Building Simulation 2005

Ninth International IBPSA Conference
Montréal, Canada
August 15-18, 2005

AIRFLOW MODELING AND FIRE SMOKE PROPAGATION IN THE NEW ECOLE POLYTECHNIQUE BUILDING

Daniel Barrero, Eddy Petro, Juan Abanto, Marcelo Reggio,
École Polytechnique de Montréal. 2900 boul. Édouard-Montpetit.
Campus de l'Université de Montréal, Montréal (Québec), Canada H3T 1J4
email: Marcelo.Reggio@polymtl.ca
http://www.professeurs.polymtl.ca/marcelo.reggio

ABSTRACT

This paper addresses the application of fire and smoke simulation and computational fluid dynamics tools to events taking place in the new engineering building on the campus of the University of Montreal. The development steps include 3D CAD modeling of the entire building, geometry transfer and airflow, and fire simulation with enhanced rendering considering the optical properties of smoke and fire. The Fire Dynamics Simulator solver developed at NIST has been used to generate the necessary data for the fire visualization. The commercial software Fluent has been used to carry out the calculate airflow simulation.

1. INTRODUCTION

One of the most fascinating aspects of computer-based simulation is the creation of digital replicas of reality. In fact, nowadays more and more 3D models are being constructed and assembled to create virtual worlds, and this is having a profound impact at the scientific and engineering levels. However, scientists and practitioners need not only geometric models while working in the parallel, virtual world, but also the physics surrounding them in order to duplicate real events.

In this line of thinking, our study concentrates on the prediction of ventilation and fire and smoke propagation in buildings. The specific case is the new building currently under construction at the École Polytechnique de Montréal. Interest in such a problem revolves around the impact that air quality, in normal operating conditions, and smoke concentration, in the event of a fire, may have on the occupants of these buildings. Simulating these phenomena is a complex task, both scientifically and technologically.

From the scientific perspective, it has been found that fire is a hot and turbulent gas of irregular shape, and smoke is a fuzzy phenomenon. Modeling fire is achieved via the Navier-Stokes equations coupled with turbulence models, the combustion chemistry of the processes and equations describing the transport of species. Now, the representation of the fire phenomenon is not easy, because it involves a flame front separating fresh and burnt gases, smoke density, visibility and illumination.

In terms of technological challenges, detailed modeling of airflow and air quality parameters in complex structures requires CFD implementation for a precise understanding of how properties (velocity, MAA, etc.) behave within particular spaces such as classrooms and atria. This requires significant modeling effort at the geometric level, expertise for meshing complex spaces, including furniture, and, obviously, computational resources for calculating the solution.

For this work, the 3D model of the entire building was constructed using Rhino[1] on the basis of 2D Autocad information. These common data were then shared by complementary CFD and fire simulation approaches. Airflow circulation and smoke distribution were calculated according to a fire scenario occurring in our future engineering building.

For the airflow simulation, Fluent's Gambit preprocessor[2] was selected as the starting element for the meshing process. Then, Fluent's [3] finite-volume flow solver was applied to carry out the flow simulation.

For fire simulation, the FDS computer program developed at NIST[4,5] was applied. Since this solver uses a Cartesian mesh and requires "shape adaptation" to convert objects generated by standard CAD programs into objects defined by a 3D Cartesian grid, a previously reported dicemerge algorithm [6] was applied to translate the CAD model of the building into the format needed by FDS.

Although a full 3D model of the entire construction was produced, the applications only tackle smaller areas. Specifically, CFD has been applied to the study of airflow circulation in a classroom, while fire simulation has been applied in the central part of the building which encompasses the atrium and adjacent areas.

The final step of the complementary applications involves visualization. For a better characterization of the building's volume, an architectural-type of rendering was applied on the original model. For the fire application, a rendering which accounts for the optical properties of smoke and fire considering particle system and fuzzy blob techniques is briefly described. The optical properties of the particles and field parameters resulting from the FDS simulation at a given point in space are used to control the translucency, illumination and influence over the color of the objects around a particle.

It is believed that the techniques and results presented here, which include temperature distribution, air freshness, carbon monoxide concentration, environment visibility and chronology of events, can be useful in the design and fire protection design phases of a building characterized by the presence of a relevant number of people.

2. THE FIRE SIMULATION ENVIRONMENT

The FDS fire solver characterizes the simulation space by means of a uniform Cartesian grid containing the description of the objects (obstacles), as well as the fluid space To define the obstacles, the objects are approximated by a series of axis-aligned parallelepipeds which cover most of the volume of the object. This description can be created by hand for simple scenarios composed of objects of basic geometric shapes. The problem arises when attempting to model more complex scenarios containing organic shapes, especially when modern architectural tendencies and natural objects are to be included in the simulation, like humans or other curve-shaped obstacles.

In general, to solve the difficulty of manually creating the many parallelepipeds required by FDS, we take advantage of the geometries created by engineering and architectural modelers. In this study, the model was created with the commercial tool Rhino[1], which uses NURBS modeling technology. The configuration is rich in detail and is transformed into a Cartesian-based geometry by applying a two-pass dice-and-merge algorithm described in Ref.[6]. For a more realistic visualization of the results computed in the Cartesian world, the data obtained from the simulation step are later projected back onto the original model.

After the preprocessing step, the driving parameters (inlets, outlets, fire sources, chronological events) are defined and the actual fire simulation can take place. The code uses a finite-volume method to solve the Navier-Stokes equations, and turbulence is modeled using a large-eddy simulation (LES) approach [7]. Fire dynamics is completed by a mixture fraction combustion model, species transport and a pyrolysis model, along with the energy and the radiative transport equations. Fire growth and spread are calculated using thermal boundary conditions and the burning characteristics of the materials involved [4-5].

3. VISUALIZATION

Once fire-related data are generated by the numerical model, typical elements like fire and smoke need to be represented. In spite of the maturity of CFD technology, fire simulations differ from conventional CFD applications. In real-life fires, a major part of the flow field is not invisible (flames and smoke), while fresh air is. Thus, new techniques need to be applied to provide a realistic representation of fire on-screen which takes into account the optical properties of the burning gases. We briefly mention below aspects related to flame identification, illumination and rendering. More details can be found in [6].

3.1 Flame illumination and rendering

Smoke, flame and the flame surface that corresponds to a very thin fuzzy boundary surrounding the volume of the flame, are represented as a series of fuzzy blobs based on the mixture fraction and heat release rate per unit volume (HRRPUV) of the gas. Fuzzy blobs [8] are weightless particles representing a density distribution on a portion of the space. These particles are tagged as free air or burning particles based on the simulation results. Each particle is then associated with a potential field to produce an implicit representation of the smoke and fire density volume for every point in space.

The density distribution is an implicit representation based on the mixture fraction and HRRPUV of the gas over a volume of a given radius around the particle center (Fig. 1).

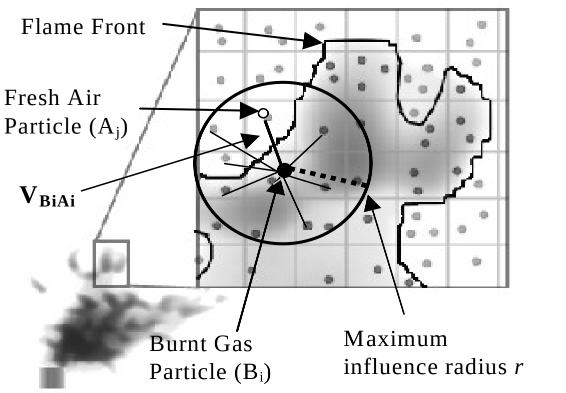

Fig. 1. Flame front

In addition, the distribution of the volume for each blob ρ_i will be weighted by a relation between burned and unburned particles to keep the distribution enclosed by the flame surface. This is given by:

$$fB_i(x,y,z)=1-\left(\frac{r^2-\sum_{Y_{Aj}\in A}(Y_{Aj}-Y_{Bi})*(<x-x_{Bi},y-y_{Bi},z-z_{Bi}>\cdot \mathbf{V_{BiAj}}))}{(x-x_{Bi})^2+(y-y_{Bi})^2+(z-z_{Bi})^2}\right)$$

in which $\boldsymbol{r}$ indicates the radius of each blob before formation, $\boldsymbol{x,y,z}$ denote the coordinates of a given blob and $\boldsymbol{Y}$ its mass fraction, $\boldsymbol{B_i}$ is an index for the current blob and $\mathbf{V_{Biaj}}$ is the vector pointing to each of the neighboring particles.

$$f_{flamesurf}\ (x,y,z)=\sum_{B_i\in B} fBi\ (x,y,z)$$

The next step to consider in fire visualization is the contribution of energy from the flame to the environment. To make sure that all the objects receive the luminosity emitted by the fire, the local temperature through radiation is taken into account. However, the resulting spectrum obtained from blackbody emission using Planck's law[9] has a very marked tendency towards the red side of the spectrum at lower temperatures. This problem has been solved by applying a shifting based on the chemical composition of the burning materials [6]. Finally, the actual rendering of flame and smoke takes place with OpenGL [10].

3.2 The digital ventilation layout

In this part of our work, the intention is to apply CFD technology to obtain solutions driven by the configuration of the classroom ventilation set-up. Today, the CFD avenue, which was not initially intended for building engineering, is used by commercial and research groups around the world, among them [11,12,13,14]. Traditional CFD involves geometric modeling, mesh generation, flow solution plus visualization and solution analysis.

The geometry of the classroom is extracted from the full building model and additional elements like four-way diffusers, heaters, desks and chairs have been incorporated into the scene. Among these objects, we distinguish the diffuser, which plays a key role in airflow distribution.

The grid for the classroom is generated with Gambit[2], the Fluent [3] preprocessor. For the current application, hexahedral and tetrahedral elements are used.

The flow solution step involves the numerical solution of a set of partial differential equations describing the fundamental laws of fluid motion. Mass, momentum, energy, turbulence and species conservation equations were solved using the commercial flow solver Fluent [3]. This uses the finite-volume method, which is one of the most widespread approaches in CFD. Then, the solution was converged, and computed basic parameters, including air temperature, relative humidity and air velocity, were combined to obtain the mean age of the air (MAA).

4. RESULTS

4.1 Fire simulation

The methodology has been applied to the computation of a fire scenario and airflow calculation involving the new building at our university. The building has seven floors and the dimensions are 76x105x40 meters.

The full 3D model was created using the Rhino [1] software based on 2D Autocad files. Fig. 2a displays the model which has been cut to highlight the interior. An architectural type of rendering of the building, applied on the original model, can be seen in Fig. 2b.

In the edifice, an imaginary fire develops near the bathrooms on the ground floor. On the basis of average values used in real fire tests for such a construction, a large, 6 MW source was imposed and propane was chosen as the fuel. The simulation was conducted over 300 seconds, applying the hypothesis that the systems installed for the extraction of smoke and sprinklers are not operational. The walls are made of concrete and no furniture was considered.

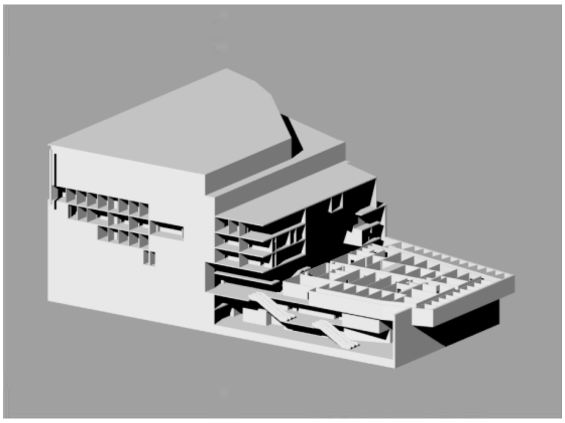

Fig. 2a. Model and interior

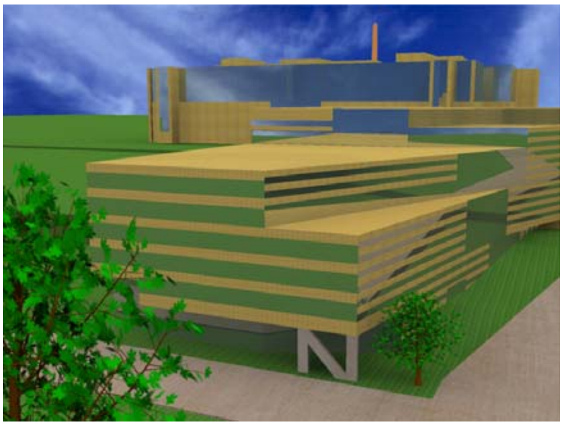

Fig. 2b. Rendered model

The goal of the study is to predict and analyze the propagation of the smoke through the corridors and in atrium generated by a virtual fire. The area where the fire simulation is considered is located towards the center of the building, and is shown in Fig 3. It includes stairs, which are obviously important for the evacuation of the occupants for whom smoke concentration and visibility are crucial parameters to consider.

Figs. 4a and 4b illustrate fire and smoke spreading at 30 and 120 seconds after ignition. Guidelines have been added to provide a better understanding of how fire and smoke spread through the corridors.

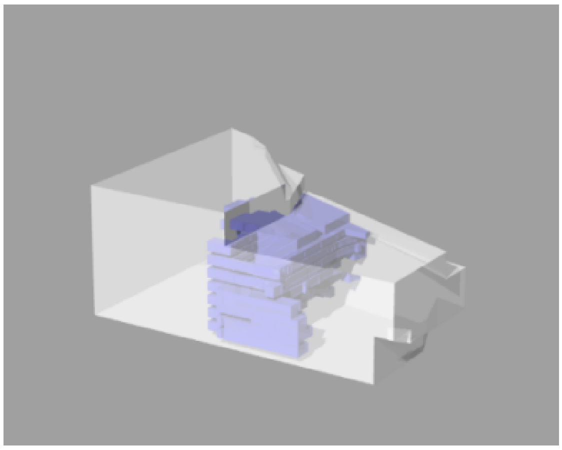

Fig. 3. Simulation region

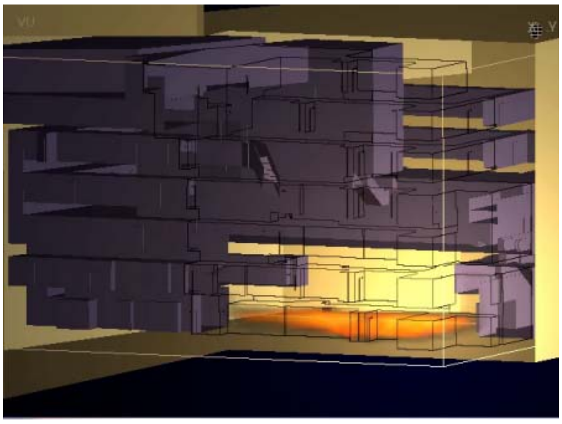

Fig. 4a. Fire propagation at 30 seconds.

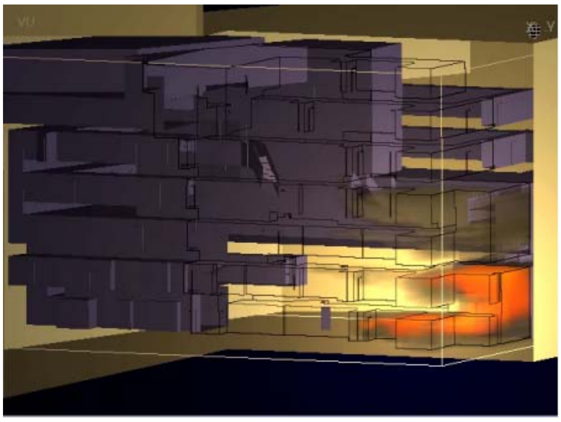

Fig. 4b. Fire propagation at 120 seconds

In order to obtain detailed information, a set of 52 sensors was distributed all over the area being studied.

The fire computed with FDS predicts a reasonable behavior of a fire spreading for the occurrence considered, which was extreme. This shows that the fire completely engulfed the central part of the ground floor after only two minutes, with temperatures near a peak of 500ºC. The flames had already progressed throughout the first and second floors, rapidly reaching the stairs at the fourth level.

In Figs. 5a and 5b, the evolution of CO concentration and temperature levels are presented at selected locations on different floors. The sensors chosen are located near various stairways between the third and the seventh floor. Although the exact position of these sensors is known, these are not specified here because, in the current context, this information provides no greater insight.

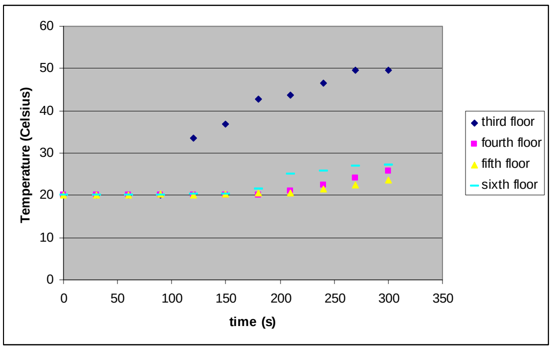

Fig. 5a. Temperature evolution

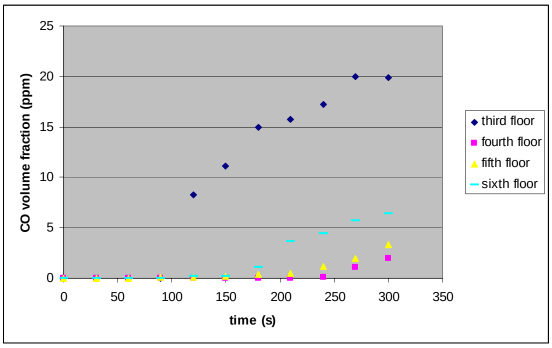

Fig. 5b CO concentration

4.2 Classroom ventilation

A classroom in our future engineering building located on the campus of the University of Montreal was selected to conduct a ventilation study. The volume of the classroom is 780 m^3. Four four-way diffusers and eight fluorescent panels have been inserted in the ceiling, together with 6 heaters along the bottom part of the walls.

The mass fraction of water was 0.006, while the temperature at all inlets was 290 K. The turbulent intensity at one inlet and one outlet was 1%. All four diffusers provide 1 ARCH of fresh air (or 780 m^3/hour). The heaters add 150 W/m^2 and the fluorescents 50 W/m^2.

For the current application, the widely used re-normalization group (RNG) $\kappa-\varepsilon$ was selected for the treatment of the turbulence, especially near the air-supply diffusers where strong streams of air are present. Adiabatic wall conditions were chosen for the vertical walls, the ceiling, the floor, the chairs, the inlets, the outlets and the tables. No radiation was considered at this stage. The mesh size used for this calculation has 2054671 cells and 456362 vertices. It should be noted that it was not possible to include all the diffusers or all the ceiling lights and heaters, due to the large number of grids that this would require. In fact, each diffuser requires about 110000 grid cells. This large number of elements is important because the diffuser's shape plays a key role in the air distribution in the room. Figs. 6a and 6b (not to the same scale) show cuts on the mesh in the y and z planes in the diffuser region respectively.

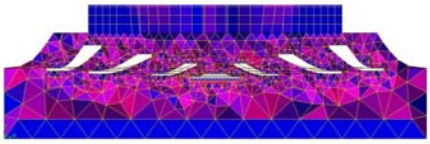

Fig. 6a. Diffuser: meshing in the y plane

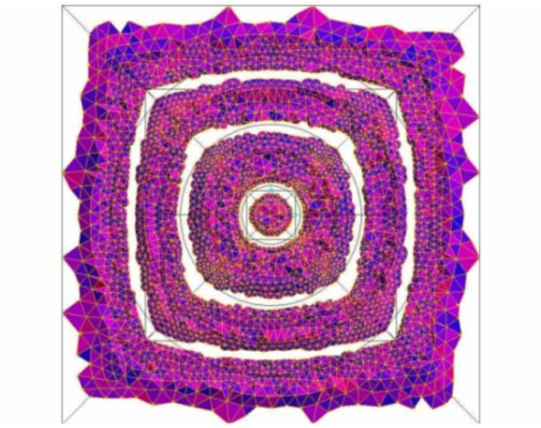

Fig. 6b Diffuser: meshing in the z plane

In Fig. 7a, the pathlines of particles from the diffusers are shown. Here, the benefit of the shape of the ceiling diffusers, which allow good circulation of the air in the room close to the ceiling, can be seen.

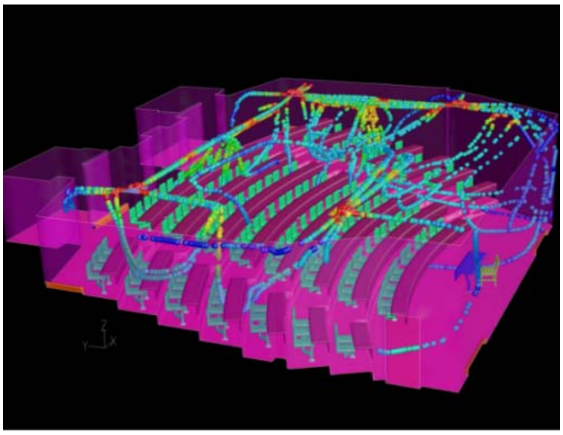

Fig. 7a. Streamlines in the classroom

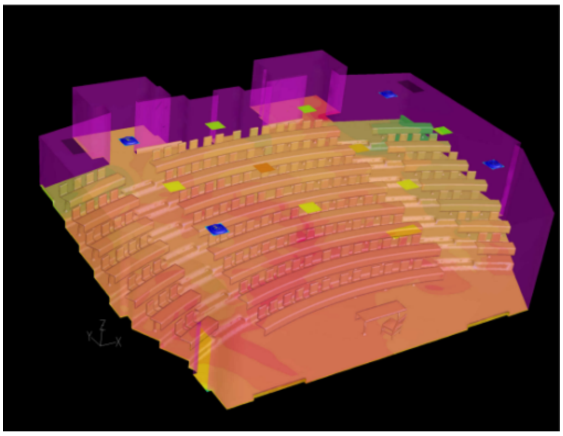

Fig. 7b MAA distribution on the floor

Once the airflow simulation had been carried out, the mean age of air (MAA) thermal comfort quantity, which gives an indication of the air “freshness”, has been calculated as a user-defined scalar following the equation:

$$\frac{\partial}{\partial x_i}\left(\rho u_i \phi - \Gamma \frac{\partial \phi}{\partial x_i}\right) = S_\phi$$

where ϕ is the MAA scalar, ρu_i is the mass flow rate, $\Gamma=2.88\rho \times 10^{-5}+\mu_{EFF}/0.7$ is the diffusion coefficient of ϕ and S_ϕ the source term, which depends on the density of the air. Fig. 7b shows the MAA distribution on the floor, with better freshness found near the sides of the room, while the professors’ area indicates less renewed air because of the greater distance between the floor and the ceiling in this area.

In order to obtain a better assessment of the numerical solution, two column sensors were placed on the aisles at the back of the room. Fig. 8 indicates their location. The endpoints of these sensor bars are (-5,0,0.61),(-5,0,3.1) and (+5,0,0.61), (+5,0,3.1). The temperature and the MAA distribution along these bars, from the floor to the ceiling, are shown in Figs. 9a and 9b. These show a slight variation of these parameters on the two sides of the room (line 1, left, and line 2, right), indicating, from the properties’ variation point of view, a well-designed ventilation system.

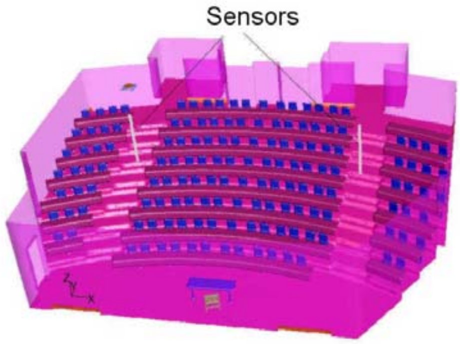

Fig. 8. Sensor locations

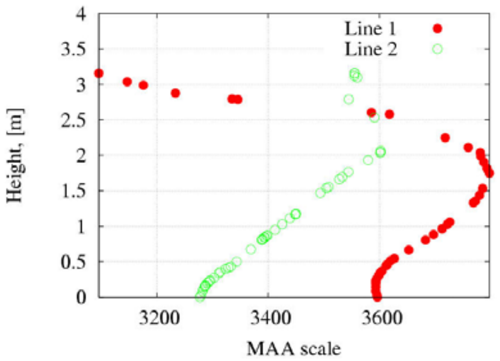

Fig. 9a. MAA distribution.

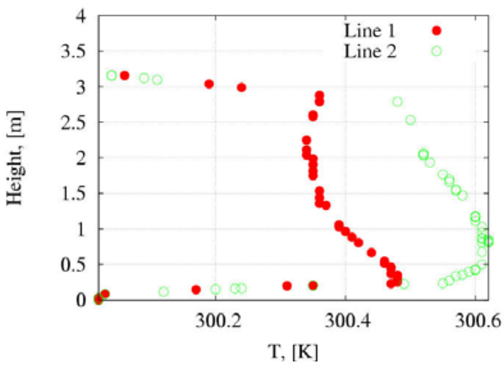

Fig. 9b   Temperature distribution

5. CONCLUSION

This paper has described research activities in building simulation related to indoor ventilation, smoke movement and fire propagation. A detailed 3D model of the new building at the École Polytechnique de Montréal (University of Montreal) has been constructed using the NURBS-based modeler, Rhino. Simulations of fire propagation and airflow ventilation were carried out in different areas of the building.

The commercial program, Fluent CFD, was used to analyze the data generated by models representing the complexity of the flow patterns that evolve inside the ventilated spaces of a classroom. This CFD analysis allowed us to predict air flow conditions and to estimate the MAA in the room. Real-world geometries, like diffuser inlets, have a strong influence on overall fluid flow behavior and are a must for this type of analysis.

Fire simulation was conducted on the central area of the building using the FDS program developed at NIST. A photorealistic visualization technique has been developed and adapted to represent data obtained with this fire simulator.

The results predict a plausible fire and its combustion products spreading along corridors, stairways and the atrium. Realistic depiction of flame position and smoke density (visibility) make the phenomena easier to understand and is thus useful for increasing the occupants' awareness, for researchers and for emergency planners.

Now that efforts invested in the modeling of the building have been completed and primary tests have been conducted, a study coupling fire simulation, ventilation and smoke evacuation is required to respond to more realistic conditions.

REFERENCES

[1] Rhinoceros, "NURBS modeling for Windows, version 3," Robert McNeel & Associates, 2002

[2] Fluent Inc.'s "Gambit 2.0," Fluent Inc., December 2001

[3] Fluent Inc.'s "Fluent 6.0," Fluent Inc., November 2001

[4] McGrattan, K.B., Forney, G.P., Prasad, K., Floyd, J.E., Hostikka, S. 2002. Fire Dynamics Simulator (Version 3) – User's Guide. NIST, 81p. NISTIR 6784

[5] McGrattan, K.B., Forney, G.P., Rehm, R.G., Baum, H.R., Prasad, K., Floyd, J.E., Hostikka, S., Hamins, A. 2002. Fire Dynamics Simulator (Version 3) – Technical Reference Guide. NIST, 51p. NISTIR 6783

[6] Barrero, D., Abanto J., Hardy J-P., Reggio M., "Serious color full dynamics in fire simulation," AIAA paper 2005-1151, Reno, 2005

[7] Piomelli, U., Large-eddy Simulation: Achievements and Challenges. Progress in Aerospace Science, Vol. 35, pp.335-362, 1999

[8] Stam, J., Fiume E., Turbulent Wind Fields for Gaseous Phenomena, ACM Computer Graphics, SIGGRAPH 93, USA, Addison-Wesley, 369-373, August 1993.

[9] Gaussorgues, G., "Infrared Thermography," Microwave Technology, Vol. 5, pp. 11-52, 1994.

[10] Shreiner, D., Woo, M., Neider, J., Davis, T., OpenGL Programming Guide: The Official Guide to Learning OpenGL, Version 1.4, 4th edition, Addison-Wesley, 2003.

[11] AFC, www.airflow.ethz.ch

[12] Department of Mechanical Engineering at the University of Strathclyde in Glasgow, www.strath.ac.uk

[13] Centre for Building and Systems at Technische Universiteit Eindhoven of The Netherlands, www.kcbs.nl

[14] Department of Building at National University of Singapore, www.bdg.nus.edu.sg