TURBULENT FLOW INSIDE AIR-CONDITIONING DUCT OF COMPLEX GEOMETRY

Rubens Romani
EMBRAER – São José dos Campos –SP
rubens.romani@embraer.com.br

Nide G. C. R. Fico Jr.
ITA - São José dos Campos –SP - Brazil
nide@aer.ita.br

Abstract. *This work presents the numerical analysis of turbulent flows inside air conditioning ducts of complex geometry, with application in the aeronautical industry. The main goal is to make the flow at the air distribution duct exit, located inside the cockpit of a regional jet, as uniform as possible. Another important aspect of the present numerical simulations is pressure loss minimization. Internal guiding vanes are used as tools to modify the flow conveniently. The flow inside the duct is studied in different situations: normal operation with one or two units of air conditioning, with or without air recirculation, and operation without the units of air conditioning (emergency ventilation). The numerical results obtained, confirmed by experimental data, indicate that good velocity-profile uniformity at the ventilation ducts exit is achieved with the installation of three vanes to guide the flow along the air-conditioning duct.*

Keywords*. Internal flow, turbulence, numerical methods, aeronautical industry*

1. Introduction

The air-conditioning system has an important function in commercial aircraft. It guarantees an inhabitable environment inside the aircraft, as it serves as pressurization source. Further, such a system keeps the level of oxygen for breathing and also guarantees the temperature control in the interior of the airplane.

The air-conditioning system, of the majority of the big commercial aircraft, extracts air from stages of the engine compressors through bleed valves. Such air can be cooled by the air-conditioning units, also called packs. The mixture of cooled air with the pack by-pass air gives a cold and hot air mixture that can be adjusted to satisfy the conditions required of the aircraft inside. The cooling of the air bled from the engines or from the APU is due to the expansion through the pack's turbines. The air supply must keep the aircraft at a pleasant temperature, in any situation, independent of the external environment. Extreme cases are: (i) when the aircraft is grounded under high ambient temperatures, (ii) when it is in grounded in days of intense cold or (iii) in flight where the external temperature is very low.

The air-conditioning system must also guarantee the ventilation of the electronic components installed in the cockpit in order to keep them operating at adequate temperatures. This system has fans that are capable of promoting the recirculation of part of the air of the passenger's cabin and of the cockpit. With the two on fans, the air conditioning distribution in regional airplanes is approximately 50% of recirculation air and 50% of fresh air. The crew has the option to choose whether or not to turn on the recirculation fans. The air used in the aircraft for the air conditioning system must be pure and free of any contamination as dust, gases of exhaustion of the engines etc. For this purpose special filters are used. The distribution system must also guarantee enough ventilation to remove smoke in the case of a smoke event in cockpit. The FAR/JAR 25.831(d) requirement states that smoke removal from the cockpit must be achieved in less than 3 minutes of operation of the smoke evacuation system. The air-conditioning system is very important to guarantee the adequate fresh-air flow for aircraft in all of its operation conditions. The system must guarantee the minimum ventilation of the airplane at a 0.55 lb/min flow rate per occupant in normal conditions of operation and at 0.4 lb/min for occupant in abnormal operation, as requisite FAR/JAR 25.831(a). All the requirements mentioned above show how important the design of air conditioning ducts on cockpit is.

For aircraft flying at great altitudes, it is essential for comfort of all occupants to maintain the inside pressure. That is basically obtained by air-conditioning system operation and by the aircraft's flow discharge control valve. As a result the passenger cabin may be kept at a pressure close to its sea-level value, in fact, at pressure correspondent to 8,000 ft altitude in order to have lower level of pressurization, lighter structures and lower aircraft weight. A uniform air distribution inside the cockpit guarantees the crew the ideal environment with a homogeneous temperature, without the bother of an irregular air distribution and without annoying noises at their work place. Such characteristic is, therefore, of utmost importance. Several operating conditions may arise, such as:

- two air-conditioning packs in operation.
- one air-conditioning pack in operation.
- no air-conditioning packs operating (emergency ventilation).

This last case indicates an emergency condition if an aircraft depressurization occurs. This incident, evidently, indicates a malfunctioning in the pressurization system. The condition of the two packs operating is the most usual one. Another operation possible condition is that of only one pack working properly. In this case, the aircraft has to fly at a lower altitude. One-pack operation is also simulated in this work.

Finally, it should be emphasized that the geometry of the air-conditioning duct in a regional aircraft is, as a rule, irregular and complex due to limitation of internal space, especially in the cockpit. In turn, the flow inside it becomes very complex. Thus, all the available resources to improve the air distribution and, as a consequence, reduce the pressure loss become even more important.

2. Numerical Method

For the 3-D simulation of the flow in cockpit mid ceiling duct of Figure 1 with FLUENT (1995) in the different operating settings, the following inlet boundary conditions have been considered. It was used for the simulation of this work a mesh was generated with 3D triangle elements. The mesh is fine in the places where it is found high velocities gradients like near walls and vanes. The mesh is coarse in places which do not have large velocities gradient.

The typical grid for FLUENT simulation is shown on Fig. 2. At the exit of the duct from ceiling panel of Fig.1, there is a connection to nine rectangular grills that allows the air discharge to the cockpit ambient.

The input for FLUENT, was the inlet duct pressure and inlet duct temperature. The input temperature and pressure were known. The duct inlet temperature (Ti) was measured by thermocouple installed in the cockpit supply duct. The inlet pressure (Pi) was measured by pressure transducer. The inflow (M) is also known for each operating conditions. This work presents the simulations for conditions described on Table 1.

The turbulence intensity considered at duct inlet is 4% for all operating conditions simulated. The duct outlet conditions are also known. This duct discharges into cockpit. The duct outlet temperature is the cockpit temperature and the duct outlet pressure is the cabin pressure.

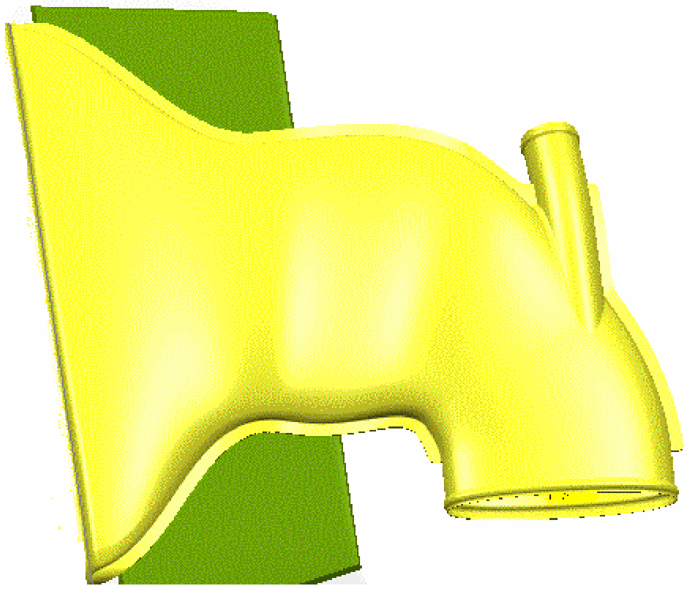


Figure 1: Ceiling panel without guiding vanes - Configuration 1.

For configuration 1 the ceiling central panel has no internal guiding vane, see Fig. 1. For the numerical simulation, using FLUENT, the cockpit was shaped as a big square shaped box with temperature (To) and pressure (Po) known. It must be pointed out that the grid of exit of the duct was not considered, therefore has small effect in the flow.

FLUENT has three different solution modes: segregated, coupled implicit and coupled explicit. All three are capable of supplying accurate results. However, in some cases, a particular formularization may have a better performance, that is, produce the solution faster than other two. The segregated and coupled modes differ in the form that the equation of continuity, momentum, energy and conservation of species equations are solved. The segregated solution of these equations is done sequentially, one equation after the other. On the other hand, the coupled solution mode solves all equations simultaneously. Both methods solve any additional equation, for example, the turbulence modeling ones, sequentially. In the case of the simulation of this work the segregated method was used. The turbulence model used in these simulations with FLUENT was the standard k-ε model. This particular k-ε version used is described by in Peric et al (2001) and Warsi (1992) and it is the most appropriate for the present work. The general equations used in this simulation to solve to flow inside air conditioning duct and to get the velocity and pressure distribution are continuity equation, momentum equation and k-ε turbulence model equations. These equations are described on FLUENT Manual (1995).

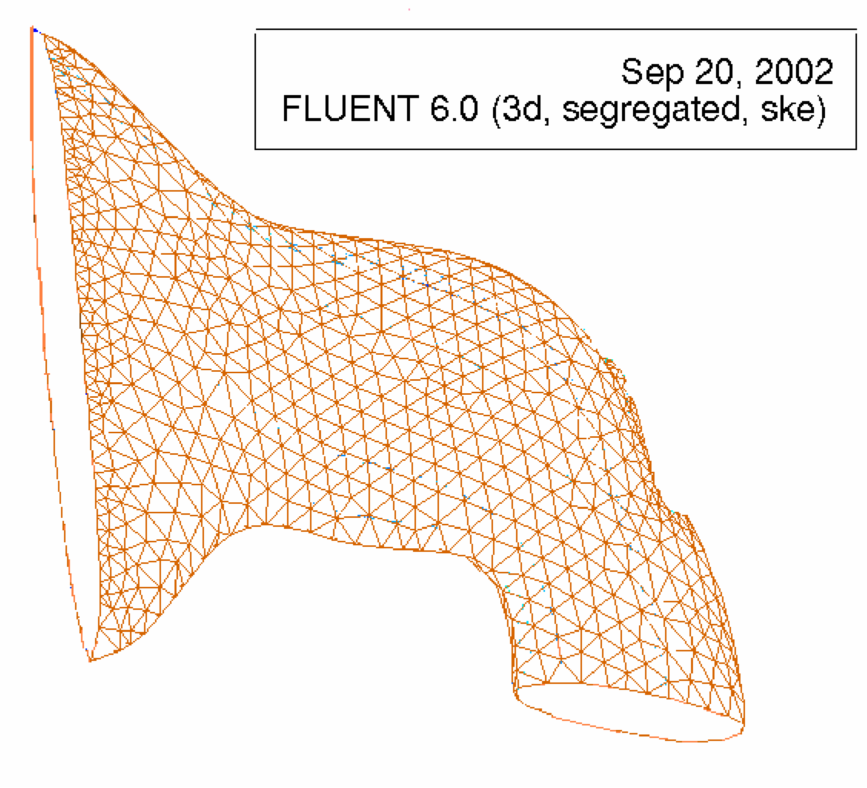


Figure 2: Typical Grid for FLUENT simulation.

3. Results

During the development of the ceiling central panel for a regional aircraft, the airflow in a variety of operating conditions was numerically simulated, as well as several geometries. Hereafter a comparative study is done among three configurations for the ceiling central panel. In configuration 1, with no vanes, it can be seen, through the streamlines plot, that the air distribution is not uniform. Due to its inertia, the flow tends exit the duct through half the available area although before the complex turn it must negotiate the mass flow distribution is fairly uniform across the duct, see Fig. 3. This, of course, should be avoided and the need of guiding vanes is clearly necessary.

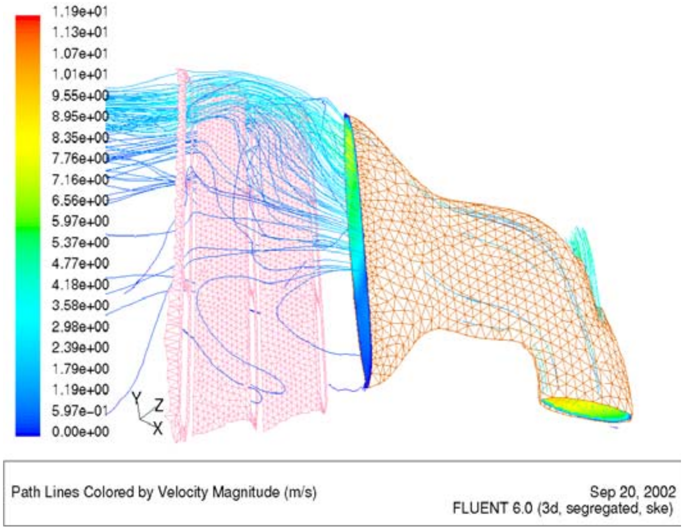


Figure 3: Flow in ceiling panel without guiding vanes - Configuration 1

In configuration 2, shown in Fig 4, the ceiling panel is provided with two turning vanes. These are of fairly simple shape and placed as to subdivide the turn into approximately three equal channels. However, in this particular configuration, the vanes excessively deviate part of the flow towards the internal side of the turn. In effect the flow is now “overturning”. Consequently, the flow does not have uniform distribution at the duct exit plane, as it can be seen in Fig. 5.

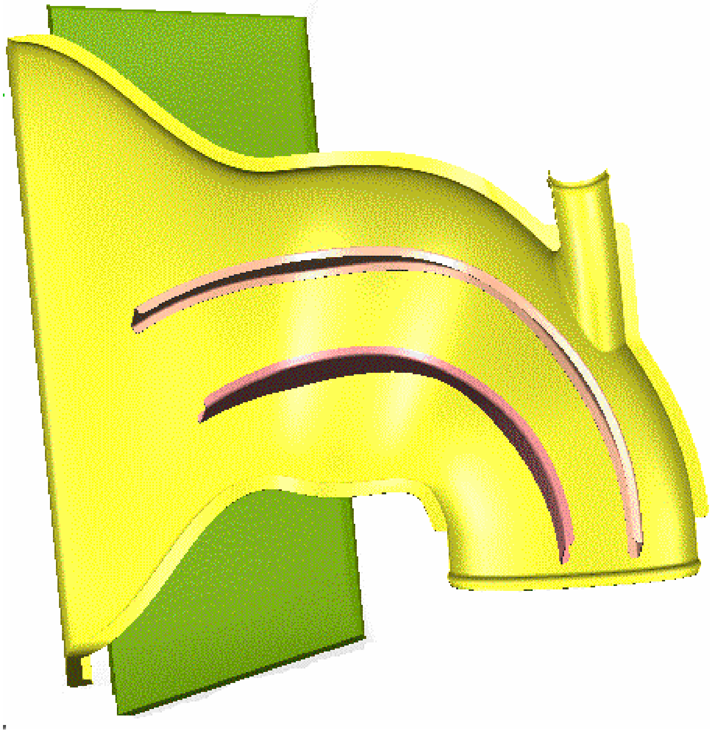


Figure 4: Ceiling panel with two vanes.

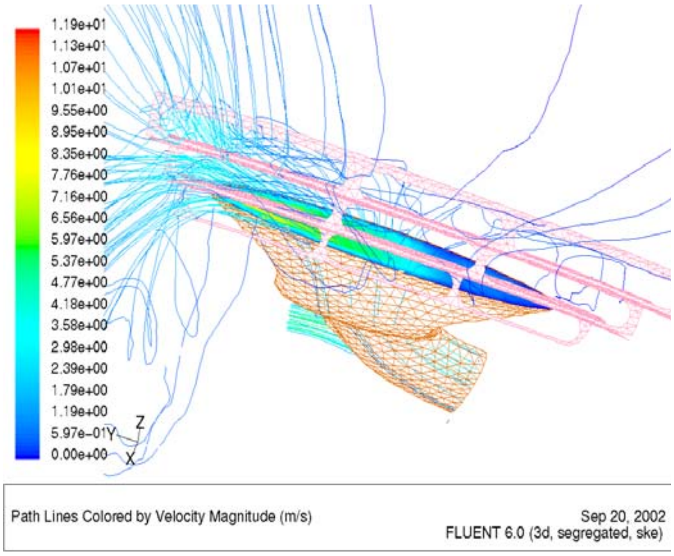


Figure 5: Flow in ceiling panel with two vanes - Configuration 2.

In configuration 3, shown in Figure 6, all three vanes have different lengths. The central one being longer than the others. Each of the three vanes has its own particular shape thus being more suited in guiding the flow at their particular location. It is also important to note that the guiding vanes are not equally spaced. In configuration 3, the turning vanes configuration was optimized based upon streamlines of the flow inside the panel with no vanes. This particular arrangement yields a very regular flow distribution at the panel exit plane, as indicated in Fig 7.

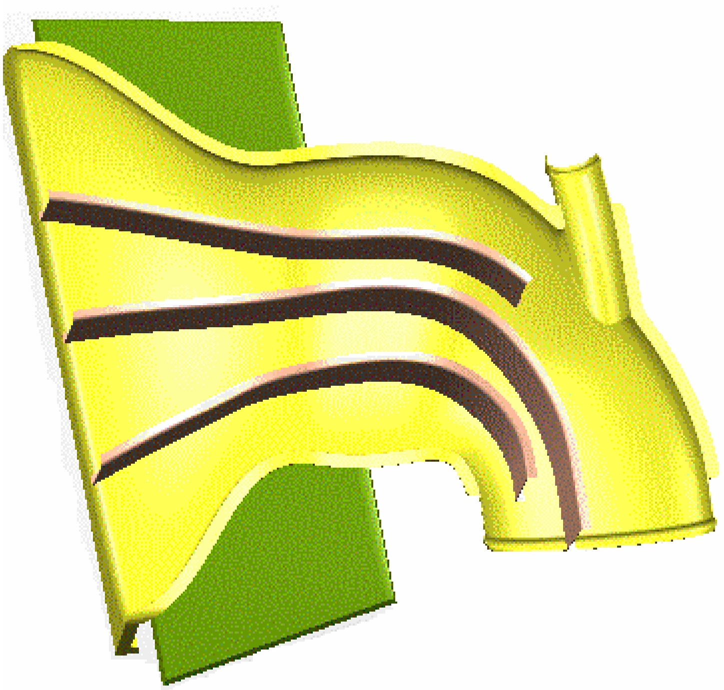


Figure 6: Optimized Ceiling Panel with three turning vanes optimized - Configuration 3.

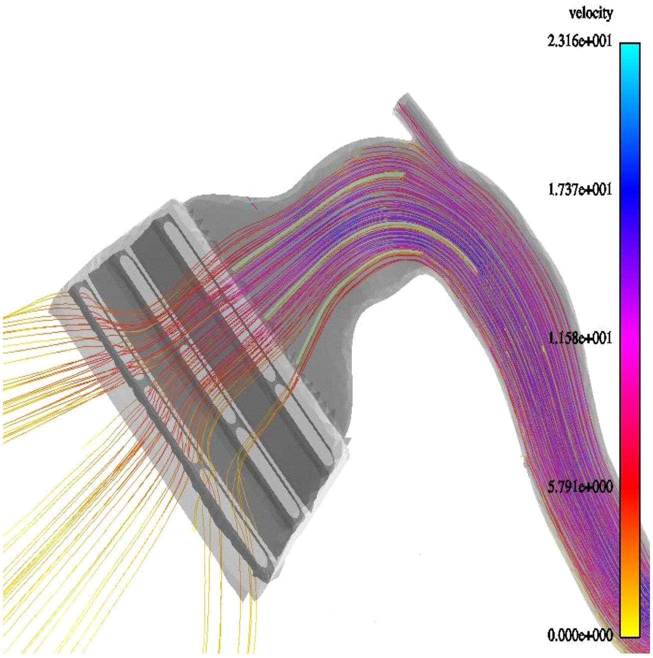


Figure 7: Flow along the optimized ceiling panel with three turning vanes - Configuration 3.

Having arrived at an acceptable flow pattern at the panel exit the next results are for Configuration 3 at different operating conditions. Table 1 show the four cases investigated.

Table 1: Operating condition for the air-conditioning system and respective flow rates inside the typical ceiling panel of regional jet.

Case	Operating Condition	Reynolds
1	One pack on and no recirculation fan.	50,000
2	One pack and one recirculation fan on.	65,000
3	Emergency ventilation (packs and recirculation fans not operative).	75,000
4	Two packs and recirculation fans on.	95,000

The conditions above were taken from flight at 10,000 ft. It is useful to emphasize that the pack flow decreases with flight altitude.

If, for any reason, the two air-conditioning packs fail the emergency ventilation system enters in operation. This system allows external air to be sucked into the aircraft through scoops. In this condition, the valves of the emergency ventilation system, ram-air valves, open. It must be kept in mind that the speed of the aircraft affects the emergency ventilation directly. The outflow of the air-conditioning system, during emergency operation, must be such that the minimum ventilation values are guaranteed inside the aircraft. On the other extreme, it should not as high as to cause discomfort with high speed of air in the cabin. The recommended values for flow velocity inside an aircraft is given in norm ARP 85E.

The air conditioning duct design is obviously a key element to ensure the right environment inside the airplane as well as low energy consumption. A badly conceived project may result in flow recirculation zones along the air conditioning duct resulting in a pressure loss increase. The same problem can happen if the duct geometry is inadequate. Excessive pressure loss indicates inefficiency of the system that can require an increase of the air extraction from the engines and, also an increase of fuel consumption. Another negative aspect is the possibility of internal noise level amplification with direct impact on the comfort of the passengers and crewmembers. The results of the simulations for the cases listed on Table 1 are presented in Figs. 8 to 11.

At first view, the introduction of vanes seems to produce a significant increase of pressure drop due to the increase of friction in the new surfaces now present in the airflow, however the vanes are normally very thin and they are positioned according to the streamlines in the flow in duct without vanes. The effect is a negligible pressure drop increase. The introduction of vanes in the airflow can sometimes reduce the pressure drop due to the elimination of the regions of vortex.

In the present work, the pressure loss analysis shows that no significant pressure drop occurs. It is important to emphasize that the increase pressure drop is associated to increase of noise level on duct. Therefore, the introduction of turning vanes in the airflow, when properly designed, produces negligible increase of noise level. The experimental data of pressure drop on ducts and curves show that when ratio radius bend-duct diameter increases the pressure loss in curves decreases. It means that curves with short radius produces high pressure drop. These data are summarized on SAE Handbook (1990) and Perry (1990). The pressure loss in curves may be reduced by dividing the curve with a number of flow dividers called “splitters” These splitters divide the larger curve into a number of smaller curves, which having a larger radius to duct diameter ratio and therefore a smaller pressure drop. The pressure drop of the new curve with splitters is calculated by using a pressure loss coefficient based on the bend radius to duct width ratio of the subdivided curves. The Reynolds number should also be based on the subdivided curve. It should be mentioned that turning vanes might be lighter and show advantage in Reynolds number if a large number of duct division are used.

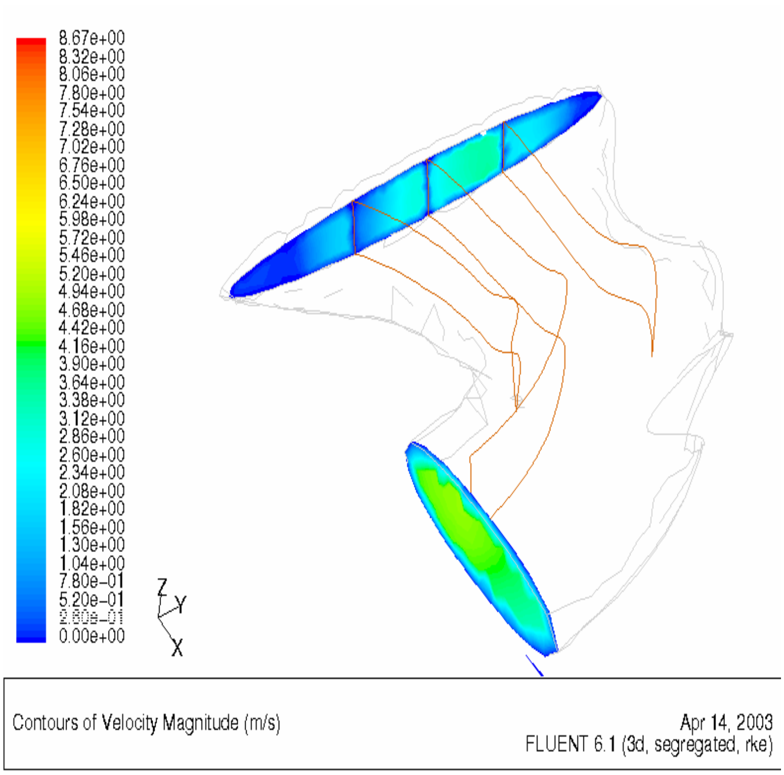


Figure 8: Panel configuration 3 – Re = 50,000.

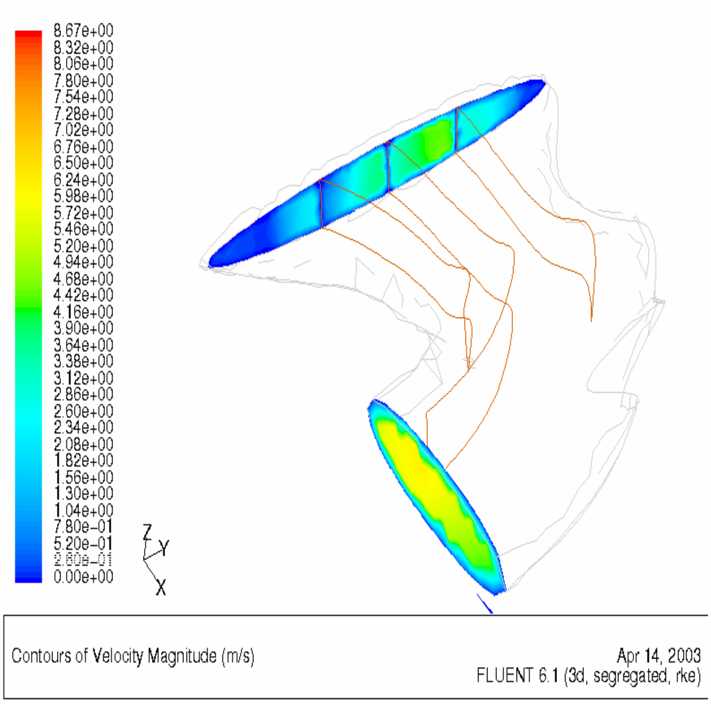


Figure 9: Panel configuration 3 – Re = 65,000.

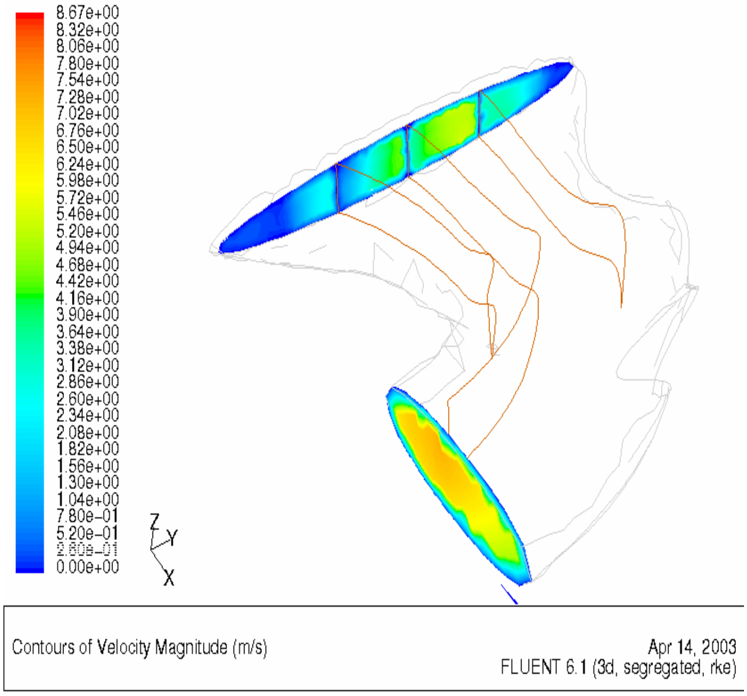


Figure 10: Panel Configuration 3 – Re 75,000.

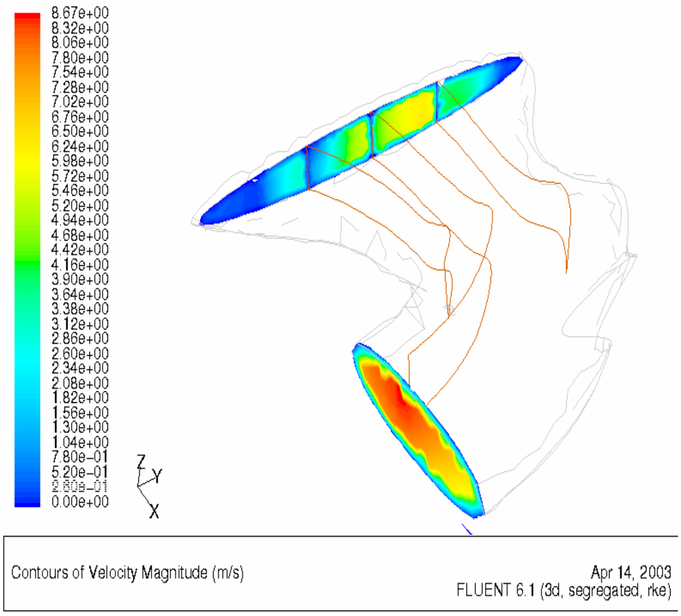
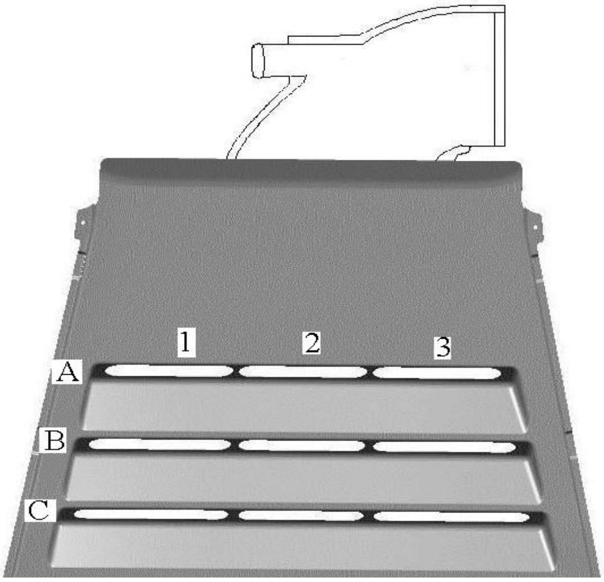


Figure 11: Panel Configuration 3 – Re = 95,000.

Analyzing the air distribution at the exit of configuration 3, for several flow rates, it is verified that for smaller flow rates produce a more uniform exit velocity profile. This should not come as a surprise. For the higher flow rates the velocity magnitude is much larger at the center, compared with the near wall region, at the inlet plane as shown in Figs. 8 through 11. The greater flow inertia also contributes for a slightly superior velocity to the outside of the central vane.

The experimental data obtained using hot wire anemometer when two packs are operating and the recirculation fans on confirm the flow uniformity in the ceiling panel equipped with the three turning vanes – Configuration 3, see Fig. 12. The measurements were done several times at the center of each outlet grill and we considered the average value for each grill. The Experimental data also show that the airflow is also uniform for other operating conditions described on Table 1. That means that the turning vanes form configuration 3 produces good results for a large range of airflow. The experimental data shows that it is not uniform the airflow distribution for panel without vane – Configuration 1.



	Flow Velocity (ft/min) - average		
Location	1	2	3
A	540	500	500
B	700	550	500
C	750	750	500

Figure 12: Flow velocity measurements at nine points at panel's exit with two packs and on fans of recirculation

4. Conclusion

The results presented here are consequence of an initial effort to numerically simulate the flow inside air conditioning ducts with complex geometry. Several realistic situations were reproduced raging from normal operation to emergency conditions. The space limitation inside the cockpit impacts negatively the air conditioning duct system. The turns are not as smooth as they should be and, consequently, the air discharged into the pilot's cabin does not have, in general, the desired distribution. On the other hand, the cockpit's environment has to comply with the ARP 85 recommendations. Such specifications are not met if the flow inside the air conditioning ducts is not properly treated. Guiding vanes must be efficiently placed to guide the flow around the tight curves it must negotiate. The present simulations using FLUENT have indicated that the optimum guiding vanes arrangement is not symmetric. As shown by the results obtained for configuration 3 all vanes are not quite equal to each other, their peculiarities being closely related to their position inside the duct. Further spacing them equally across the duct does not give the best flow distribution at the duct exit.

5. Acknowledgement

The authors would like to thank Ramon Papa from Embraer for his kind contribution in helping set up the numerical simulations with FLUENT.

6. References

FLUENT - Manual do FLUENT 6.1 – 1995, Computational Fluid Dynamics Software and Services.
Ferziger, J.H, Peric, M, 2001, Computational Methods for Fluid Dynamics, Springer.
FAR/JAR Part 25, Federal Aviation Regulation and Joint Airwhortiness Requirements.
Perry John H., Chemical Engineers Handbook, New York- McGraw Hill Company, Inc., 1990, 370-373.
SAE ARP 85, 2002, Air Conditioning Systems for Subsonic Airplanes, rev.E.
SAE – Aerospace Applied Thermodynamics Manual, SAE Committee AC9, Aircraft Environmental Systems New York, pages 1-27 – 1990.
Warsi, Z.U.A., 1992, Fluid Dynamics – Theoretical and Computational Approaches, CRC Press.