NUMERICAL SIMULATION OF AN AIRCRAFT CARGO COMPARTMENT FIRE SUPPRESSION SYSTEM USING CFD TOOL

João Batista do Porto Neves Júnior, portojr@ita.br
Cláudia Regina de Andrade, claudia@ita.br
Edson Luiz Zaparoli, zaparoli@ita.br
Instituto Tecnológico de Aeronáutica. Praça Marechal do Ar Eduardo Gomes, 50; Vila das Acácias - CEP 12228-900 São José dos Campos – SP.

Abstract. *This work focuses on a numerical simulation of a fire suppression system in an aircraft cargo compartment, that uses the halon 131 ($CBrF_3$) as the fire extinguishing agent. The Computational Fluid Dynamics (CFD) tool has been used to analyze a time-space evolution of the halon volumetric concentration that is the main requirememt in the certification process. This mathematical model also allows to verify the influence of different parameters as infiltration rate and mixture leakage rate on the fire suppression system design. A mesh has been generated involving the cargo compartment domain to solve the governing differential equations. Numerical and experimental results showed a good agreement for the halon volumetric concentration inside the cargo compartment and satisfying the certification requirements. It is concluded that the model developed in the present work is an important tool to be used in the preliminary analysis during the aircraft conceptual design stage or to modify a pre-existent fire extinguishing system.*

Keywords. *Suppression System, Numerical Simulation, Computational Fluid Dynamics, Halon*

1. INTRODUCTION

Historically, the use of the fire is linked directly to the humanity's development, because many are the residential and industrial applications indispensable to the modern man's life. However, in the course of time the man discovered that as important as to use the fire was to avoid that the same happened without being wanted due to its great destruction power and that is a concern in all of the branches of human activity. In the aviation, this is a concern still larger, because many aerial accidents attributed to fires inside aircrafts, as we can verify in DOT/FAA/AR-TN98/32, as listed in Table 1.

With the intention to increase the efficiency of the equipments and systems used by the aircrafts, organizations regulators as Federal Aviation Administration (FAA, 2007), in the United States and Joint Aviation Authority (JAA), in Europe dictate rules and procedures for necessary certification to the commercialization of the aircrafts. Among these certifications, the systems of fire suppression in compartment of load of the aircrafts will be the focus of this work.

The aircraft fire suppression system is composed of detection and a fire extinguishing equipment. The detection .system acts in the presence of smoke with fire lights in the cockpit that alarm the pilot in the case of fire occurrence.

Actually, the great majority of the aircrafts uses the halon 1301 ($CBrF_3$) as agent extinguisher, however this gas when discharged in the atmosphere reacts with the layer of ozone destroying it. Because this fact, the halon production has been forbidden in the Treaty of Kyoto and Montreal, being just been able to use the existent reservations.

Several works in the literature have focused on the smoke transport simulation employing the CFD tool for different environments (e. g., gymnasium, auditorium, tunnels and aircraft cargo compartment).

Chen et al. (2005) investigated and assess fire smoke transport and management in a realistic indoor sports center using a CFD code. In the event of a fire, the smoke management system must guarantee that the bottom of the smoke layer be kept high enough to prevent the life threatening impact of smoke and heat on the people for a long enough period of time for them to be evacuated. The authors verified that results show that door effect must be carefully considered in the fire safety design and the mechanical ventilation is not always an effective improvement to natural ventilation.

The CFD tool has been used also to study the CO_2 turbulent dispersion 3-D phenomenon in an auditorium (Papakonstatinou, 2002). Two different ventilation systems are considered: an air induction and an abduction flow rate. Besides, the people numbers as well as the lighting equipment were taken into account. Numerical results for the fluid flow pattern inside the auditorium presented a reasonable agreement with literature data.

Bari and Naser (2004) performed a numerical study of smoke emitted by vehicles caused by traffic jam in a road tunnel employing the CFD technique. Results showed that the vehicles downstream of the fire had enough time to escape from the tunnel through the exit portal. Due to the action of jet fans, most of the smoke was pushed downstream of the fire, putting the passengers in this region at great risk.

Papa et al. (2005) presented a CFD model for fire and smoke generation inside an aircraft cargo compartment. The model can provide information on smoke transport under various conditions. The flexibility of the model allows the simulation of numerous fire scenarios in a short period of time. The sensors positioning effect on the compartment shape were also evaluated.

Penteado (2004) performed a numerical simulation of the halon extinguishing system in a regional aircraft cargo compartment using the lumped parameter method. The time-evolution halon concentration, after the rapid discharge bottle (higher volume) and slow one (lower volume) was obtained. These numerical results presented a satisfactory agreement when compared with experimental data (tests performed to satisfy certification requirements).

Table 1. Aerial accidents that happened due to fires

Date	Location	Operator	Airplane Model	Fatalities
8/19/80	Riyadh, Saudi Arabi	Saudia	L-1011	301
9/23/83	Abu Dhabi, UAR	Gulf Air	B737	112
5/11/96	Florida Everglades	ValuJet	DC-9	110

Usually, the aircraft suppression system is alike the scheme presented in Teixeira (2004) shown in Figure 1, where the pilot after testing the efficiency of the detection system works the fire extinguishing system shooting the bottle 1. This fast discharge bottle has the purpose of elevating the volumetric concentration of halon instantly above 5% as requested by the certification criteria. The second bottle has the purpose of compensating the mass flow rate of infiltration of air and mass flow rate of mixture leak for seeking to maintain the minimum concentration of 3% that is minimum concentration that inhibits the reverse-ignition of the cause of the fire.

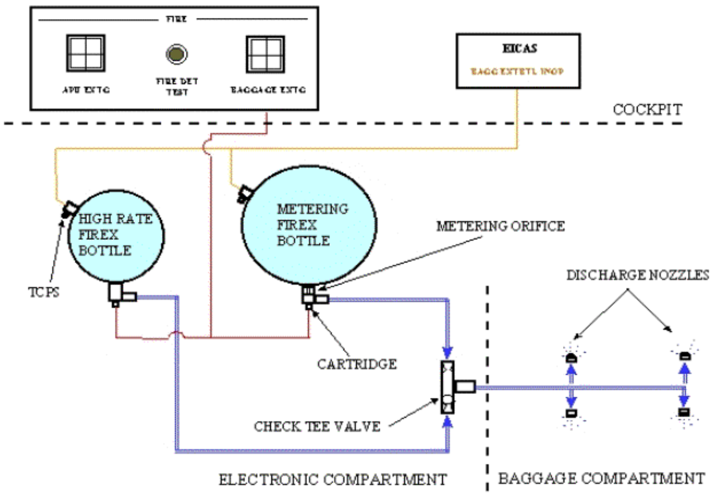

Figure 1**.** Fire suppression system schematic representation.

The certification test described above is an expensive process that happens in the final phase of the aircraft design. On the other hand, some aircrafts clients demand modifications in the original project of the cargo compartment that should be appraised to avoid a great impact in the suppression system.

Kurokawa et al (2004) developed a mathematical model to simulate the halon volumetric concentration time-evolution inside an aircraft cargo compartment. The lumped parameter approach was applied and the differential equation system was solved using a fourth-order Runge-Kutta scheme. The adopted methodology allowed determining the transient behavior of the following parameters: halon volumetric concentration, halon and air masses and the cargo compartment pressure. Although the formulation did not show halon concentration spatial distribution non-uniformities, it provided useful and rapid information to design the aircraft cargo compartment fire extinguishing system.

The present work focuses on the use of CFD tool to simulate the air and halon transient flows in an aircraft cargo compartment (Figure 2) considering the 3-D spatial variation and time evolution of the fire extinguishing agent concentration. This procedure can be an efficient tool to be used in the design preliminary phases or still in the evaluation of the possible modifications in the original project, as already mentioned.

2. MATHEMATICAL FORMULATION

The air and halon transient flows inside the aircraft cargo compartment were modeled using the Fluent CFD commercial package. Both fluid are considered to be constant fluid properties and are incompressible. The Reynolds-Averaged Navier-Stokes equations (continuity, momentum, species transport) in conjunction with the standard k-ε turbulence model (Fluent Inc. 1996) are stated as:

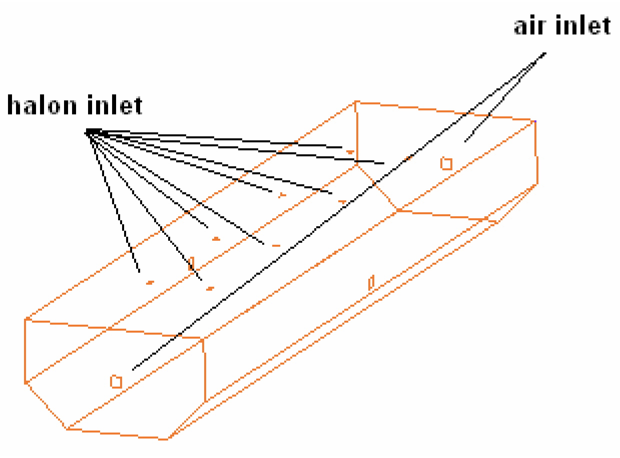

Figure 2. Schematic diagram of the aircraft cargo compartment geometry, showing the air and halon inlets.

Continuity Equation

$$\frac{\partial \rho}{\partial t}+\frac{\partial}{\partial x_i}(\rho u_i)=0 \tag{1}$$

Momentum equations

$$\frac{\partial(\rho u_i)}{\partial t}+\frac{\partial}{\partial x_j}(\rho u_i u_j)=-\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}-\frac{2}{3}\delta_{ij}\frac{\partial u_l}{\partial x_l}\right)\right]+\frac{\partial}{\partial x_j}\left(-\rho\overline{u'_i u'_j}\right) \tag{2}$$

Species Conservation equation

$$\frac{\partial \rho_A}{\partial t}+\frac{\partial(\rho_A u)}{\partial x}+\frac{\partial(\rho_A v)}{\partial y}+\frac{\partial(\rho_A w)}{\partial z}=\frac{\partial\left(D_{AB}\frac{\partial \rho_A}{\partial x}\right)}{\partial x}+\frac{\partial\left(D_{AB}\frac{\partial \rho_A}{\partial y}\right)}{\partial y}+\frac{\partial\left(D_{AB}\frac{\partial \rho_A}{\partial z}\right)}{\partial z} \tag{3}$$

k-ε turbulence model equations

$$\frac{Dk}{Dt}=\frac{\tau_{ij}}{\rho}\frac{\partial u_i}{\partial x_j}+\frac{\partial}{\partial x_j}\left[\left(\nu+\frac{\nu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]-\varepsilon \tag{4}$$

$$\frac{D\varepsilon}{Dt}=C\varepsilon_1\frac{\varepsilon}{k}\frac{\tau_{ij}}{\rho}\frac{\partial u_i}{\partial x_j}+\frac{\partial}{\partial x_j}\left[\left(\nu+\frac{\nu_t}{\sigma_k}\right)\frac{\partial \varepsilon}{\partial x_j}\right]-C\varepsilon_2 f_2\frac{\varepsilon^2}{k} \tag{5}$$

where:

u	velocity component of the x direction
v	velocity component of the y direction
w	velocity component of the z direction
p	pressure
ν	kinematic viscosity
ν_t	turbulent kinematic viscosity
D_{AB}	diffusion coefficient of the halon in the air
ρ	Specific mass
ρ_A	halon specific mass
t	time
k	turbulent kinetic energy
ε	turbulent dissipation rate
τ_{ij}	stress tensor component

The diffusion coefficient for a binary system is a function of temperature, pressure, and its composition (Bird, 1960). At low pressure, the D_{AB} coefficient is in inversely proportional to the pressure, elevates with increasing temperature, and is almost independent of composition for a given gas-pair. For the non polar pair halon-air (Sissom and Pitts, 1972), the D_{AB} coefficient is calculated as:

$$D_{AB} = 0.0069 \frac{T^{\frac{3}{2}}}{p\left(V_A^{\frac{1}{3}} + V_B^{\frac{1}{3}}\right)^2} \sqrt{\frac{1}{M_A} + \frac{1}{M_B}} \quad (6)$$

Where: D_{AB} – mass difusivity [ft^2/h]; T – mixture temperature [°R]; p – mixture pressure [atm], M molecular weight; V molecular volume [ft^3/lb.mol].

Table 2 shows the boundary conditions implemented at the air and halon inlets (Figure **2**) and also the mass flow rates values utilized.

Table 2**.** Boundary conditions.

Name	Type	Mass flow rate (kg/s)	Turbulence Intensity (%)	Turbulent viscosity ratio	Halon Mass Fractions
Air inlet 1	Mass Flow Inlet	0.02	5	30	0
Air inlet 2		0.02	5	30	0
Halon 1	Mass Flow Inlet	0.082	5	30	1
Halon 2		0.082	5	30	1
Halon 3		0.082	5	30	1
Halon 4		0.082	5	30	1
Halon 5		0.082	5	30	1
Halon 6		0.082	5	30	1
Halon 7		0.082	5	30	1
Halon 8		0.082	5	30	1
		Flow Rate weighting			
Mixture outlet 1	Outflow	1			
Mixture outlet 2		1			

3. SOLUTION STRATEGY

Cargo compartment geometry was created with the same volume used in the certification test (Penteado, 2004) but some geometric details were modified due to restrictions of confidential information. This procedure also simplified the mesh generation process.

The solution strategy was composed by three steps: (i) pre-processor, (ii) solver and (iii) post-processor.

(i) The *pre-processing* consists of: computational domain construction, applying the boundary conditions, air and halon properties input, mesh generation using ICEM CFD package. A mesh with hexahedrycal elements was generated containing 827,740 cells, as illustrated in Fig. 3. All infiltration was condensed in two areas of the 0.01 m^2 located in the front and behind of the cargo compartment. On the other hand, all leakage also was condensed in two other areas of the 0.01 m^2 located in each side of the cargo compartment;

(ii) In the *solver* procedure, the volume finite technique was employed to solve the governing equations (Eq. 1 to Eq. 5) with a segregated formulation and second-order discretization. The pressure and velocity fields were coupled employing the SIMPLE algorithm.

(iii) In the last step, the solution visualization is obtained using the FLUENT code graphic interface. Velocity components, pressure, species concentration, and turbulent quantities (contours and plots) are available at user pre-defined planes (Fig. 4).

The time-dependence solution was obtained with time-step of 0.1s, during the first bottle discharge (to capture the quicker phenomenon evolution) and 0.2 to 0.5s to the second bottle discharge.

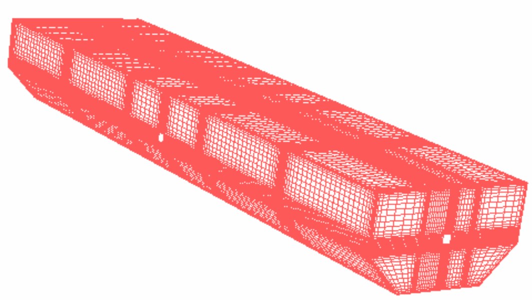

Figure 3. Illustration of the hexa mesh created in the cargo compartment

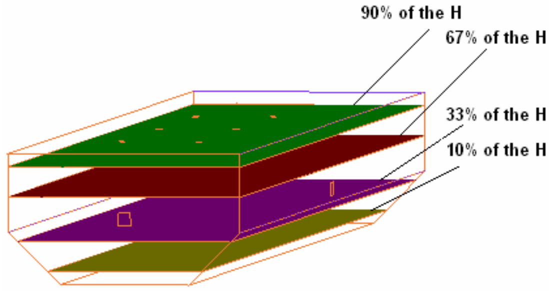

Figure 4. Planes at 10%, 33%, 67% and 90% of the cargo compartment height (H).

4. RESULTS

The temporal and spatial variations of halon volumetric concentration inside cargo compartment will be compared with test certification results. The certification test consists of a real and even flight in an altitude of 41,000 ft with the cargo compartment totally empty. After 30 minutes of even flight in the mentioned altitude, the fire suppression system is started promoting a halon discharge during 6 seconds corresponding to the high pressure bottle volume. Afterwards, the second bottle is shot by a period of 60 minutes necessary time to the pilot performs a safe landing. In the sequence the aircraft stays in the cruise flight for more 25 minutes when the certification test is finished.

To perform the comparison with these experimental results, it was created a similar numerical setup to measure the volumetric concentration of halon. 12 sensors were installed called probes that collect continually samples of the mixture halon-air which are sent for analyzing equipment of concentration and to register the variation of the same with the time. These sensors are located at 10%, 33%, 67% and 90% (Fig. 4 and Fig 5) of the height of the cargo compartment in three stems also positioned in the cargo compartment. Table 3 shows the probes coordinates and the Figure 5 illustrates the position of the probes used in the real certification test.

Table 3**.** Information about the probes position in cargo compartment.

Probe Numeration	Bar	Height	Coordinate (x,y,z)
Probe 1	1	10% de H	(0.65; 0.10 ; 1.825)
Probe 2		33% de H	(0.65; 0.33 ; 1.825)
Probe 3		67% de H	(0.65; 0.67 ; 1.825)
Probe 4		90% de H	(0.65; 0.90 ; 1.825)
Probe 5	2	10% de H	(0.90; 0.10 ; 3.650)
Probe 6		33% de H	(0.90; 0.33 ; 3.650)
Probe 7		67% de H	(0.90; 0.67 ; 3.650)
Probe 8		90% de H	(0.90; 0.90 ; 3.650)
Probe 9	3	10% de H	(1.15; 0.10 ; 5.475)
Probe 10		33% de H	(1.15; 0.33 ; 5.475)
Probe 11		67% de H	(1.15; 0.67 ; 5.475)
Probe 12		90% de H	(1.15; 0.90 ; 5.475)

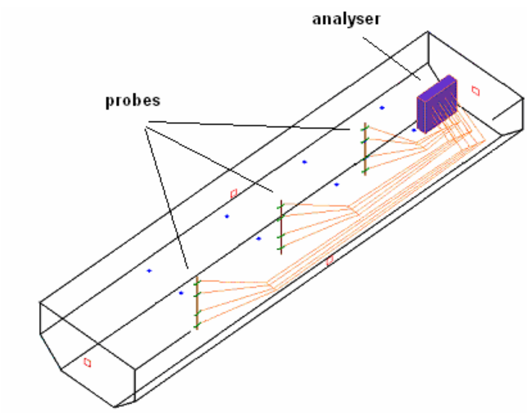

Figure 5. Illustration of the cargo compartment and the localization of the stems and probes.

In the simulated case, four planes were created located the height of 10%, 33%, 67% and 90% of the cargo compartment coincident with the probes localization, as shown in Fig. 5.

It is important to remember that the halon load depends on the air infiltration rate (determined by intentional and non-intentional aircraft cabin leakages) which isn't known a priori during the experimental setup. Due to this fact its value must be estimated as input to the numerical study. In this work, three values for the mass air flow infiltration rate were carried out: 0.02 kg/s, 0.04 kg/s and 0.08 kg/s whose results for halon volumetric concentration are shown in the

Figure 6 to Figure 10 (average CFD values obtained for planes at 10%, 33%, 67% and 90% of H, respectively). It is verified that as the air flow infiltration rate increases, the halon volumetric concentration is reduced due to its dilution inside the cargo compartment volume.

The time evolution of the certification test results are also illustrated for a 90 minutes total time and average halon volumetric concentration obtained from four values at each height (three bars in Fig. 5). Note that with the evaluated air infiltration rate, the numerical halon volumetric concentration curve has a similar behavior but the values are underpredicted. When the air infiltration rate of 0.08 kg/s is used as input, the halon concentration minimum limit of 3% (required for certification) is achieved after total second bottle discharge (see Fig. 6 to Fig. 9).

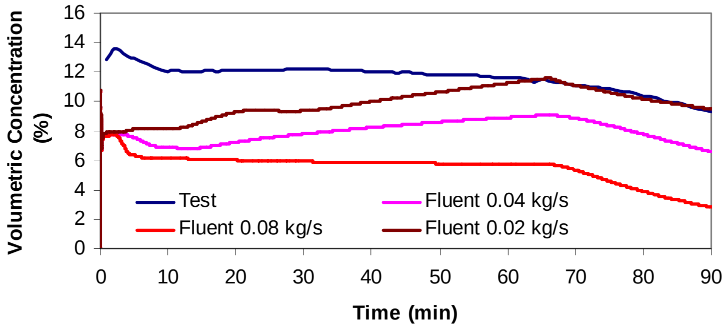

Figure 6. Comparison between test and CFD results for the plane at 10% of height.

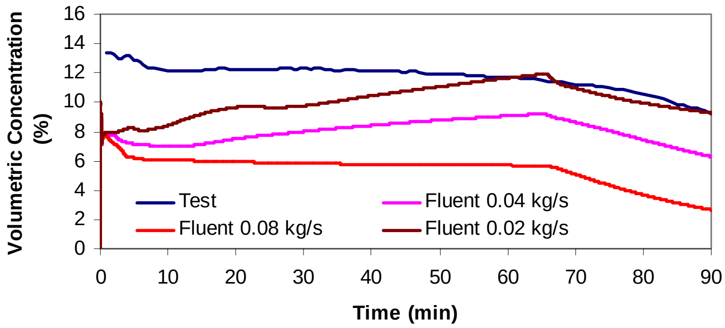

Figure 7**.** Comparison between test and CFD results for the plane at 33% of height.

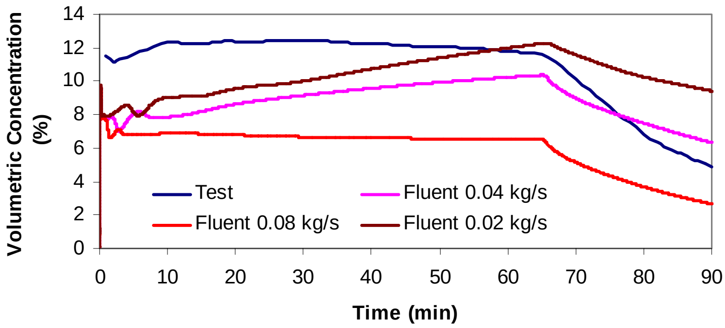

Figure 8**.** Comparison between test and CFD results for the plane at 67% of height.

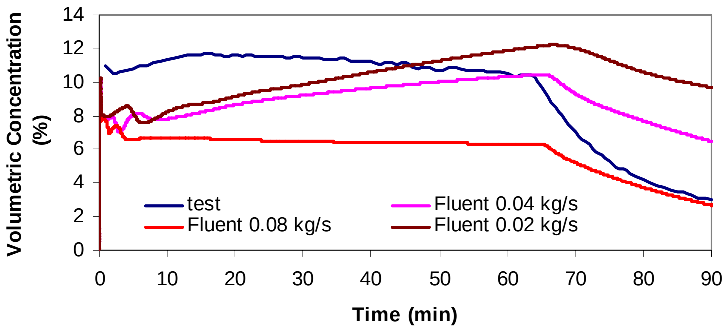

Figure 9**.** Comparison between test and CFD results for the plane at 90% of height.

Figure 10 shows numerical and experimental average values for the halon volumetric concentration. While the CFD results uses a large number of values (average employing all cargo compartment volume), the test results uses only average from 12 values (4 probes and 3 bars as showed in Fig 5). The most important result is that the CFD tool can be used to evaluate the airflow infiltration rate inside the aircraft cargo compartment.

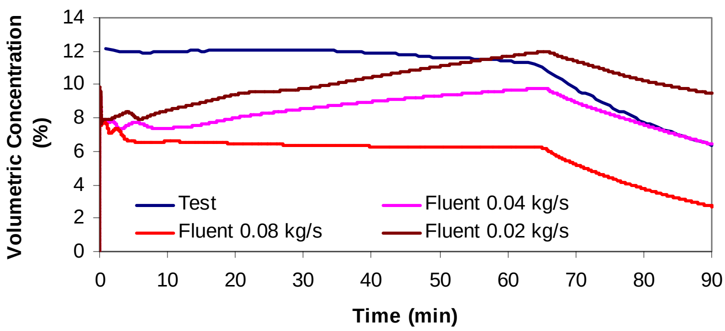

Figure 10**.** Result of the overall average of all probes (test case) and cargo compartment volume (CFD case).

Figure 11 shows the halon volumetric concentration at two planes containing the injection orifices (see halon inlets in Fig. 2) at 10 s after the system activation. Note that the halon jets are perpendicular to the cargo compartment floor, with maximum values (red color) close to the ceiling.

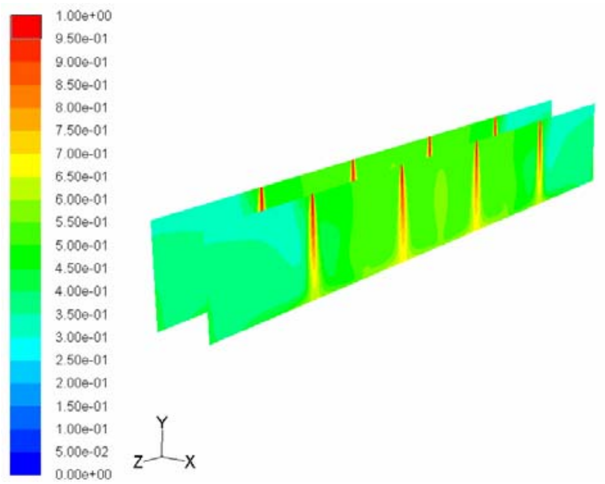

Figure 11. Halon volumetric concentration at two planes containing the halon inlets.

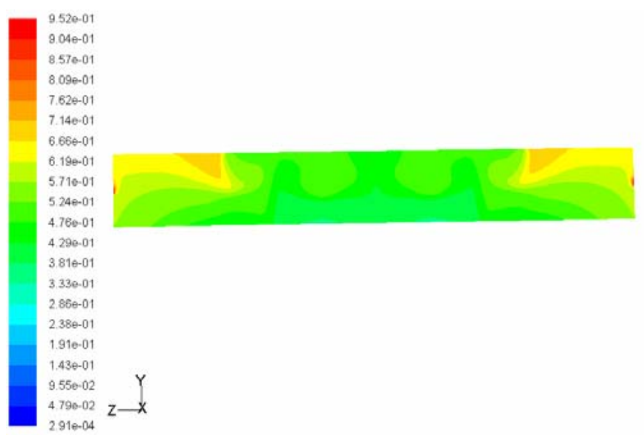

Figure 12. Air volumetric concentration at a plane containing the air infiltration inlets.

To analyze the spatial distribution at the plane zy, the air volumetric concentration is plotted at a plane containing the two opposite air infiltration inlets, Fig 12. Note that the maximum airflow values (red color) occurs close to the walls that containing the orifices. This jet air dilutes the halon concentration inside the cargo compartment.

5. FINAL DISCUSSION

At the present work a CFD tool is employed to analyze an aircraft fire suppression system based on halon discharge. The time-evolution and spatial distribution of halon concentration are obtained and compared with experimental data, showing that the airflow infiltration mass rate affects significantly the halon volumetric concentration. The numerical results can be also used to study the smoke detector positioning inside the aircraft cargo compartment and to modify a pre-existent fire suppression system.

Differences obtained between numerical and test results can be attributed to:

- The fewness of experimental data (only 12 values) while the CFD results are denser;
- In the numerical case, the halon inlets are totally perpendicular to the cargo compartment ceiling while the real test possesses a diffuse nozzle, which allows a better spatial spread of halon jet.

6. REFERENCES

Bari, S. and Naser, J., “Simulation of Smoke from a Burning Vehicle and Pollution Levels Caused by Traffic Jam in a Tunnel”, 2004, Tunnelling and Underground Technology.

Bird, R. B., Stewart, W. E. and Lightfoot, E. N., 1960, “Transport Phenomena”,Ed. Wiley International, Tokyo, Japan, 780 p.

Chen, J., Chen, H., C. and Fu, S., 2005, “Numerical Investigation of Fire Smoke Transport in the Tsinghua University Sports Center”, Tsinghua Science and Tecnology, Vol 10, Number 5, pp. 618-622.

FAA, 2007, “Federal Aviation Administration”. 2007, <http://www.faa.gov.>

Fluent User’s Guide Volume 2, 2006.

Kurokawa, F. Y.; Andrade, C. R. and Zaparoli, E. L., 2004, “Numerical Simulation of the Concentration of Fire Extinguishing Agent in Aircraft Cargo Compartment.”, In: Brazilian Congress of Thermal Sciences and Engineering – ENCIT, Rio de Janeiro, RJ. Brazil.

Papa, R., Pustelnik, M. and Santos, L. C. C., 2005, “A CFD Model for Cargo Compartment Smoke Detection” Proceedings of IMECE 2005 ASME International Mechanical Engineering Congress & Exposition November 5-11, Orlando, FL.

Papakonstatinous, K. A., Kiranoudis, C. T. and Markatos, N. C., 2002, “Numerical Simulation of CO_2 Dispersion in an Auditorium”, Energy and Buildings”, Vol 34, pp. 245-250.

Penteado, T. A., 2004, “Análise da Concentração de Agente Extintor em Compartimentos de Bagagem”. Tese (Mestrado Profissional em Engenharia Aeronáutica e Mecânica) – Instituto Tecnológico de Aeronáutica, São José dos Campos, São Paulo, 111 p.

Sidney, B. T., 2004, “Análise do Fluxo de Agente Extintor em Modelo de Sistema Aeronáutico de Supressão de Fogo.”, 78 f. Tese (Mestrado Profissional em Engenharia Aeronáutica e Mecânica) – Instituto Tecnológico de Aeronáutica, São José dos Campos, São Paulo, 2004.

Sissom, L. E. and Pitts, D. R., 1988, “Fenômenos de Transporte”, Ed. Guanabara S.A.,Rio de Janeiro, Brazil, 765 p.

7. RESPONSIBILITY NOTICE