

DIFFERENT APPROACHES FOR THE SIMULATION OF AN EXPERIMENTAL BUILDING HOSTING A CLIMATE CHAMBER DEVOTED TO ARTIFICIAL FOG PRODUCTION

Philippe ANDRE*, Nick KELLY**, Jean-Jacques BOREUX*, Patrick LACÔTE*, Christophe ADAM*

*Department of Environmental Monitoring, Fondation Universitaire Luxembourgeoise

B-6700 ARLON, Belgium

** ESRU, University of Strathclyde, GLASGOW, Scotland

andre@ful.ac.be; nick@esru.strath.ac.uk; boreux@ful.ac.be; lacote@ful.ac.be; c.adam@ful.ac.be

ABSTRACT

In the context of a european project dealing with the issue of transport safety improvement in fog conditions, an experimental building has been constructed on the FUL campus, in the southern part of Belgium. This building hosts a climatic chamber in which a given indoor climate is to be maintained (temperature and relative humidity) whatever the external climate, in order to promote the artificial production of fog by water droplets spraying.

During the design of the building and of the HVAC plant, different simulation approaches were carried out in order to evaluate and possibly optimize the technical choices concerning:

- the building envelope
- the material for the climatic chamber
- the control strategy of the HVAC system

Two categories of simulation tools were used to solve the different tasks:

- building simulation tools
- CFD programs

For the first category, simulations were performed using both ESP-r and TRNSYS with the objective of solving the following questions: optimization of the building envelope (orientation and shading of the building, insulation, fenestration); sizing of the HVAC system; comparison of control strategies (direct control, indirect control, radiative and/or convective conditioning); estimation of energy consumption and comfort

Within the second category, the FLUENT and ESP-r software were used and compared in order to predict the homogeneity of the thermal state within the test room and to calculate the effect of injecting a high amount of small water droplets on fog production, maintenance and visibility. Therefore, the ESP-r CFD module program was augmented with a specific model of water droplet diffusion.

The paper will tackle the main modelling assumptions at the basis of each simulation experiment, the data preparation and the results of the calculations. For each phase, advantages and limitations of the simulation approach will be highlighted.

These developments are considered as a first step towards the implementation of a virtual testing environment in the field of automotive transport which could be complemented by additional issues dealing with passengers comfort, acoustics, air quality, level of illuminance generated by the virtual production of fog.

INTRODUCTION

In the context of a european project dealing with transport safety, a new experimental building has been constructed in the FUL campus in Arlon, South of Belgium. The building (fig. 1) includes a chamber, totally separated from the external environment, in which the internal climate is controlled in order to maintain the adequate conditions for the artificial production of fog. Therefore, the building is equipped with a specific HVAC system. Non conventional energy solutions were selected (solar collectors, reversible heat pump) for environmental reasons. The design of the building as well as that of the HVAC system was supported by different numerical simulation approaches. The aim of this paper is to show how simulation was used, to support different aspects of the design:

- design of the building envelope
- design of the climatic chamber
- design of the HVAC system control strategy

A presentation of the general Building Life Cycle framework in which this simulation work took place is given in (André et al, 2002).



Fig.1: General view of the simulated building

Two categories of simulation tools were used in this work:

- building simulation tools: ESP-r (Clarke 2001) and TRNSYS (Klein et al, 2000)
- CFD simulation tools: the ESP-r CFD component (Clarke 2001) and FLUENT¹

The following paragraphs will show the application of these different tools all along the different phases of the Building life Cycle.

SIMULATION AIDED DESIGN OF THE BUILDING ENVELOPE

The design of the building was mainly assisted by the TRNSYS software.

Objectives and constraints

The design of the building was influenced by a number of constraints:

- budgetary limits
- available space on the FUL campus
- time available for building construction

In order to improve the control in the climate in the test room, it was intuitively decided to use the concept of “double-envelope” with the test room totally separated from the external conditions by a space (a so-called “buffer space”) where temperature would be controlled.

The combination of the program and the constraints led to a conceptual design which is shown by fig. 2.

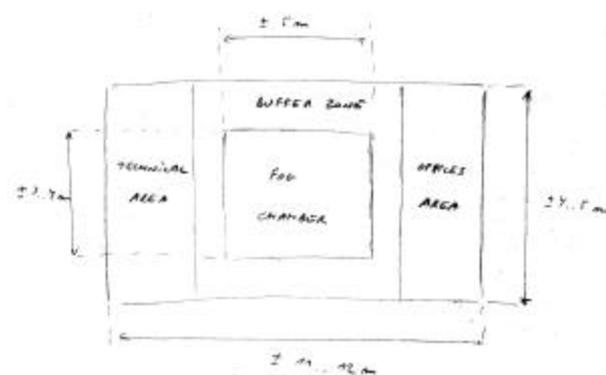


Fig. 2: Conceptual design of the building

The selected dimensions are a compromise between cost and functionality. Fig. 2 identifies 5 specific zones in the building : the fog chamber, the buffer zone, the offices zone, the technical area, the attic which is situated above the fog chamber and the buffer zone.

Aside of that, the building project was explicitly addressing environmental issues, as being developed within an Institution devoted to environmental

research and education, with the objective of building a pilot project. This general objective was translated into a number of concepts:

- Maximum use of "natural" (and consequently renewable) energy sources
- Selection of materials : use of "ecological" materials : wood, natural stone, natural tiles.
- Minimization of environmental annoyances : air pollution, water and soil pollution, noise, odours.

Building simulation was used to answer the following questions:

- comparison of envelope variants
- assessment of the overheating risk

Comparison of envelope variants by Building simulation

The basic criteria to verify was related to the usability of the testing infrastructure. The criteria as defined as the “time required to reach the prescribed experimental conditions (typically a 10°C temperature) in the test room whatever the period of the year. Therefore, the a priori targeted variant (wooden construction) was compared to more classical solutions.

Three variants were compared using a detailed thermal simulation program ("TRNSYS")

- "classical" envelope : concrete + brick masonry
- "hybrid" envelope : concrete + wooden covering
- "ecological" envelope : wooden structure + wooden covering

TRNSYS is a well-known simulation program developed following a modular structure (each component model is encapsulated as a “Type”). The multizone building component (Type 56) has been the object an extensive validation work (Judkoff, 1993) in similar conditions as those encountered in this project.

For the simulation of the building, two typical meteorological periods were considered :

- a winter period (January month)
- a summer period (August month)

The main problem was to simulate conditions close to that of the future operation of the building, which is not classical at all. Consequently, for each period, a standardized "numerical experiment" was defined, made of 2 weeks with the following distribution :

¹ www.fluent.com

- a "numerical" pre-conditioning period of 10 days
- an "experimentation" period of 2 days : this includes climate control and, eventually, fog production
- a relaxation period of two days.

The main component of the simulation is of course the building which is modeled by Type 56. This model is able to calculate, for a building divided into an arbitrary number of zones, the evolution of both the temperature and the psychrometric variables in each zone. Therefore, the following information has to be provided to the model :

- the zoning strategy; 5 zones were defined : test room, buffer zone, office, technical area and attic
- the walls composition
- the construction materials
- the infiltration rate in each zone
- the ventilation rate (with the associated temperature and relative humidity) in each zone
- the heating and cooling power injected in each zone (associated to an "idealized" control)
- the internal gain, expressed in terms of thermal power (both convective and radiative) and moisture (water vapour).

In this application, the transfer of moisture through the walls was not taken into account because of the relatively long time constant associated to these physical processes.

As only the thermal behaviour of the building envelope was of interest here, a very simple control strategy was elaborated, defined as such : during the experimentation period, the temperature in the buffer zone and in the climate chamber is maintained to a given temperature.

Table 1 shows the time required to reach the prescribed conditions in the chamber in function of the type of construction and period of the year.

	Winter	Summer
Wooden construction	22 h	22 h
Hybrid construction	28 h	28 h
Classical construction	28 h	28 h

Table 1 : Thermal pre-conditioning period of the reduced scale fog chamber

The table clearly shows that the constitution of the external envelope has not a strong influence on the pre-conditioning time of the fog chamber. The "all-wood" construction offers less thermal inertia and consequently allows a quicker preparation of the test room.

SIMULATION ASSISTED CONTROL DESIGN

Simulation was also used to assist the design of the control system . Therefore, a building simulation approach was developed within the ESP-r system. The main question to be solved was how to control the test room and its surrounding environment (the buffer space) in order to maintain the adequate conditions for fog production.

The applicability of ESP-r for the simulation of a fog chamber was assessed by a validation exercise performed on the data from an existing facility, similar in principle to the one concerned by this project.

The outcome was that ESP-r's predictions of air and surface temperatures within the fog chamber were generally within 0.5°C of the measured data (Figure 3).

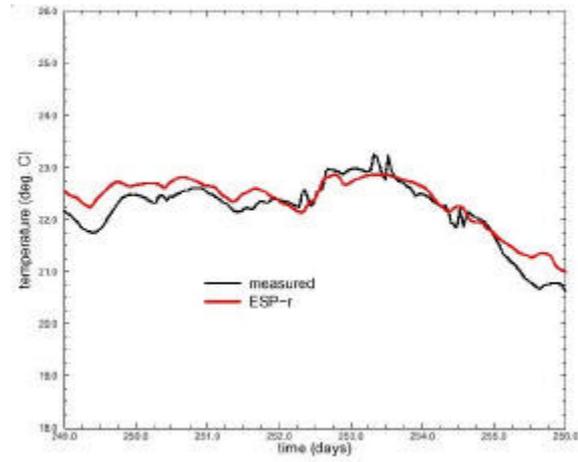


Fig. 3: Comparison of ESP-r outputs and measurements for the simulation of a fog chamber

In a first approach and based upon a building simulation model implemented within the ESP-r program, several control scenarii were examined:

- direct control of temperature and humidity inside the test room
- direct cooling of the test room to its dew point
- use of underfloor heating/cooling to maintain stable temperatures
- indirect control of test room temperatures through conditioning of the surrounding buffer space

The effectiveness of each control method was examined in relation to the stability conditions

achieved and energy required to maintain control. Based upon these criteria, it was found that:

- direct control was the most effective means of achieving stable conditions with a low expenditure of energy (fig. 4)
- dew point control delivered high humidity levels at low expenditure of energy; however, due to the nature of this control method temperatures varied slightly during control periods and also varied between different tests (fig. 5).
- Underfloor heating/cooling gave poor control of conditions, with a greater expenditure of energy and can be discounted as a viable means of controlling conditions (fig. 6)
- Indirect control resulted in unstable test chamber conditions (fig. 7).

Additional simulations were conducted with the aim of examining the performance of the direct control methods in more realistic conditions and improving the performance of the indirect control of the buffer space.

Firstly, the impact of heightened humidity levels on the dew point controller was examined: these simulations revealed that increased humidity levels drastically decreased the energy requirements for dew point control, however the problem of variations in temperature between and during tests remained.

Secondly, more realistic control algorithms were employed with the direct temperature and relative humidity controller:

- these simulations indicated that with a more realistic PID controller model, the performance deteriorates slightly; particularly at the beginning of the control period. However, allowing an adequate test start up period should mitigate the problem.
- Finally, attempts were made to improve the performance of the indirect controller, through a change of control strategy and use of PID control, however the performance remained unsatisfactory, with high energy consumption and poor temperature control.

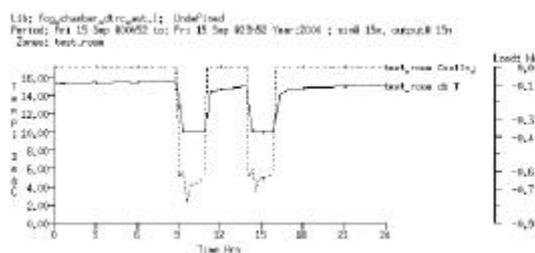


Fig. 4: Direct control of the test chamber

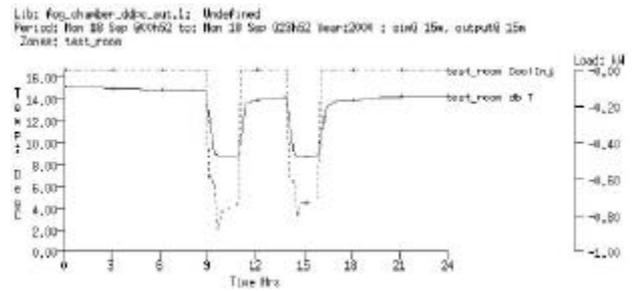


Fig. 5: Dew point control of the test chamber

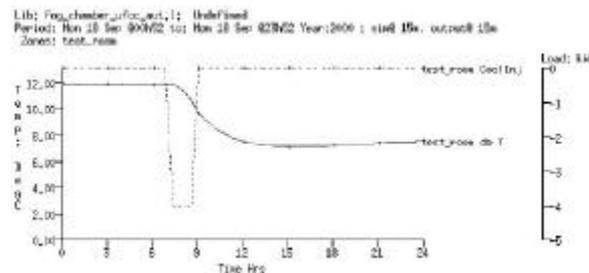


Fig. 6: Underfloor heating/cooling control of the test chamber

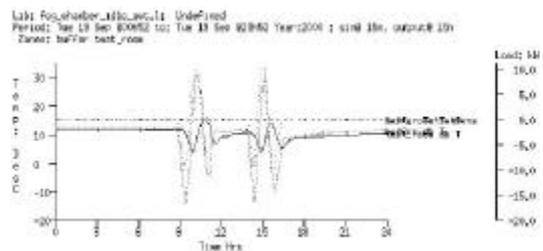


Fig. 7: Indirect heating/cooling control of test chamber

SIMULATION ASSISTED DESIGN OF THE CHAMBER

The final stage of the use of simulation in this project concerned the specific problem associated with the design and operation of the chamber

Design of the chamber

Concerning the design of the chamber, the main problem to solve was the selection of the material for the construction of the envelope of the chamber.

The constructions looked at were:

- 8mm plate glass
- 8mm plexiglass
- existing external wall timber-insulation-timber
- 1mm steel - insulation - 1mm steel
- 100mm block

A summer day and winter day simulation was run for each construction type and the performance assessed based on air and surface temperatures and plant energy consumption. The simulations are run using a characteristic North-European climate.

The simulations consisted of holding the air inside each test room to a fixed value between 07:00 and 13:00 using a coupled HVAC system, underfloor heating and heating of the buffer space. Lighting is assumed to be on between 11:00 and 12:00, with 500W of gains entering the test chamber.

The summer set point is 15°C while the winter set point is 10°C (the HVAC system's low flow rate causes it to struggle in reaching 10°C in summer). The flow rate from the HVAC system is 350m³/h, while the floor has a heating/cooling capacity of 4.5kW.

WINTER SIMULATION

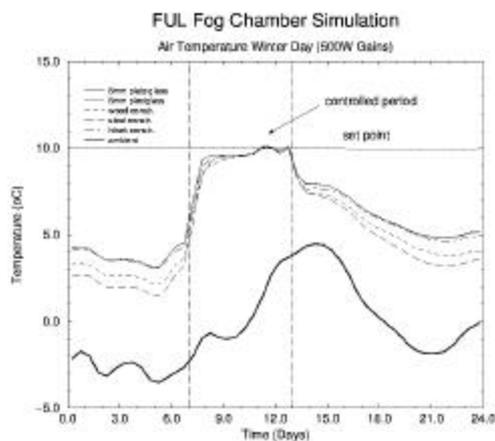


Figure 8: evolution of chamber air temperature in typical winter conditions

Figure 8 shows the variation in air temperature during the winter day simulation. The immediate point to note is that during the controlled period, the performance of all the constructions is very similar. The test room air temperatures associated with the plate glass and plexiglass constructions (as expected) respond most rapidly to heat input. During unheated periods the air temperature associated with the insulated steel and wooden constructions are 1-2°C lower than the uninsulated homogeneous, glass, plexiglass and block constructions. Radiant temperature lags behind air temperature, with the plate glass and plexiglass constructions performing slightly better than the other three, in that they respond more rapidly to heating action.

The mean radiant temperature of the steel construction is as much as 4°C less than the air temperature at the beginning of the heating period. The glass and plexiglass constructions are around 2°C less.

All the mean radiant temperature are within +/- 0.5°C of the air temperature by the end of the heating period.

SUMMER SIMULATION

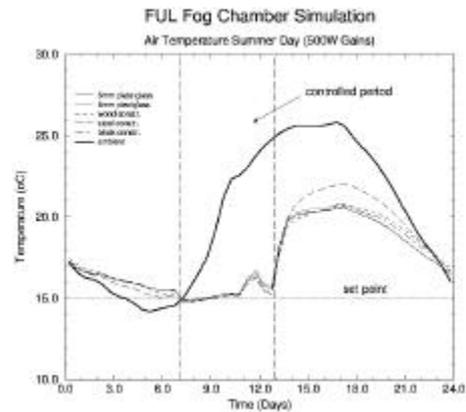


Figure 9: evolution of chamber air temperature in typical summer conditions

Figure 9 shows the variation in air temperature during the summer day. Again, it is immediately apparent that the air temperatures, during the controlled period, are very similar for all the constructions. Looking more closely at the period of internal gains 11:00 - 12:00 it was hoped that the more massive block construction would "damp out" temperature fluctuations. This is not the case: the damping effect is very slight, with a 0.5°C reduction in peak temperature during gains between the block construction and the light plexiglass construction. Outside the controlled period the air temperature associated with the steel construction is 1-2°C higher than the other constructions, this requires some further investigation. The performance of the constructions is reversed compared to the winter case, with the glass and plexiglass mean radiant temperatures being up to 1°C warmer than the air temperature during the controlled period. The wooden and steel insulated structures perform slightly better, with their radiant temperatures being the closest to the air temperature. Again note the curious behaviour of the steel-insulation construction surface temperature outside the controlled period. The analysis of heating and cooling energy consumption was inconclusive as there was very little difference between the energy consumption associated with each construction.

*The simulations indicate that there is very little difference in terms of environmental performance between the 5 constructions examined.

*It should be noted that the floor construction remained the same for all 5 constructions and this will tend to be the dominating factor in terms of thermal mass within the test room.

*The plate glass and plexiglass constructions performed well in winter heating mode but less well in summer cooling mode.

*The reverse was true for the insulated wood and steel constructions.

*Overall in terms of test chamber performance the wall material has little effect.

*Selection of materials should therefore be done based on other factors e.g.

internal reflections with glass and plexiglass constructions during experiments, effects of moisture absorption in wood and block constructions, requirements for visibility of experiment (glass and plexiglass, etc.).

Operation of the chamber

A simulation based approach of the operation of the chamber was carried out with two main purposes:

- to analyze with a higher level of details the control of the internal climate in the test room. Issues related to temperature distribution and thermal stratification and heterogeneity in the chamber were tackled
- to analyze the behaviour of the water droplets injected in the test room to produce artificial fog using a particle model or a droplets diffusion and stability model.

Both objectives were addressed by using CFD models (ESP-r and FLUENT). The applicability of ESP-r for the detailed simulation of the internal climate in rooms has been assessed by several validation studies a review of which can be found in (Strachan, 2000).

Detailed thermal analysis

The detailed thermal analysis of the chamber was performed using the ESP-r system. Using a CFD model integrated within ESP-r and connected to a building simulation model, the effect of the HVAC action on temperature homogeneity in the test room can be illustrated. Figures 10 and 11 show internal conditions during a period when the HVAC system is supplying cold air into the space to counteract the effect of a gain. In this case, there is a variation on temperature across the internal space of approximately 2.5°C

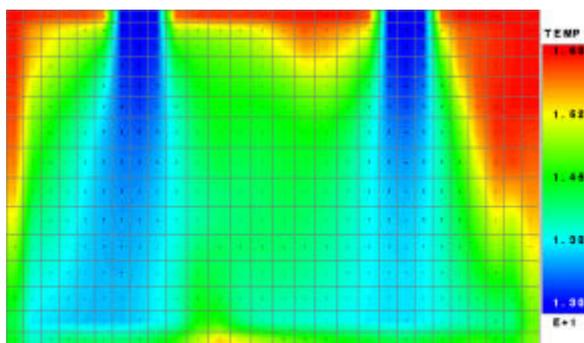


Fig. 10: Test room and air temperature distribution, summer, high gains, frontal view

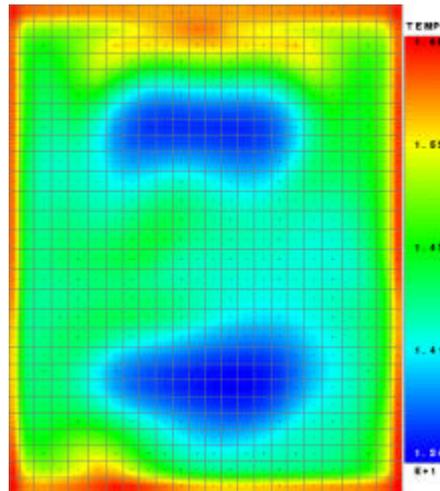


Fig. 11: Test room and air temperature distribution, summer, high gains, plan view

From the output of these calculations, the control strategy used with the HVAC model was changed: the test space is cooled or heated to the set point temperature and the incoming air temperature is fixed to that point.

Modelling of droplets spray

For this part of the work, two steps using two different tools were carried out:

- use of an existing particle tracking algorithm included within the FLUENT CFD program
- development and integration of a droplet model into ESP-r

For the first step, a limited number of simulations were conducted using the FLUENT CFD software. The aim of these simulations was to examine the effectiveness of a particle tracking algorithm in the modelling of small droplet sprays.

A small model of the interior of the proposed FUL chamber was developed. As the test chamber is rectangular and the nozzles are evenly spaced it is possible to model only $\frac{1}{4}$ of the chamber and assume two vertical symmetry planes: the droplet distribution in the other quadrants of the test chamber should be near-identical. 4 or 16 evenly spaced nozzles were placed inside the chamber slightly below the ceiling level. A flow of air is also introduced into the chamber at ceiling level equivalent to 350 m³/h.

Three cases were modelled using the CFD:

- a base case with the nozzles spraying down vertically into the zone and no ventilation
- the nozzles spraying vertically into the zone with ventilation
- the nozzles spraying horizontally into the zone with the same ventilation rate

The behaviour of the air inside the test chamber is modelled through discretisation of the space into small volumes. Equations of continuity, momentum and energy (based upon Navier-Stokes equations) are then solved for each volume.

The behaviour of the fog particles is calculated using a combined mechanical-statistical approach. A representative sample of diameter $3\ \mu\text{m}$ are released from each nozzle and tracked through the fog chamber. The basic equations of motion of the particles are based upon simple Newtonian mechanics, however the influence of turbulence on their paths is accounted for statistically. If turbulence was not incorporated into the model, the paths of the particles would be a projectile-like curve through the fog chamber and the diffusion of the particles would be less than in reality. Mass and momentum exchange between the particles and the surrounding air is calculated in the model.

There are some limitations and assumptions made with the particle model and these should be kept in mind when considering the results:

- the particles modelled are all of size $3\ \mu\text{m}$
- it was assumed that the cone formed by the spray was 30°
- only a limited number of particles are tracked
- droplet collisions are not modelled
- droplets reaching the surface are assumed to adhere to the wall and form a liquid film

Fig. 12 shows an example of results where the droplets form dense columns under the nozzles, in the case where ventilation is introduced in the room

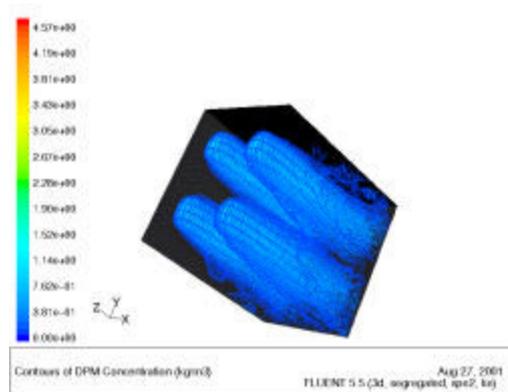


Fig. 12: CFD simulation of droplets diffusion modelling using the FLUENT program: ventilated room case

As a first conclusion, it appears that introducing a ventilation air flow into the test room has a detrimental effect on the homogeneity of droplet density distribution. The action of the airflow seems to impede the diffusion of droplets

In a second step, The ESP-r CFD model has been enhanced to allow modeling of droplet sprays through the addition of

- a droplet transport calculation;
- a droplet/air mixture density calculation;
- droplet sources and sinks (nozzles/droplet adhesion);
- droplet to air momentum exchange.

The output from the ESP-r CFD model's air temperature and velocity predictions is compared to output from the FLUENT CFD tool and references are made to other validation work using experimental data. ESP-r's predictions show good agreement with FLUENT. ESP-r's thermal model has been shown to give good agreement with other experimental data. ESP-r and FLUENT also show agreement on droplet distribution characteristics under different ventilation characteristics, however there are some problems relating to calculation of droplet concentration.

The enhanced BSim-CFD model of the FUL test chamber was used in a series of sensitivity simulations relating to nozzle spacing and ventilation characteristics. The main conclusions from these simulations were as follows:

Simulations with the ESP-r model reinforce the findings from previous CFD simulations using FLUENT that the use of ventilation during fog tests is detrimental to homogeneity droplet distribution. (fig. 13)

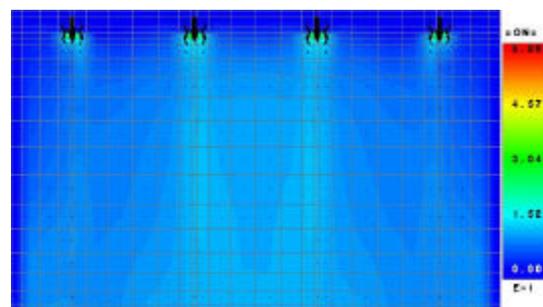


Fig. 13: CFD simulation with ventilation showing heterogeneity of droplets distribution

- The proposed 16 nozzle configuration appears to give a reasonably even droplet distribution when there is no ventilation.
- A test with 4 nozzles revealed a deterioration in the homogeneity of droplet distribution.

A droplet stability model was finally introduced in the CFD model. For each cell in the CFD domain ESP-r can calculate a critical volume V_N and volume stability limit V_C . The formulas to do this are as follows:

$$V_N = \frac{4}{3} \frac{\pi r_d^3 \rho_d}{\rho_m V_C}$$

and

$$V_C = \frac{\pi d_d^4 \rho_d T R_s}{4 a P_{2d}}$$

where

$$a = \frac{3.15 \times 10^{-5}}{273.15}, \quad P_{2d} = P_{2\infty} \left[1 + \frac{2d_d}{a} \right], \quad P_{2\infty} = 6108 e^{\left(\frac{17.2T}{T+2373} \right)}$$

The value of V_N/V_C is computed and stored; if $V_N/V_C > 1$ i.e. $V_N > V_C$, then the droplets in this part of the CFD domain can be considered unstable. Using the CFD tool it is possible to produce plots of droplet stability as shown for instance by Figure 14a.

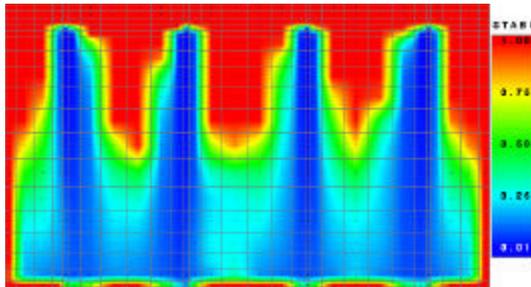


Fig. 14a: droplets stability plot for droplets size 3 microns.



Fig. 14b: Observation of droplets concentration in a fog production experiment.

Fig. 14b shows the visual effect as observed in a real fog production experiment. This picture demonstrate

the qualitatively good agreement between the model and the observations. The model used had 16 spray nozzles located in a regular grid within the test chamber, all three figures show a vertical cross section through the chamber.

CONCLUSION

As a conclusion, the advantages and disadvantages of “building” and “CFD” simulation tools in the design process of a building can be identified:

Building simulation:

- Short simulation time
- Validation results available
- Approximate results but sufficient for the pursued objective

CFD:

- Long computation time
- No dynamics
- Appraisal of small spatial variations

ACKNOWLEDGMENT

The work related in this paper was funded by the European Commission within the project “Improvement of transport safety by control of fog production in a chamber”, contract n° G6RD-CT-2000-00211.

REFERENCES

- Andre et al. 2002.. Simulation of a non conventional HVAC plant in an experimental building devoted to fog production. Proceedings SSB’2002 Congress Liège.
- Clarke, J.A. 2001. Energy Simulation in Building Design, 2nd Ed. Butterworth-Heinemann, 2001
- Klein et al. 2000. “TRNSYS 15 reference manual”, Solar Energy Laboratory, University of Madison-Wisconsin.
- Lomas, K.J. et al. 1994. Empirical validation of thermal building simulation programs using test room data. IEA Task 12/Annex 21 final report. International Energy Agency
- Strachan , P.A. 2000. ESP-r: Summary of validation studies, ESRU , Glasgow