NUMERICAL SIMULATION OF THE SMOKE TRANSPORT INSIDE AN AIRCRAFT CARGO COMPARTMENT USING A PASSIVE SCALAR APPROACH

Ramon Papa
Luis C. de C. Santos
EMBRAER – Empresa Brasileira de Aeronáutica - Environmental Control Systems CFD Team
Cláudia Regina de Andrade
Edson Luiz Zaparoli
Instituto Tecnológico de Aeronáutica – ITA, Departamento de Energia
Praça Mar. Eduardo Gomes, 50 – Vila das Acácias
São José dos Campos – SP CEP 12 228 900

Abstract. *The smoke transport due to a fire should be well understood for designing practical smoke management system. Current regulations require that aircraft cargo compartment smoke detectors alarm within one minute of the start of a fire and at a time before the fire has substantially decreased the structural integrity of the airplane. Presently, in-flight tests, which can be costly and time consuming, are required to demonstrate compliance with the regulations. This paper presents a CFD model for smoke dissipation inside the 707 cargo compartment in fire condition. The model can provide information on smoke transport under various conditions. It is also fast running, allowing for simulation of numerous fire scenarios in a short period of time. The conservation equations of heat, mass and momentum transfer were solved using the FLUENT software with a passive scalar approach to model the smoke (CO,CO_2 and soot) transport. In order to validate the whole procedure, the results of the simulations with FLUENT are compared with experimental data and the results of a CFD code developed by Sandia National Laboratories. Temperature profiles along the ceiling cargo compartment and smoke concentration are also obtained. It is shown that the proposed method should be useful for designing a smoke management system and the CFD smoke transport model presented has the potential to satisfy the certification process.*

Keywords*: fire, smoke, soot, passive scalar, RANS equations.*

1. INTRODUCTION

The smoke transport due to a fire should be well understood for designing practical smoke management system. Smoke management systems are defined as engineered systems that include all methods that can be used singly or in combination to reduce smoke production or to modify smoke movement (Chiang et al., 2003).

Apart from full-scale burning tests, fire field model or application of Computational Fluid dynamics is now a popular method to study smoke transport, Chow and Yin (2004). Nowadays The CFD tool has been used to predict fire and smoke dissipation in a great variety of both interior and exterior environments.

Stribling (2001) performed a validation study applied to fire and smoke ventilation in a square room containing a methane burner. The paper concentrates on the layout in which the burner is positioned centrally in the room and ventilation is provided by a room opening (doorway). The comparison with experimental data for temperature distribution and flow rates showed good agreement.

Bari and Naser (2004) studied the smoke transport from a burning vehicle in a road tunnel using CFD techniques. A bus was burned to release an equivalent amount of energy of 500 liters of diesel. The jet fans were able to push the smoke towards the exit portal, where there was no vehicle or passenger. Results showed that the smoke dispersed about 55 m upstream of the fire and this was the region where vehicles were at a standstill due to the burning vehicle. The authors concluded that this region was critical, as the passengers would be suffocated due to smoke and soaring temperatures.

Chen et al (2005) employed a CFD code (FDS, Fire Dynamics Simulator) to predict fire and smoke transport of large open spaces (gymnasium) with great geometric complexity. The governing equations (mass, momentum, species concentration and energy) were solved employing a control volume technique and the turbulence was modeled using the Large Eddy Simulation (LES) model. Results showed that the door effect must be carefully considered in the fire safety design. Besides, the mechanical ventilation is not always an effective improvement to natural convection.

To avoid having to break-up the air in separate species and consequently the excessive computational effort of representing not only a set of equations for each one of the species, a passive scalar representation of the chemical substances transport can also be used. A passive scalar technique has been used by Sinai et al (2004) to model aerosol in an enclosure room where the fluid was assumed to be air only, using the Boussinesq approximation and turbulence was modeled with the k-ε closure. According those authors, the results are very satisfactory and the approach was considered to be sufficiently reliable for design purposes.

At the present work, a numerical study of the resin burning products (CO, CO_2 and soot) convective heat and mass transfer was accomplished to verify temperature and smoke distribution near aircraft cargo ceiling, where the smoke sensor are positioned. The RANS equations, standard k-ε turbulence model in conjunction with scalar passive approach are employed to model this problem.

2. MATHEMATICAL FORMULATION AND NUMERICAL SOLUTION

The airflow patterns, temperature distribution and smoke transport (resultant from the resin block burning) inside an aircraft cargo compartment are herein numerically simulated. A schematic diagram of the geometry and main boundary family names is presented in Figure 1. The computational domain dimensions are the same of the experimental FAA (Federal Aviation Administration) Technical Center set-up, presented in Blake (2000), in which a forward 707 cargo compartment was instrumented as depicted in Figure 2.

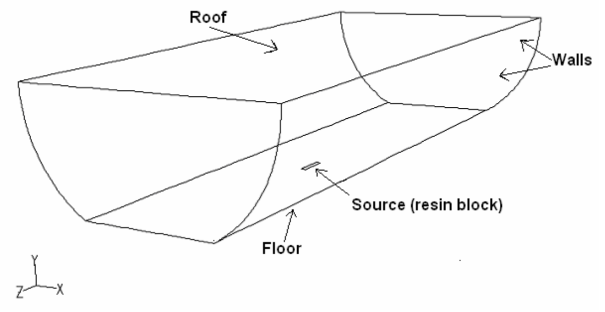


Figure 1 - CFD geometry model and boundary family names

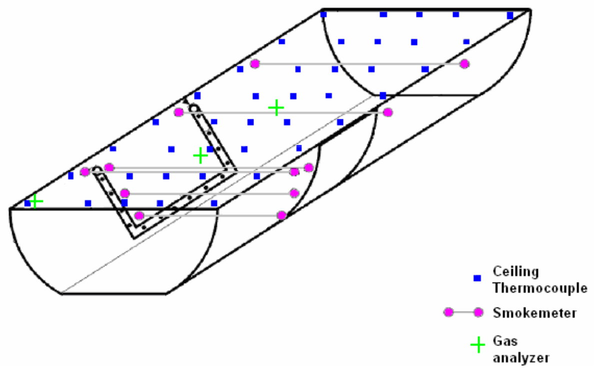


Figure 2 - Forward 707 cargo compartment: 910 ft^3, volume

The Reynolds-Averaged Navier-Stokes (RANS) and energy equations, in conjunction with a passive scalar model, are applied to solve the transient problem using a CFD code (Fluent package). The airflow is assumed following an ideal-gas law and the turbulence effects are taken account employing an eddy viscosity based model (standard k-ε) with a wall-function treatment. Thus, the governing equations (mass conservation, momentum, energy and turbulence model) are stated as:

$$\frac{\partial}{\partial x_i}(\rho \bar{u}_i) = 0 \tag{1}$$

$$\rho\frac{\partial \bar{u}_i}{\partial t} + \rho \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_i} = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left[(\mu_L + \mu_t)\left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i}\right)\right]; \quad \text{with } \mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \tag{2}$$

$$\rho\frac{\partial \bar{T}}{\partial t} + \rho \bar{u}_j \frac{\partial \bar{T}}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\frac{\mu_L}{Pr} + \frac{\mu_t}{\sigma_t}\right)\frac{\partial \bar{T}}{\partial x_j}\right] + S_T \tag{3}$$

$$\rho\frac{\partial k}{\partial t} + \rho \bar{u}_j \frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\frac{(\mu_L + \mu_t)}{\sigma_k}\frac{\partial k}{\partial x_j}\right)\right] + \mu_t\left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i}\right)\frac{\partial \bar{u}_i}{\partial x_j} - \rho\varepsilon \tag{4}$$

$$\rho\frac{\partial \varepsilon}{\partial t} + \rho \bar{u}_j \frac{\partial \varepsilon}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\frac{(\mu_L + \mu_t)}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial x_j}\right)\right] + C_1\mu_t\frac{\varepsilon}{k}\left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i}\right)\frac{\partial \bar{u}_i}{\partial x_j} - C_2\rho\frac{\varepsilon^2}{k} \tag{5}$$

where x_i = Cartesian coordinates, and $\bar{u}_i$ = corresponding average velocity components; t = time; $\bar{T}$ = average temperature; k = turbulent kinetic energy; ε = turbulent energy dissipation rate; ρ = fluid density; μ_L = molecular viscosity; μ_t = turbulent viscosity; Pr = molecular Prandtl number; S_T = volumetric heat source-term. The constants of the standard k-ε turbulence model are given by the following values: C_μ = 0.09, C_1 = 1.92, σ_k = 1.0, σ_ε = 1.3 and σ_t = 1.0.

In the passive scalar approach the transported quantity must have an enough low concentration and non-reactive to be considered passive with respect to the air. At the present work, the smoke components CO, CO_2 and soot are represented by (ϕ_k) and its transport are modeled as:

$$\frac{\partial \phi_k}{\partial t} + \bar{u}_j \frac{\partial \phi_k}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\frac{\mu_L}{Sc} + \frac{\mu_t}{Sc_t}\right)\frac{\partial \phi_k}{\partial x_j}\right] + S_k \tag{6}$$

where ϕ_k = transported variable, S_k = source term, Sc_t = turbulent Schmidt number and Sc = molecular Schmidt number. The molecular Schmidt number is calculated as:

$$Sc = \frac{\mu/\rho}{D_{AB}}, \qquad D_{AB} = \text{specie A to specie B diffusivity coefficient.} \tag{7}$$

As the N_2 concentration in the air is higher (about 78%) than the other components, the air is considered only composed by N_2. As smoke concentration is low, it is supposed that there is no diffusion among these smoke components CO, CO_2 and soot. Each smoke component and the N_2 is considered a binary mixture. The diffusivity coefficient D_{AB} for a non-polar binary mixture is a function of temperature, pressure, and composition At low pressures, D_{AB} is inversely proportional to the pressure, increases with increasing temperature, and is almost independent of composition for a given gas-pair. The equation for estimation of D_{AB} (Bird, 1960) is given by:

$$\frac{pD_{AB}}{(p_{cA}p_{cB})^{1/3}(T_{cA}T_{cB})^{5/12}\left(\frac{1}{M_A}+\frac{1}{M_B}\right)^{1/2}} = a\left(\frac{T}{\sqrt{T_{cA}T_{cB}}}\right)^b \tag{8}$$

with $a = 2.745 \cdot 10^{-4}$ and $b = 1.823$. Table 1 presents the CO, CO_2 and soot diffusivities in N_2, obtained applying Eq. 8, and used as input data in the CFD code.

Table 1 - Smoke components and its diffusivity

Component A	**Component B**	**D_{AB} (m^2/s)**
CO	N_2	1.6 e-5
CO_2	N_2	2.0 e-5
Soot	N_2	9.9 e-6

The experimental heat release rate curve (Figure 3) obtained during the resin block burning performed by FAA test (Blake, 2000) were adjusted and included in a User Defined Function (UDF), which is a piece of external C code that can be linked to FLUENT code. These UDFs are used as input to a volumetric heat source-term (S_T in Eq. 3).

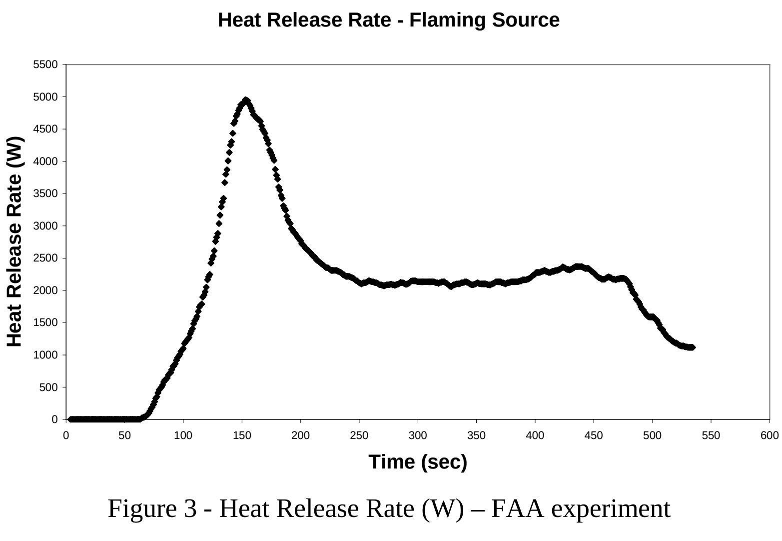


Figure 3 - Heat Release Rate (W) – FAA experiment

Note that after ignition, a fast increase in the heat release rate occurs reaching a peak, then a sharp decrease followed by a plateau, or slow reduction, and finally there is a decrease until the fire extinction. Analog curves for the CO, CO_2 and soot concentrations were extracted from the experimental data. After they were also used as input data (S_k in Eq. 6) employing an UDF C-language routine.

Initial values (pressure field, velocity components, k and ε) are set to zero and initial compartment temperature is 293 K. Boundary conditions at the walls, roof and floor (Figure 1) are no-slip flow and also a constant temperature (293 K).

CFD simulations can be very costly and one of the most significant factors influencing the computation time is the size of the computational mesh. It is important to determine an appropriate grid size for a given computational domain, in order to accurately capture the flow features, but not over-resolving by an excessive number of grid points. A grid sensitivity study was performed and the finest mesh (generated with ANSYS ICEM CFD 5.1) has a total of 98,304 hexahedral elements and is shown Figure 4.

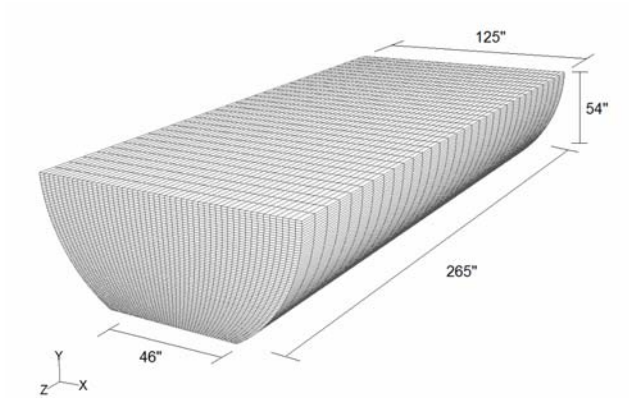


Figure 4 - Mesh 3 with 98,304 elements

For the compartment numerical simulation, the governing equations, Eq. (1) to Eq. (6), are solved using the finite volume technique and are treated employing second-order discretization method. The incompressible flow hypothesis is enforced and the PISO pressure-velocity coupling scheme in conjunction with a standard k-ε model (enhanced wall treatment). A dual-time step transient analysis is performed with a time step size 0.1 s, 1800 time steps and a max of 30 iterations per time step.

2. RESULTS

In order to study the numerical solution accuracy, the time-evolution temperature profiles obtained using the present CFD model based on FLUENT and a FAA code developed by Sandia Labs. (Blake, 2004) are compared with experimental results. Figure 5 shows this comparison along the thermocouples installed in the cargo compartment ceiling.

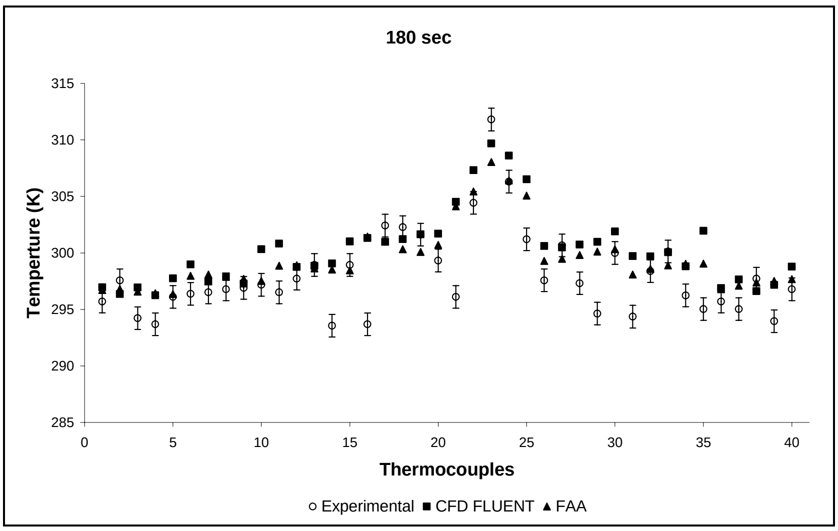
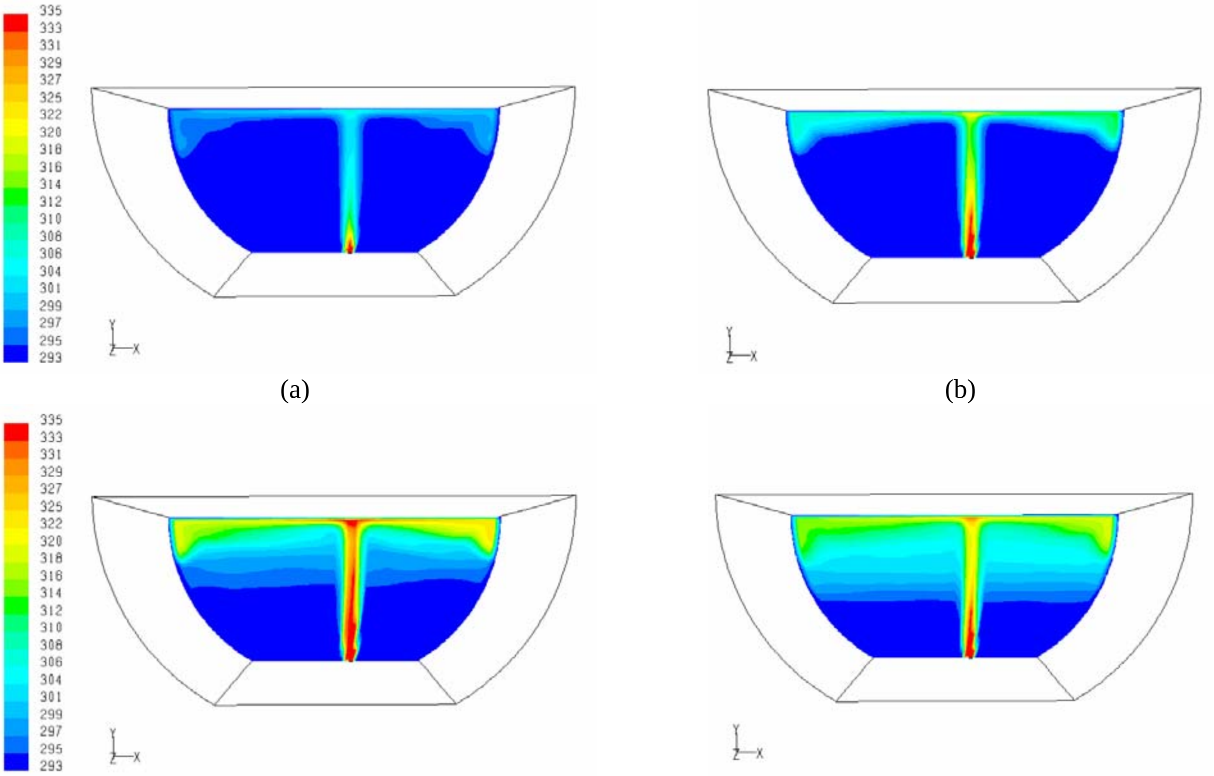


Figure 5 - Temperature profile along the ceiling of the compartment after 180 seconds

The choice of the grid resolution is an important issue for the good prediction of the temperature in the fire area (Bounagui et al, 2004). So, the mesh illustrated in Figure 4 was chosen once it has better captured the peak in the temperature distribution exhibited in Figure 5.

Figure 6(a)-(d) shows the temperature time-evolution over plume mid-plane cross section of the cargo compartment. After the burning process start, the plume temperature increases progressively as the reactive heat release (Figure 3) also elevates, cumulating hot fluid along the cargo compartment ceiling.

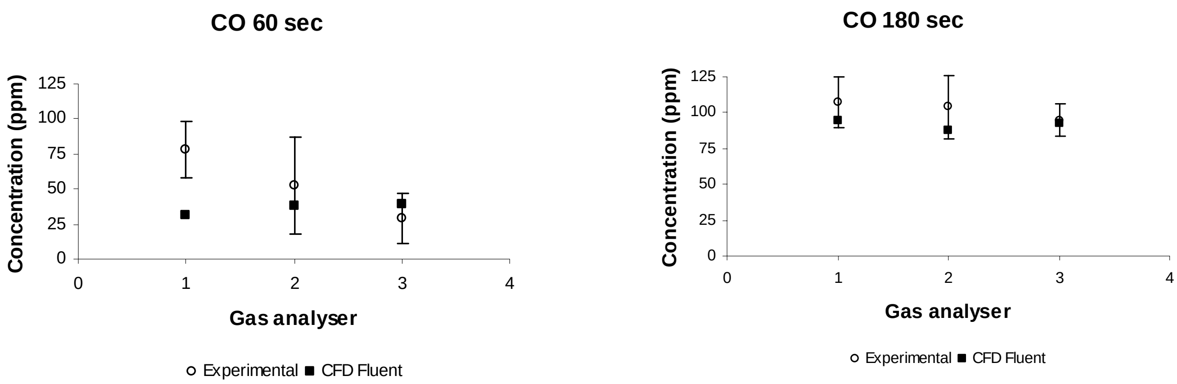


(a) (b) (c) (d)

Figure 6 - Temperature contours at the plume mid-plane: (a) 30 s; (b) 60 s; (c) 120 s and (d) 180 s.

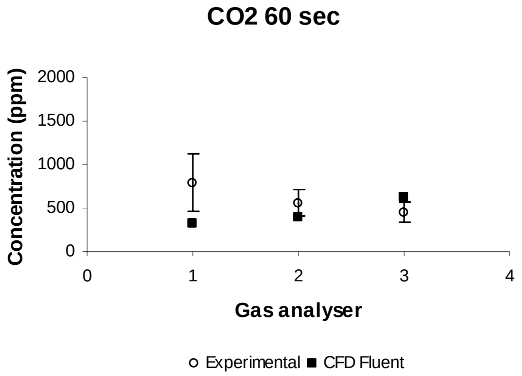
The maximum temperature value occurs at the ceiling plume stagnation point, as already verified in Figure 5. Besides, Figure 6(d) presents a temperature reduction following the heat release rate decrease. The hot fluid layer accumulated in the top gradually fulfills the cargo compartment volume. This study allows the suitable smoke detector positioning and evaluating its time response after fire ignition.

Figure 7-8 presents the reaction products concentration of the resin block burning, CO and CO_2, at three gas analyzers (shown in Figure 2). Note that, as expected, the CO_2 concentration level is higher than the CO one at all time-instants due the almost complete resin oxidation. Numerical predictions for CO concentration are in good agreement with experimental results, except the 60 s CO concentration for gas analyzer #1 that was outside the experimental error range.

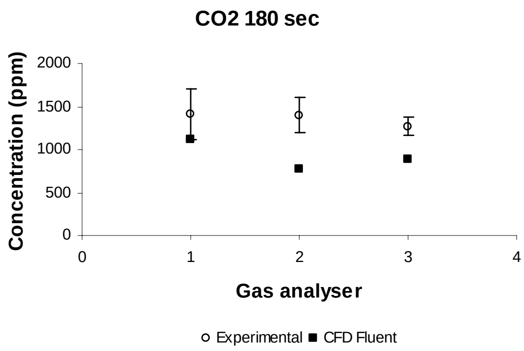


(a) (b).

Figure 7 - CO concentration: (a) 60 s; (b) 180 s.



(a)



(b)

Figure 8 – CO_2 concentration: (a) 60 s; (b) 180 s.

Differences between CFD and experimental results for CO_2 concentration are lesser at the reaction start but increases with time. As mentioned in the correlated literature, a possible reason for this discrepancy is the soot fouling on CO_2 gas analyzer. This fact (verified in experiment photographs provided by Suo-Antilla, et al, 2003) can slightly affect the experimental measurements quality.

4. FINAL REMARKS

At the present work, the smoke transport in an aircraft cargo compartment was numerically simulated. Results showed that the passive scalar adopted methodology is suitable to capture the airflow patterns, temperature and smoke components spatial distributions. As the numerical results presented a good agreement with available experimental data, this CFD procedure is suitable to locate smoke detector on aircraft cargo compartment ceiling, evaluating the time interval between fire start and the alarm generated by its sensor. This work also shows that the CFD tool can contribute to reduce both the elevated time and costs attributed to experimental tests in the aeronautical manufacturing.

5. REFERENCES

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

Bird, R. B.; Stewart, W. E., 2002, Transport Phenomena, John Wiley. New York.

Blake, D., 2000, Aircraft Cargo Compartment Fire Detection, International Aircraft Systems Fire Protection Working Group Atlantic City, NJ August 29-30.

Blake, D., 2004, The FAA Smoke Transport Code - Users Manual, FAA William J. Hughes Technical Center, November 16.

Bounagui, A., Kashef, A., and Bénichou, N., 2004, CFD Simulation of the Fire Dynamic for a Section of a Tunnel in the Event of Fire, Proceedings of the12th Annual Conference of the Computational Fluid Dynamics Society of Canada, Ottawa, May 9, pp. 1-8.

Chiang C. K., Lieh W.Y. and Sheng T. H., 2003, Fire Model Analysis and Experimental Validation on Smoke Compartments, Journal of Fire Sciences, vol. 21.

Chen, J., Chen H. and Fu, Song., 2005, Numerical Investigation of Fire Smoke Transport in the Tsinghua University Sports Center, Tsinghua Science and Technology, 10(5), pp 618-622.

Chown, W. K., and. Yin, R., 2003, A New Model on Simulating Smoke Transport with Computational Fluid Dynamics, Department of Building Services Engineering, 11.

Sinai Y. L., Sykes, N., and Everitt, P., 2004, Validation of a CFD model for several displacement ventilation geometries, ROOMVENT, Coimbra, Portugal.

Stribling D., 2001, Fire Modeling - FLOVENT -Validation Report, available at www.flomerics.com/flovent/technical_papers/v35.pdf, accessed in 04/18/2007.

Suo-Anttila J., Gill W., Gallegos, C. and Nelsen, J., 2003, Computational Fluid Dynamics Code for Smoke Transport During an Aircraft Cargo Compartment Fire: Transport Solver, Graphical User Interface, and Preliminary Baseline Validation, DOT/FAA/AR-03/49.

5. RESPONSIBILITY NOTICE

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