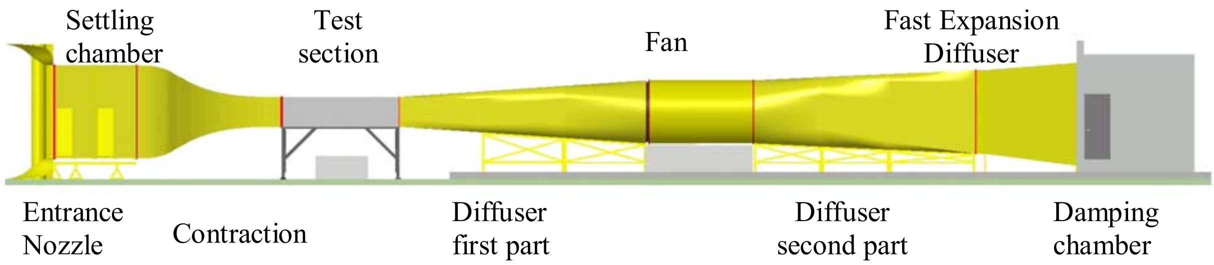

Figure 1: General view of the ITA's teaching and research wind tunnel and its elements.

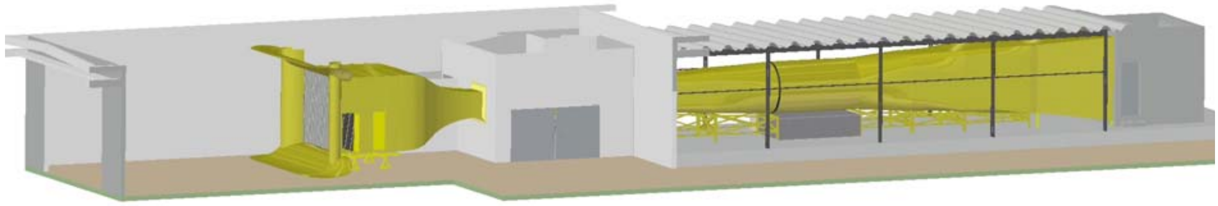

Figure 2: Wind tunnel inside ITA's Prof. K.W. Feng Aeronautical Engineering Laboratory.

ITA's new wind tunnel is the answer to the long-term need of equipment, which allows aerodynamic testing at low operating cost as well as low cost for the experimental apparatus implementation, it is important to remember that complex models manufacturing can be very expensive. The achieve the compromise of having a low-cost facility as well as generating useful results for EMBRAER's engineering needs it was decided to build a wind tunnel having a 1.0 x 1.28 m test section. The required maximum flow velocity, at the test section is, at least, 70 m/s which correspond to a Reynolds number of 10^6.based on a 0.30m characteristic length. This new testing facility, equipped with a 200 hp electric motor, demands much less power than the IAE-CTA wind tunnel, where part of the aerodynamic tests are performed to the Brazilian aeronautical industry and, in particular, to the EMBRAER. At the IAE-CTA wind tunnel the maximum velocity is approximately 100 m/s, along it's 2.1 x 3.0 m test section, requiring 1200 hp of power. Comparison of these figures justifies the need of ITA's wind tunnel as a complement to the IAE-CTA facility.

The investigation of the aerodynamic behavior of a wing section is frequently obtained through wind tunnel tests of airfoil models. These models are so called "two-dimensional" due to the fact that they spam from wall to wall inside the test section. Thus, at least ideally, the flow is normal to the body span wise direction. In reality this is not the case because of the interference of the boundary layers along the tunnel walls with the one that develops along the airfoil model. During the experiments these ends effects induce premature boundary-layer separation over the model in the region that it is fixed into the tunnel walls. In turn, the ideal 2-D lift coefficient is degraded Flow visualization indicates that the flow over the airfoil model is far from being two-dimensional. The greater the angle of attack the more pronounced these effects become. This departure from the bi-dimensionality brings about erroneous results and therefore is highly undesirable. In particular the EMBRAER engineers have a very hard time to determine a very important airfoil parameter, namely, the maximum lift coefficient. As a consequence of the flow tri-dimensionality stall occurs first at the airfoil sections close to the tunnel wall. Thus the overall lift coefficient measured is smaller than the true 2-D value. In this context, it may be found in the literature several methodologies to control the boundary layer over the 2-D model and minimize the flow tridimensionality, namely [5]: (i) the use of end plates, (ii) boundary-layer blowing or suction and (iii) use of vortex generators.

ITA's wind tunnel started operating on February 2003 with a series of experiments to calibrate the test section flow. This work is still going on [3,4]. After this phase, two research programs will be implemented to reach the following objectives: (i) Experimental methodology development to minimize the three-dimensional flow observed in two-dimensional airfoil models at high angles of attack. This problem occurs due to the interaction between the airfoil extremity and the tunnel wall boundary-layer flow and cause great uncertainty in the measurements of the airfoil Cl_{max}. (ii) Development of a methodology for estimating a wing Cl_{max}, once the airfoil Cl_{max} is known. In order to accomplish this objective a set of experiments will be conducted to understand the separated flow evolution, at the upper surface of a wing, while the angle of attack is incremented up to the wing stall. It is worth to mention that these two research programs were proposed by the EMBRAER personnel to solve important practical problems. In this context it is very important to control the boundary layer over the airfoil model in order to ensure 2-D flow over most of it. The blowing devices are located at the top and bottom walls of the test section as seen in Fig. 3.

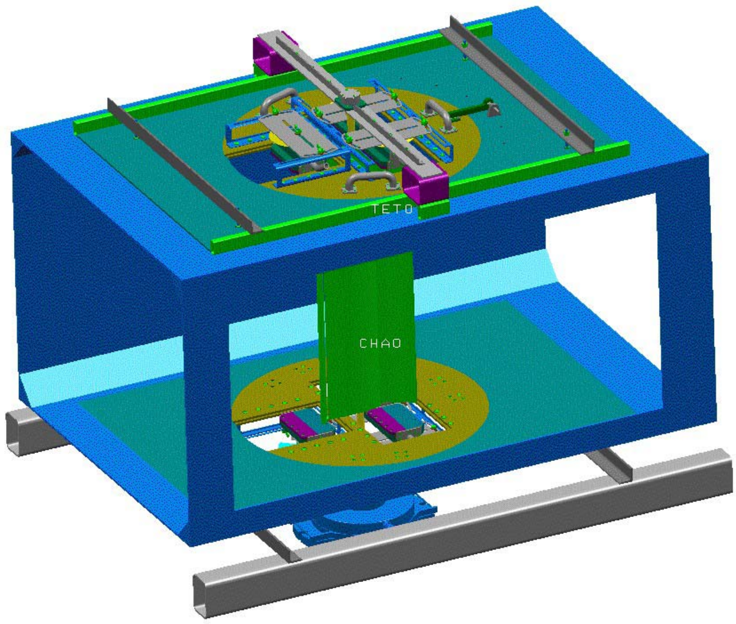

Figure 3: Blowing slots located at the top and bottom walls of the test section.

Notice that the slots as well as the airfoil are fixed onto a turntable in order to allow modification in the angle of attack. The mechanism was designed in such a way that the slot angle and its position relative to the airfoil can also be slightly changed if necessary. Although this flexibility might come in handy for a fine adjustment during the tests one should bare in mind that there is no room for significant modifications. Should they be needed, a whole new piece of equipment will have to be built. Consequently, the project will be penalized with a time delay and also a waste of money. The best location for the blowing slots depends on many variables. There is a need, for example, to determine the position of the air jets in respect to the airfoil and their angle in respect to the airfoil chord. In order to access all the relevant parameters an initial two-dimensional numerical simulation was undertaken. The numerical simulation of the flow, inside the wind tunnel, in the blowing region was performed using the commercial software FLUENT that has a history of great success in several aeronautical applications at EMBRAER. Therefore, the objective here is to report these initial 2-D results. Such results will guide the study into a 3-D numerical simulation. As a consequence of the overall effort it will be possible to fine tune the design of the boundary-layer control system thus minimizing technical risks.

2 BRIEF DESCRIPTION OF THE NUMERICAL METHOD

The flow field was numerically simulated with the aid of the well-known CFD commercial package FLUENT 6.0 [6]. The equations are discretized in an unstructured grid context using a Finite Volume algorithm. Each equation was solved independently that is in a segregated fashion and was used the Implicit formulation. The turbulent momentum transport was accounted by the Realizable *k*-*ε* model, more applicable for complex behavior like jet impingement, separating flows, secondary and swirling flows. The Standard Wall Functions was adopted for the near-wall treatment because is robust, economical and has reasonably accurate. All cases were performed at steady state.

3 RESULTS

In the present study both the slot width and the slot angle in respect to the tunnel's centerline were varied. For all cases the flow velocity at the test section entrance was fixed at 50m/s. The mass flow rate inside the blower device was kept equal to 0.6 Kg/s. These values were chosen because they are typical of the tunnel-operating regime. Table 1 summarizes all the cases that were numerically simulated.

Table 1: Cases analyzed

Flow Velocity at the test section inlet plane = 50 m/s Blower mass flow rate = 0.6 Kg/s		
	Blower Angle (degree)	Slot Width (mm)
Case 1	10	1
Case 2	10	2
Case 3	20	1
Case 4	20	2

Figure 4 shows a typical computational mesh used for the current simulations. Typically, around 7,000 points were used. Note that it extends all the from the blower inlet, where high-pressured air is supplied, to its outlet at the tunnel test section. Regions of high gradients, close to the tunnel walls and at the blower exit, were adequately discretized.

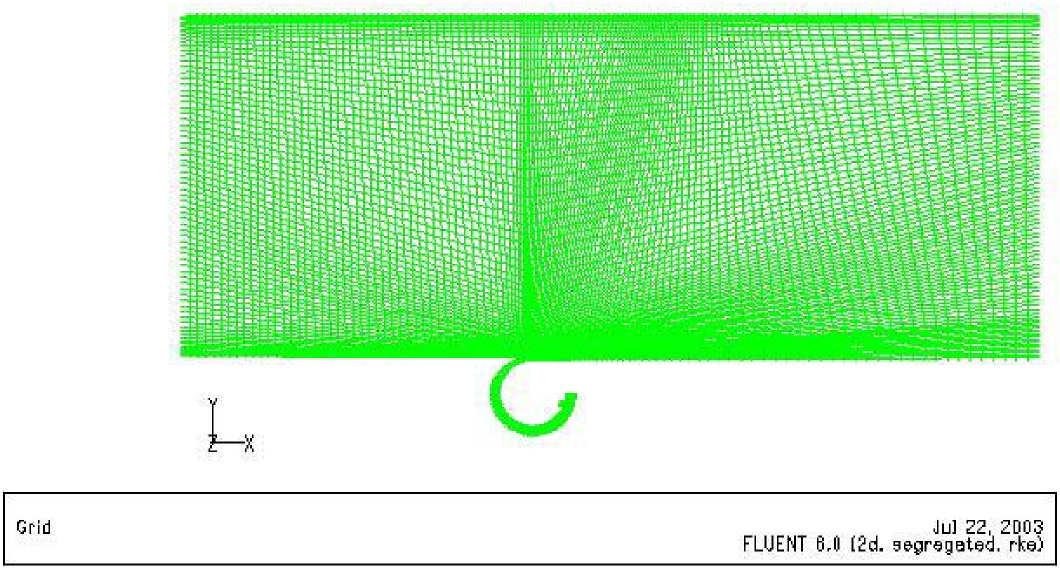

Figure 4: Typical computational grid.

The velocity contours associated with case 1 are shown in Fig. 5. The velocity is fairly constant over the whole computational domain except in the region close to the wall downstream of the blower. The air jet emanating from the slot diffuses into the tunnel main stream therefore increasing its width and decreasing its velocity. The momentum increase due to the blower air jet may be observed looking at Fig. 6. The velocity profile upstream of the slot exit shows the presence of a boundary layer, see red diamonds, but at the slot exit the velocity has a peak near the wall, see black diamonds, indicating an energizing of the low-speed boundary layer flow. Figure 7 shows the velocity profile evolution for case 2, that is the air jet angle is 2 degrees in respect to the tunnel center line, the same as in case 1, but the slot width is now 2 mm instead of 1 mm. It can be seen comparing Figs 6 and 7 that the larger air jet width contributes for a more significant overshoot of the velocity vector magnitude close to the tunnel wall. The maximum velocity associated with case two is over 120 m/s, see Fig. 7, while for case one this value was just under 110m/s.

Figure 8 shows that at a distance equal to 0.20m downstream of slot exit the velocity profile tends to become more uniform although the velocity peak is more pronounced. In the context of the present work this information is very interesting as it signalizes the position the air blower should be placed along the airfoil chord to achieve the desired flow bi-dimensionality at a specific location.

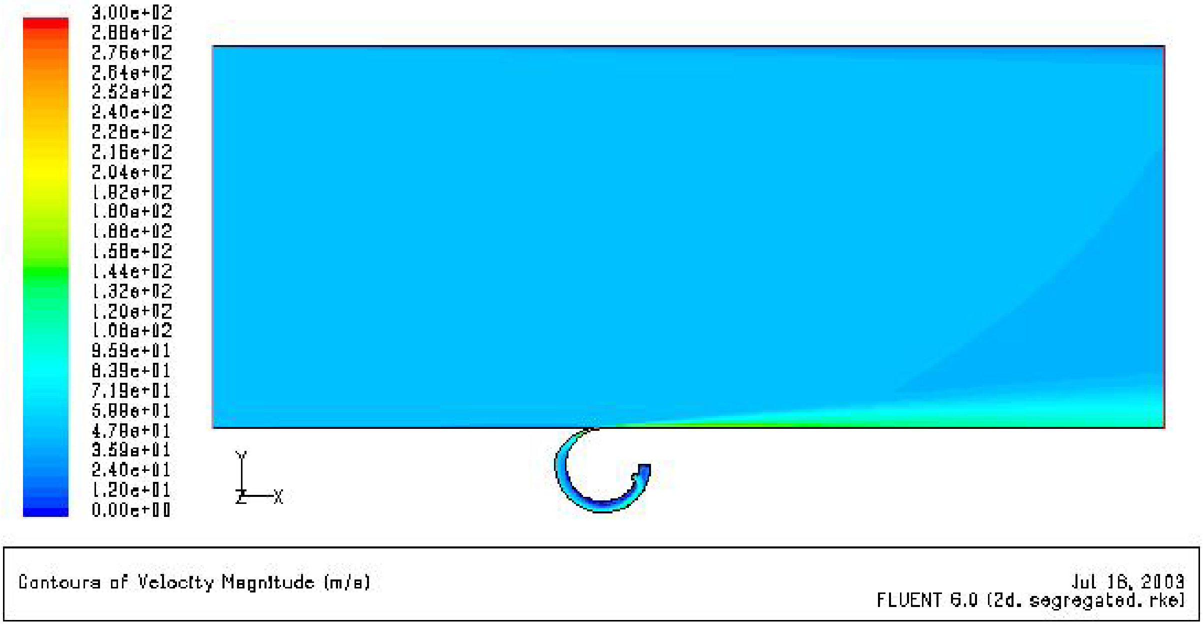

Figure 5: Velocity contours for case 1.

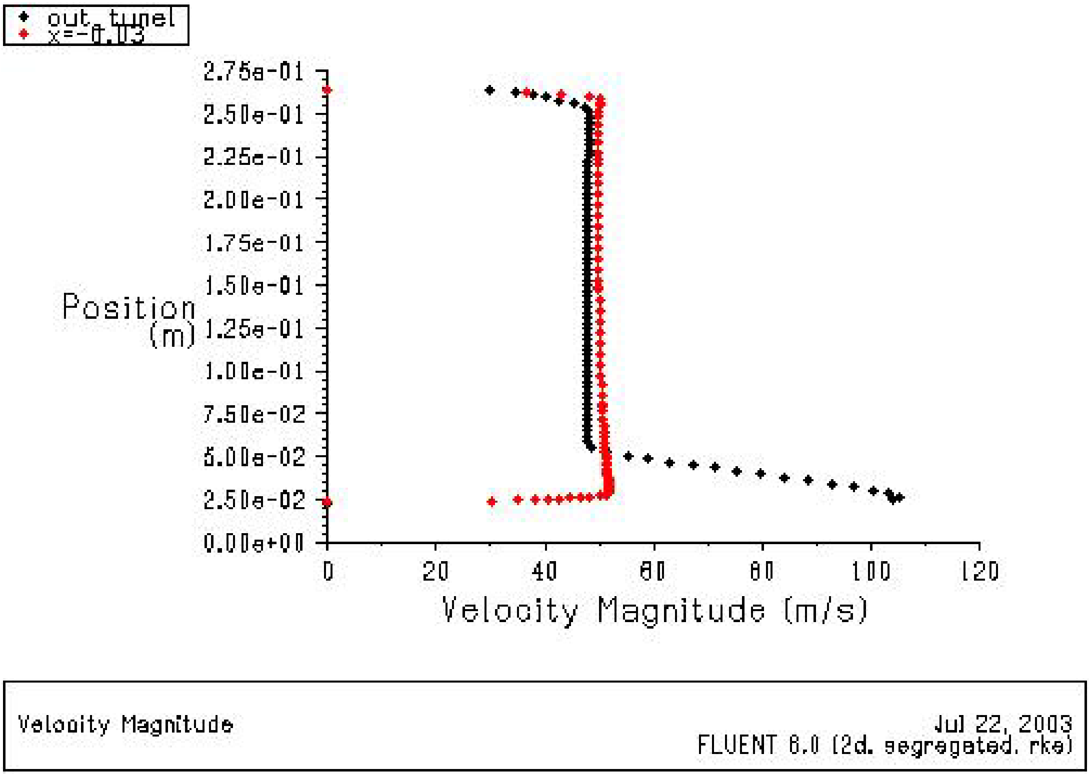

Figure 6: Velocity profiles for case 1.

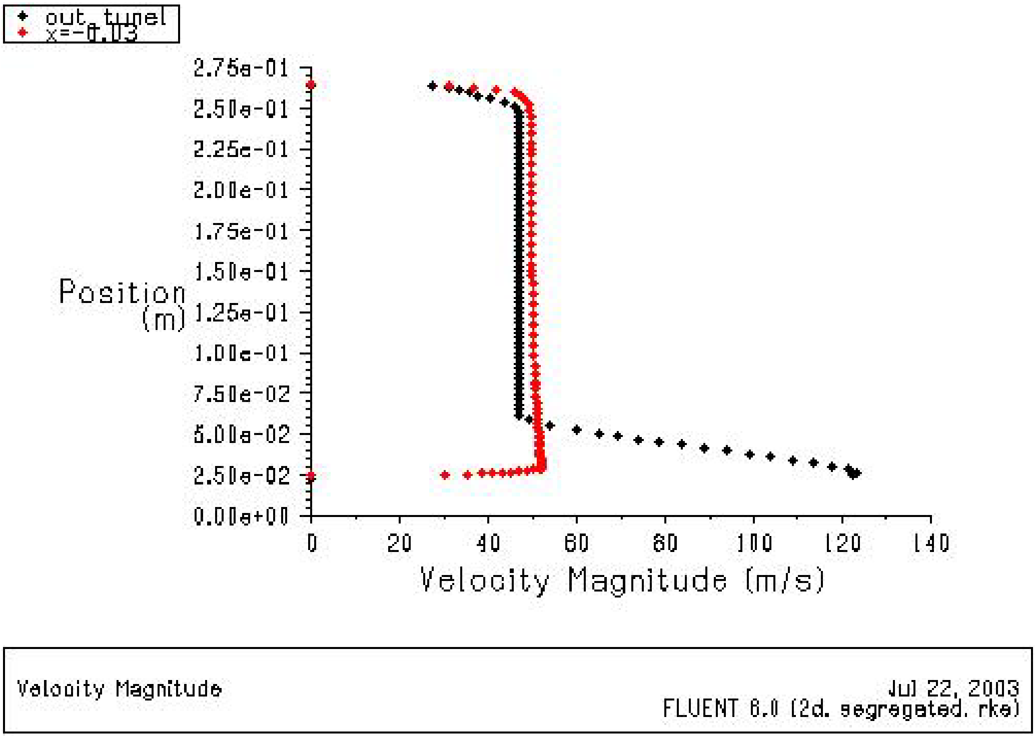

Figure 7: Velocity profiles for case 2.

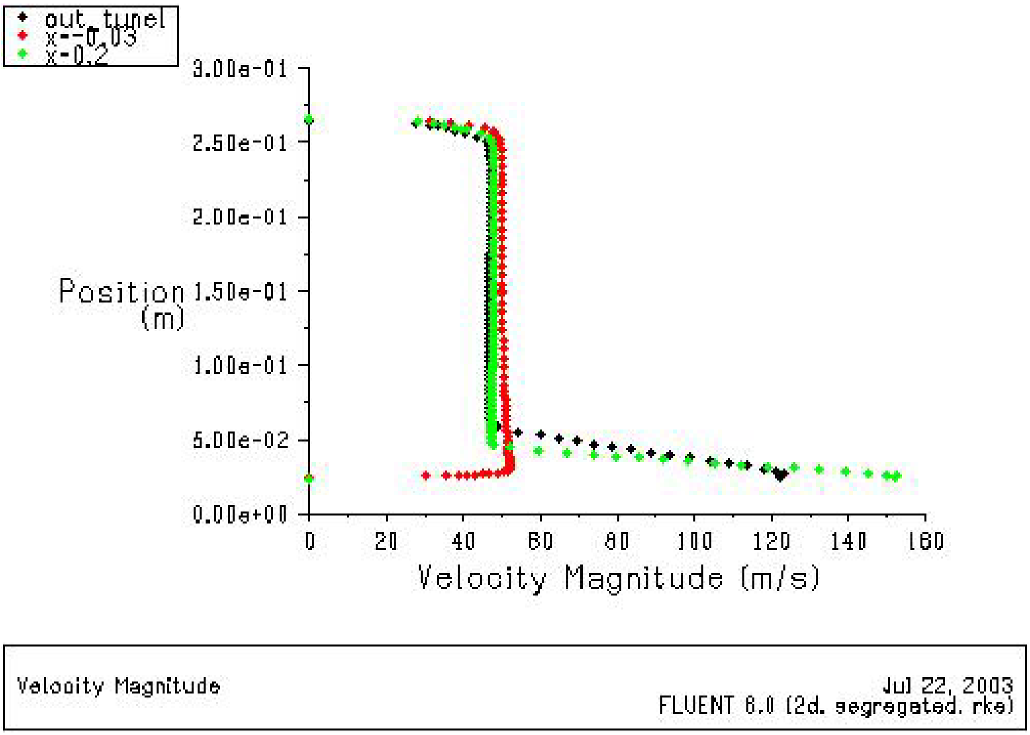

Figure 8: Velocity profiles for case 2.

Figures 9 through 11 refer to case 3. Velocities profiles along the test section, at different stations with respect to slot exit are shown. Figure 9, as Figs. 6 and 7, show velocities profiles upstream and at the blower exit. Comparing the results in Fig 9 with those presented in Fig 6 it is possible to asset the influence of the air jet angle upon the test section flow. The greater angle of case 3 has a definite impact on the maximum velocity magnitude at the slot outlet. Notice that in Fig 9 this value is just shy of 140 m/s while in Fig 6, case 1, it does not reach 110m/s. Figures 10 and 11 display the evolution of the velocity profile along the tunnel. As expected the velocity peak near the wall diminishes as the flow progresses downstream. On the other hand a very mild non-uniformity of the velocity profile is observed as the distance from the blower increases.

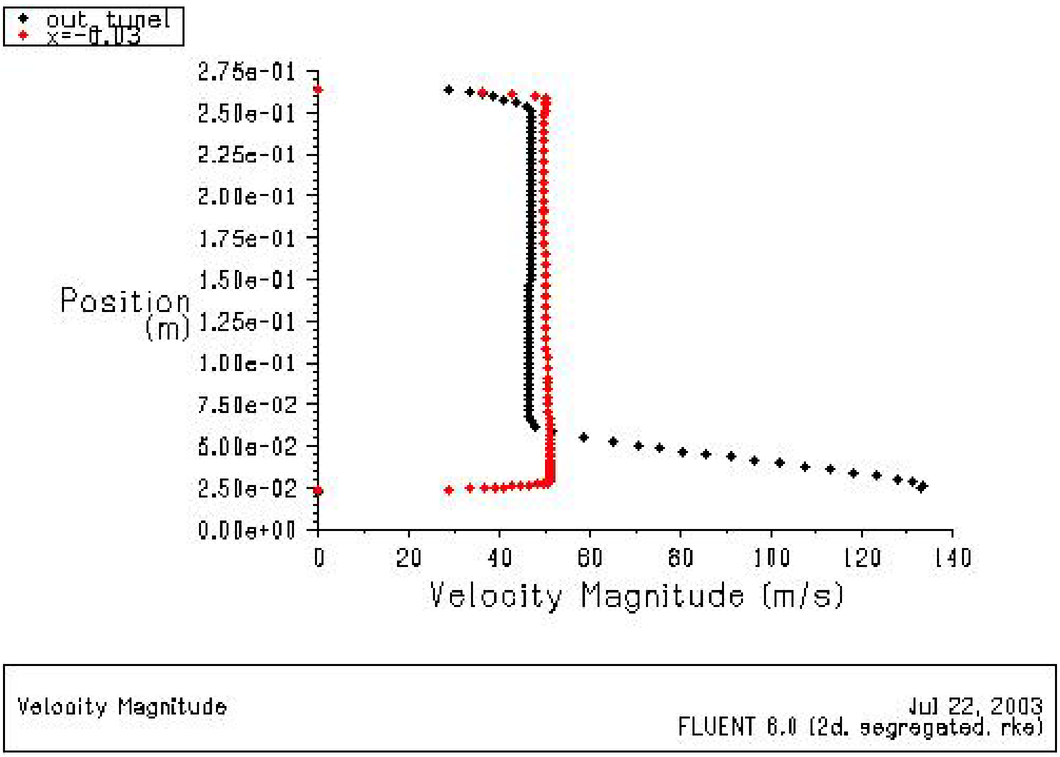

Figure 9: Velocity profiles for case 3.

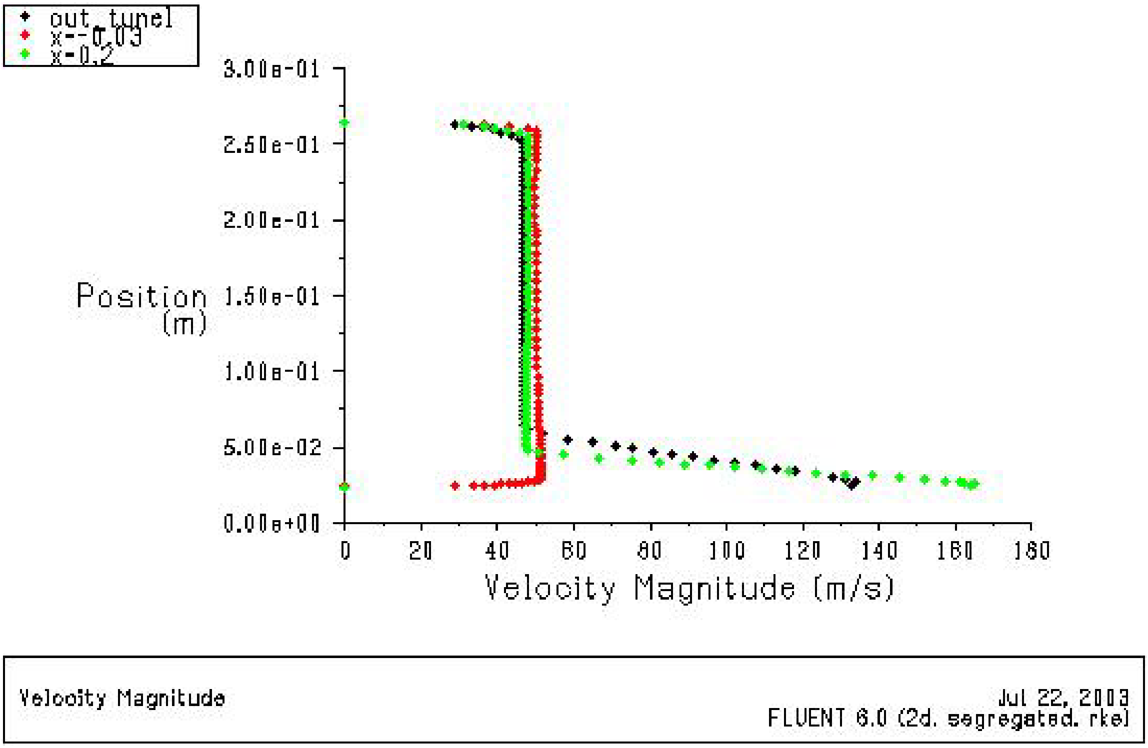

Figure 10: Velocity profiles for case 3.

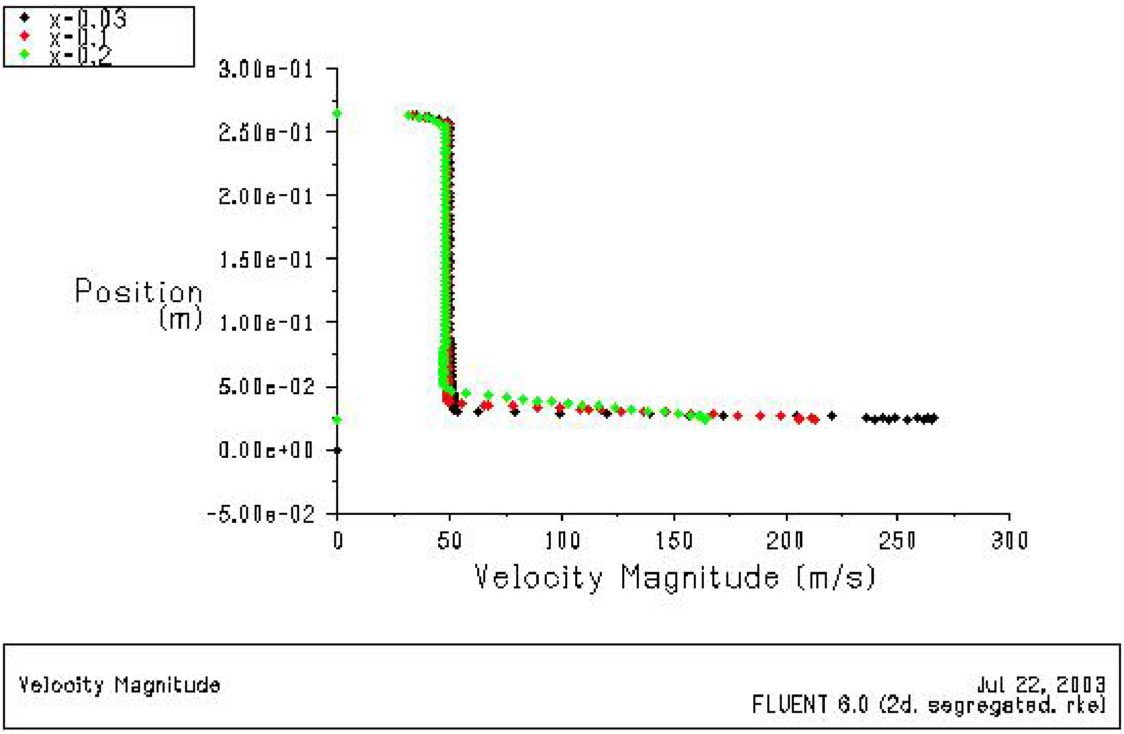

Figure 11: Velocity profiles for case 3.

Figures 12 and 13 refer to case 4. Figure 12 shows velocities profiles upstream and at the blower exit. Figure 13 shows velocities profiles along the test section, at different stations with respect to slot exit. Comparing these two Figs. 12 and 13 with Figs. 9 and 10, respectively, it is possible to infer the influence of the slot width since both cases 3 and 4 the slot angle was kept constant. The maximum velocity at the station x=0.2, that is 0.2 meter downstream of the slot, is larger for the wider slot (case 4). Apparently, the thicker jet is capable of increasing more the boundary-layer momentum than the 1mm slot of case 3. It must be remembered that all cases were run for the same mass flow rate both for the tunnel as well as for the blower.

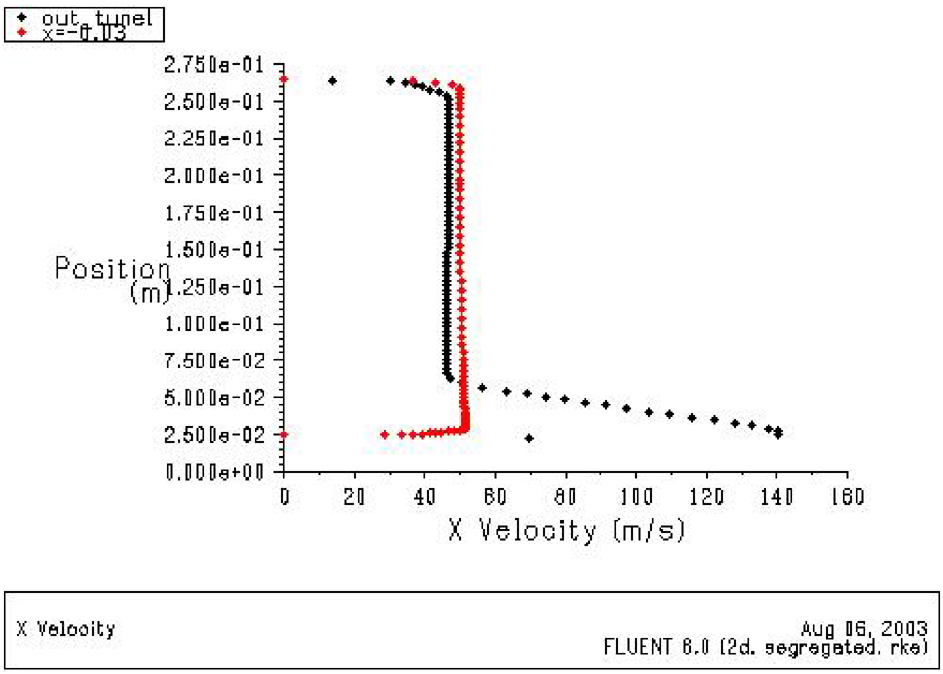

Figure 12: Velocity profiles for case 4.

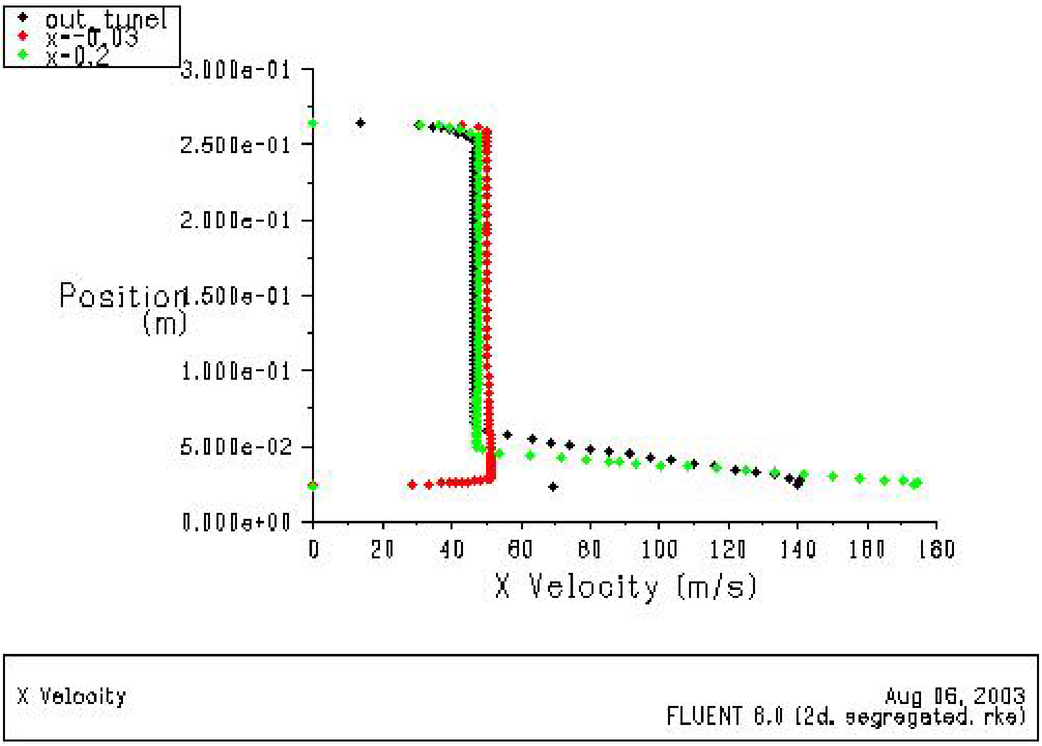

Figure 13: Velocity profiles for case 4.

Figure 14 show a close-up view of the blower exit for all cases studied. For case 1 the blower angle is 10 degrees and the slot width is 1 mm. It is possible to see that the high velocity flow region at the blower exit is very thin and seems not to interfere with the tunnel mains flow stream. When the slot width is doubled, but the jet angle is kept at 10 degrees its zone of influence is a bit enlarged. Cases 3 and 4 are associated with a higher jet angle, namely, twenty degrees. In these instances it is clear that the oncoming, flow along the tunnel test section, feels the air jet emanating from the blower. Note, for example, the darker blue just upstream of the slot exit indicating a local deceleration of the flow.

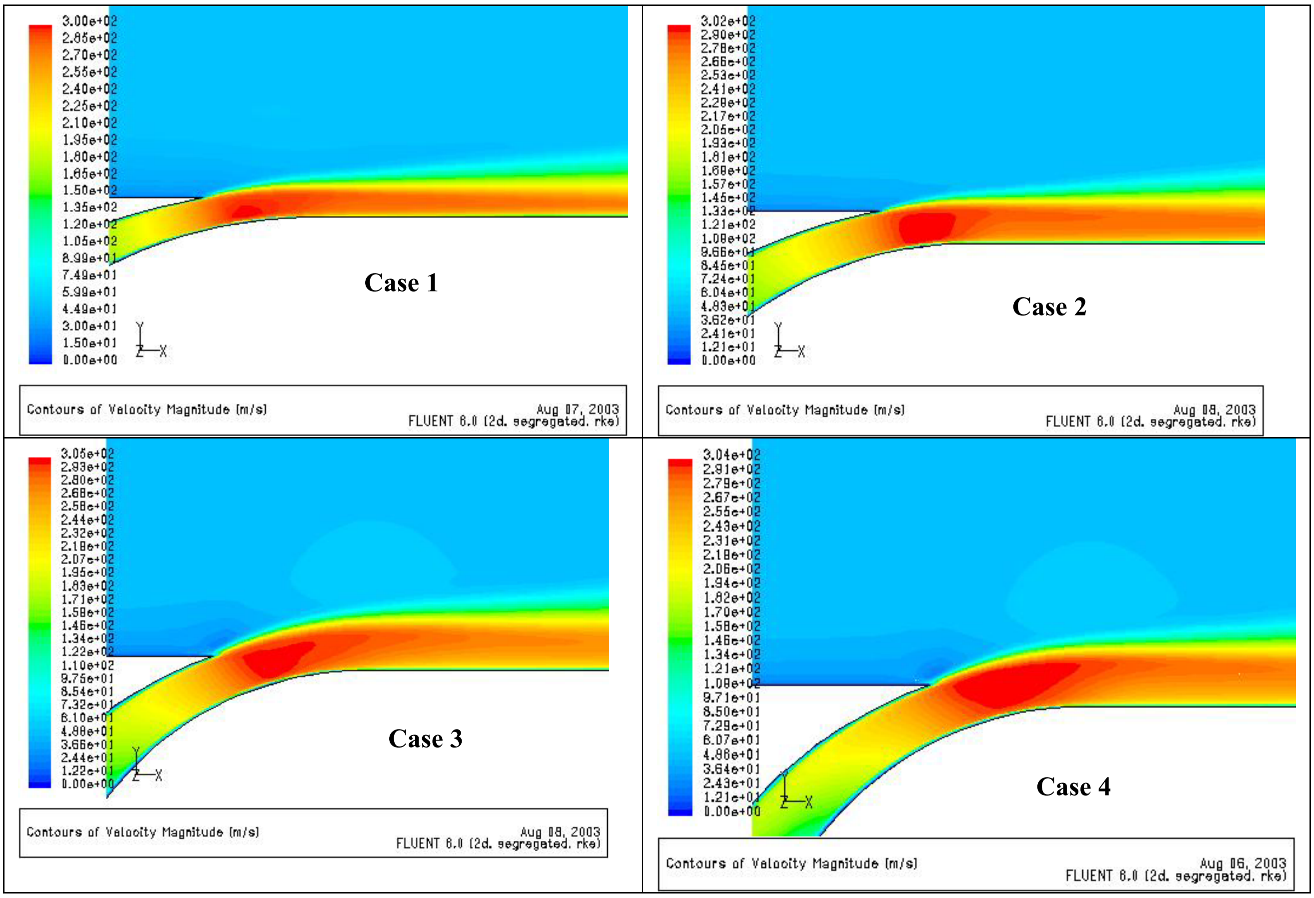

Figure 14 – Velocity contours at the blower exit for all cases studied

4 CONCLUSION

The present work is an initial effort in the direction of numerically simulating the flow along the test section of ITA’s Research Low Speed Wind Tunnel. The results obtained so far seem to indicate that a 20-degree jet angle is able to enhance the boundary-layer momentum more efficiently than the 10-degree option. The main interest is to define the boundary-layer control device geometry as well as adequate their location in respect to the airfoil model. Thus, an extension to 3-D is very desirable and is the natural step to be taken. Further, other parameters, besides the jet angle and slot width should also be considered in the near future; in particular, the influence of the ratio of the blower mass flow to the tunnels mass flow seems to be of paramount importance.

5 AKNOWLEDGMENTS

The present work is inserted in the Project of Innovation Technological PITE, financed by FAPESP and EMBRAER, under grant 00/13769-0.

6 REFERENCES

[1] Girardi, R.M., Fico Jr., N G C R, Mello, O.A.F., and Komatsu, P., "Research Wind Tunnel of the Aeronautical Institute of Technology: Manufacturing Aspects and Design Details", 17th International Congress of Mechanical Engineering (COBEM 2003), Sao Paulo, SP, Brazil, November 10-14, 2003.

[2] Girardi,R.M.; Fico Jr., N.G.C.R.; Mello, O.A.F. & Komatsu, P. "Design of the Research Subsonic Wind Tunnel of the aeronautical Institute of Technology", Proceedings of the Mechanical Engineering National Congress (CONEM 2002), João Pessoa, PB, Brazil, August 12-16, 2002.

[3] Assato, M., Girardi, R.M.; Fico Jr., N.G.C.R.; "Test Section Pressure Measurements at the Research Wind Tunnel of the Aeronautical Institute of Technology", 17th International Congress of Mechanical Engineering (COBEM 2003), Sao Paulo, SP, Brazil, November 10-14, 2003.

[4] Assato, M., Girardi, R.M.; Fico Jr., N.G.C.R.; Mello, O.A.F. & Komatsu, P. Research "Wind Tunnel of the Aeronautical Institute of Technology: Conceptual Design and Calibration", 42nd AIAA Aerospace and Sciences Meeting and Exhibit, Reno, NV, USA, January 5-8, 2004.

[5] Craven, A. H., "*Boundary Layers with Suction and Injection*", CoA REPORT AERO No.136, The College of Aeronautics Cranfield, September 1960.

[6] FLUENT 6.0 User's Guide, Vols. 1-4. Fluent Inc. Lebanon, NH, USA, 2001.